Introduction to the finite element code LAGAMINE

November 27, 2023

Forewords

This document is meant as a living document, evolving with the questions and needs that arise. Please complete and update it!

List of contributors and reviewers

Frédéric Collin
Anne-Catherine Dieudonné
Matthias Hauth
Jean-Pol Radu
Bertie Rietma
Yasmin Teixeira Trindade
Aoxi Zhang

Contents

Chapter 1

Introduction

1.1 About LAGAMINE

LAGAMINE is a finite element code born at the University of Liege at the end of the seventies. The code has been developed along with innovations and science for the last forty years. The continuous work of researchers around the world — in universities as well as in the industry — has maintained it at the leading edge of technology and research. More specifically, LAGAMINE is able to deal with complex nonlinear constitutive models, multiphysical coupling, strain localisation and multiscale approaches for applications in the fields of environmental geotechnics, engineering geology, reservoir engineering and metal forming.

1.2 About this tutorial

This tutorial is meant to provide first-hand experience with the finite element code LAGAMINE. Although this guide has been written as a beginner tutorial, it is recommended to have some basic knowledge of finite element modelling to understand how the software functions. Instead of listing all the features available in LAGAMINE, which is already available in the DOKUWIKI, this tutorial details selected examples of increasing difficulty to learn how to use the software gradually.

In the following,

- Light grey boxes are used to refer to buttons to click on, e.g. Button
- Text in boxes refers to text or command to be typed using the keyboard, e.g. Text.

1.3 Prerequisites

LAGAMINE runs under Microsoft Windows. Several programmes of the LAGAPROGS interface are written in Fortran, and it is necessary to use the full stop (.) as the decimal symbol. Therefore, in Windows 10, go to *Control Panel*, click *Change date, time, or number formats* and then *Additional Settings*. Under the tabs *Numbers* and *Currency*, select the full stop (.) as the decimal symbol and the comma (,) as the digit grouping symbol. Finally, click on *Apply*. If you do not have full admin rights and are using for example a `\localadmin` account, make sure the decimal symbol of that account is set correctly as well.

You will need a text editor to write some input files and read output files. Although any text editor can be used, it is recommended to download and use the open-source NotePad++ software.

1.4 Installation of LagaProgs

LagaProgs is the name of the user interface for the finite element code Lagamine. It is installed by running the installation file `SetupLagaProgs.exe`. The program LagaProgs is installed by default in the directory `C:/Program Files/LagaProgs`, but another directory can be selected[1]. The other sub-programs used in the LagaProgs environment are automatically installed in the directory `.../LagaProgs/prog`.

You need to execute LagaProgs as an administrator; otherwise, your folders will not appear in `C:/Program Files (x86)/LagaProgs`. This is also necessary to properly run the Debugger (for developers). To do so, right-click on the LagaProgs icon, go to properties, and tick the option *Run this program as administrator* in the compatibility tab.

Some sub-programs might give an error message at launch, indicating that `.dll` files are missing. In particular, the `Msvbvm50.dll` and `vb5fr.dll` files might be missing. In that case, these files must be downloaded from a reliable source and then placed into the folder `C:/Windows/SysWoW64`.

[1]Note that you need to possess administrator rights on the directory where Lagamine is installed. Ideally, you should also possess administrator on the `C:/` drive as you might need to add or modify some files.

Chapter 2

First step into the LAGAPROGS interface

As the program LAGAPROGS is executed, a window called *PRESENTATION* appears on the screen (Figure 2.1). The dialog box allows you to create a new username or to select an existing one. For each username, a folder is created in the LAGAPROGS directory `...\LagaProgs\Username\` so that each user can easily find their own projects and files.

Figure 2.1 *PRESENTATION* window of LAGAPROGS.

The main window of LAGAPROGS is shown in Figure 2.2. LAGAPROGS is actually a platform to quickly manage and pilot all the files and sub-programs that are necessary to carry out a finite element simulation.

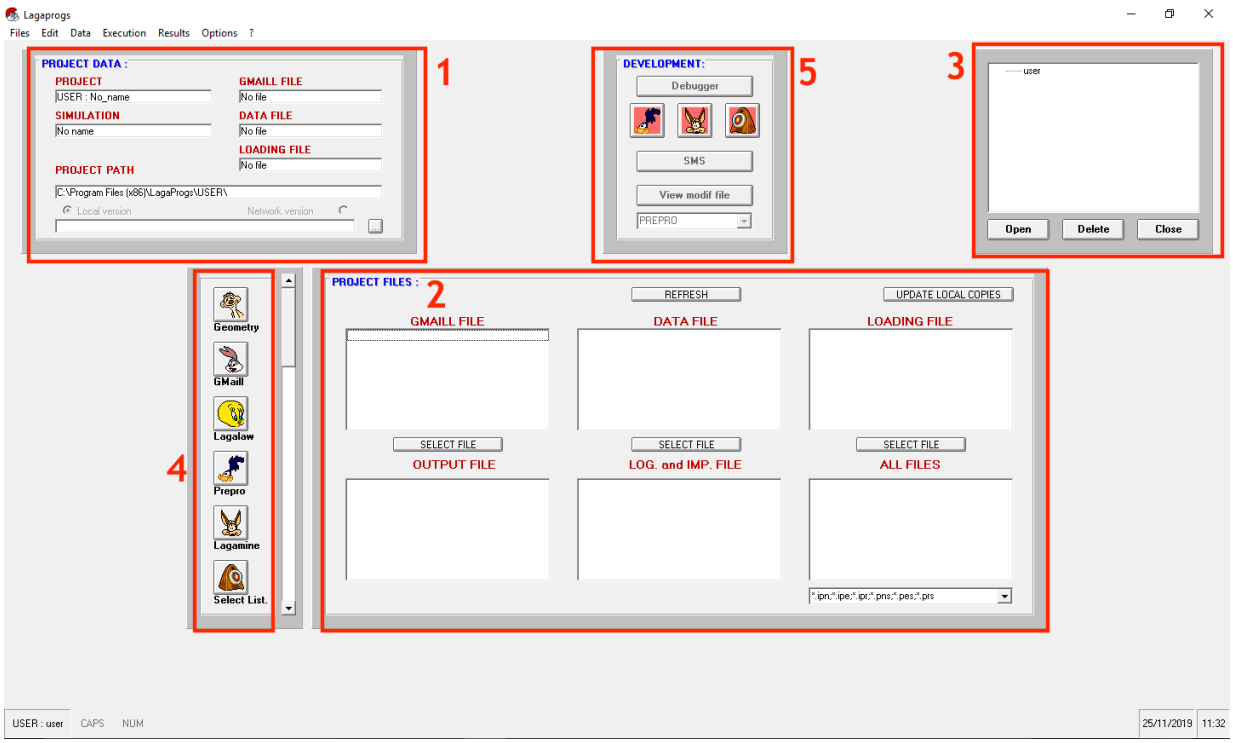

Figure 2.2 Main window of LAGAPROGS.

This platform consists of five components:

1. *PROJECT DATA*: this first part (on the top left corner) gathers information about the ongoing project, such as the name of the project, the name of the simulation and the names of the working files (*GMAILL FILE*, *DATA FILE* and *LOADING FILE*). These files are used by the different sub-programs, so you need to make sure your are working with the right files for each simulation. You cannot directly write the name of a file. Instead, you need to select each file manually from *PROJECT FILES*.
2. *PROJECT FILES*: this second part comprises all the files of the current simulation. You can double-click on the name of the file to open it as a notebook document, and modify its content yourself. To select a file for the part *PROJECT DATA*, click on the file and then on the button SELECT FILE.
3. The third part (on the top right corner) shows a tree view of the projects and the simulations defined by the user.
4. The fourth part (on the bottom left) displays shortcuts to quickly execute the different sub-programs.
5. The fifth part (on the upper middle) is only useful for developers and is not accessible in the student version.

The toolbar provides additional functionalities, but we will come to that later. The ? tab of the toolbar gives access to the manuals of LAGAMINE, explaining how the *GMAILL*, *DATA* and *LOADING* files are organized and filled by the sub-programs. If needed, updated information regarding these files is available in the DOKUWIKI.

In section 1.3, it was suggested to use NotePad++ as text editor. Therefore, click on Options on the top of the screen, and then on Programs path. A new window appears (Figure 2.3). This window is used to select the different sub-programs of the LAGAPROGS platform (see Chapter 3). Click 3 times on NEXT until the link to the text editor appears. Click on ... to select the path to NotePad++ (Figure 2.4). Click on OK to finish and close the window.

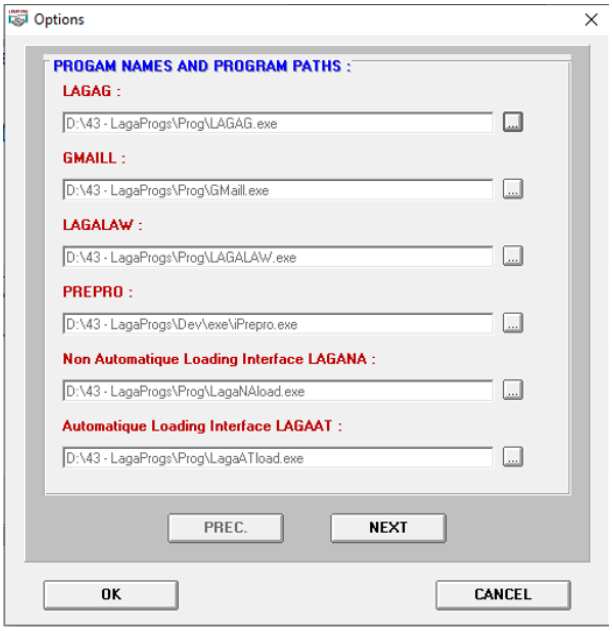

Figure 2.3 *Options* window.

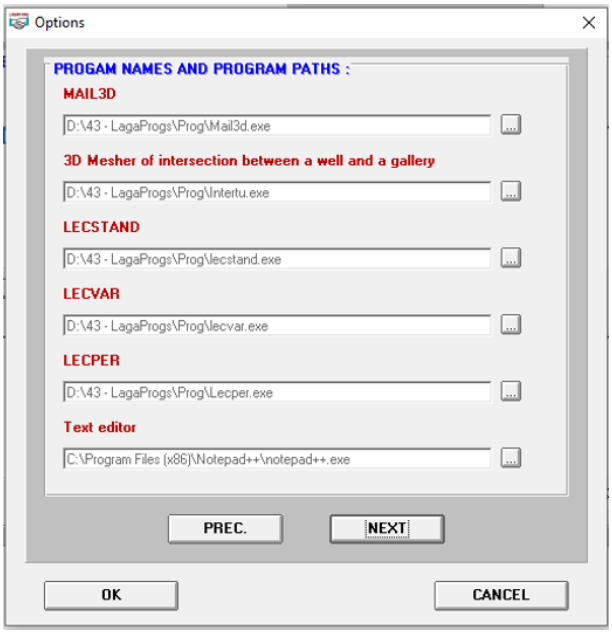

Figure 2.4 *Options* window: text editor.

Chapter 3

Main sub-programs and files

The platform LagaProgs gathers different sub-programs, which are necessary to perform and analyse a finite element simulation. These programs, including their input and output files, are briefly described hereafter. The differently formatted files follow a specific format and structure, which are described in Appendix A.

3.1 Lagag

The first step of a finite element simulation is the definition of the geometry. This is done using Lagag, which is executed by clicking on Geometry in LagaProgs.

Icon	
Purpose	Define the geometry (including discretization, element types and law numbers) and boundary conditions of the problem.
Output file	`NAMDAT.GML`

3.2 GMaill

Once the geometry is defined, the finite element mesh can be generated. This is done by running GMaill.

Icon	
Purpose	Generate the mesh and create the incomplete `NAMDAT.LAG` file. The `NAMDAT.LAG` file will be completed later to contain the information on the geometry, initial conditions, boundary conditions and material parameters.
Input file	`NAMDAT.GML`
Output file	`NAMDAT.LAG`: (incomplete) data for Prepro

3.3 LAGALAW

The next step consists in selecting material models and defining the corresponding parameters. This can be done by either modifying the NAMDAT.LAG manually, or using LAGALAW (beginners). Advanced users will generally prefer to directly modify the NAMDAT.LAG file manually as not all constitutive models are available through LAGALAW.

Icon	
Purpose	Select the material models and define the corresponding parameters. Define the initial pore pressures and temperature, in the case of a coupled analysis. Define the value of gravity (if taken into account). Define the parameters of the re-numbering strategy.
Input file	NAMDAT.LAG: (incomplete) data for PREPRO
Output file	NAMDAT.LAG: (complete) data for PREPRO

3.4 PREPRO

Before we can run the finite element simulation, the data file should be pre-processed. This is done by executing PREPRO.

Icon	
Purpose	Pre-processor
Input file	NAMDAT.LAG
Output file	NAMDAT.OUT: listing of data
	NAMDAT.FO1: permanent data (not formatted)
	NAMDAT.FO2: variables (not formatted)
	NAMDAT.DON: necessary data for SELECT and DESFIN

3.5 LAGAMINE

The simulation can now be solved using the finite element code and program LAGAMINE. Note that, in order to run a simulation, a loading file (NAMEX.DAT) should be defined. Advanced users will generally create and fill in this file manually. However, the sub-programs LAGANALOAD (manual strategy) and LAGAATLOAD (automatic strategy), accessible via the LAGAPROGS platform, enable to fill in this loading file.

Icon	
Purpose	Run the finite element simulation.
Input files	`NAMEX.DAT`: loading file
	`NAMDAT.FO1`
	`NAMDAT.FO2` (or `NAMDAT.FO3` if specified in `NAMEX.DAT`)
	Optional:
	`NAMDAT.PRI`: output data to be printed
	`.SWI`: activation/deactivation of some elements during the simulation
	`NAMDAT.LOA`, `NAMDAT.DEP` and `NAMDAT.LIC`: definition of the loading path
Output files	`NAMEX.OUT`: listing of LAGAMINE's execution
	`NAMEX.LOG`: convergence of the execution
	`NAMDAT.FO3`: variables (not formatted file)
	`NAMDAT.OTO`: variables (not formatted file)
	Optional:
	`NAMDAT.IPN`: printing file for nodal variables
	`NAMDAT.IPE`: printing file for element variables
	`NAMDAT.IPR`: printing file for reactions

In this document, `NAMDAT` and `NAMEX` can be replaced by any names. However, different names should be used since LAGAMINE generates two different output files. In practice, it is recommended to use `NAMDATex` for `NAMEX`, i.e. the same name as the `.LAG` file, followed by `ex`.

3.6 SELECT

Once the problem is solved, SELECT is used to select the results to be printed or represented.

Icon	
Purpose	Select the results to be printed or represented.
Input file	`NAMDAT.DON` and `NAMDAT.FO1`
	`NAMDAT.FO2`, `NAMDAT.FO3` or `NAMDAT.OTO`
Output file	`NAMDAT.RES`: selection of time steps for DESFIN (not formatted file)
	`NAMDAT.IMP`

3.7 DESFIN

Finally, the results can be represented using DESFIN.

Icon	
Purpose	Representation of the results.
Input file	`NAMDAT.DON` and `NAMDAT.RES`
Output file	`NAMDAT.PLO`: graphical file
	`NAMDAT.ASC`

Chapter 4

Oedometer test

This first exercise is meant to gain a hands-on experience with the LAGRAPROGS interface and the (main) associated sub-programs. It includes all basic steps of a finite element simulation, from the creation of a project to the representation of the results.

4.1 Description of the problem

The problem studied in this first example consists in the uniaxial compression of a cylindrical soil sample (radius = 30 mm, height = 20 mm). Due to the symmetry of the sample and the symmetry of the loading, the problem is treated as a two-dimensional axisymmetric problem.

4.2 Creation of the project and simulation folders

Once you are logged in LAGAPROGS, go to the toolbar and select Files, and then New in order to create a folder for a new project.

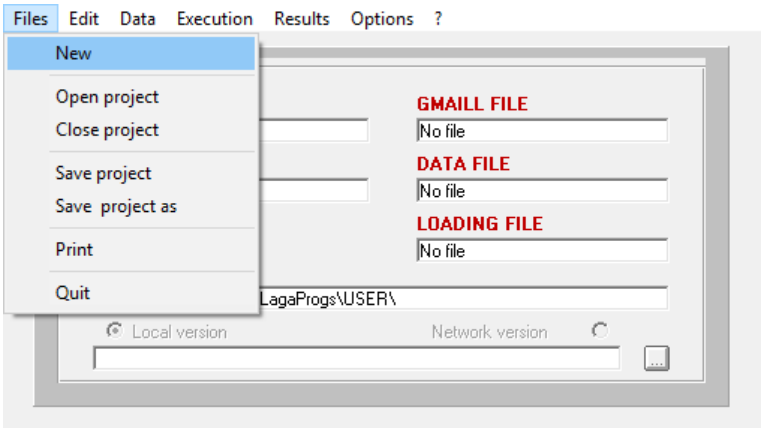

Figure 4.1

Write the name of the project under the tab Project, namely Oedometer test, and click on OK.

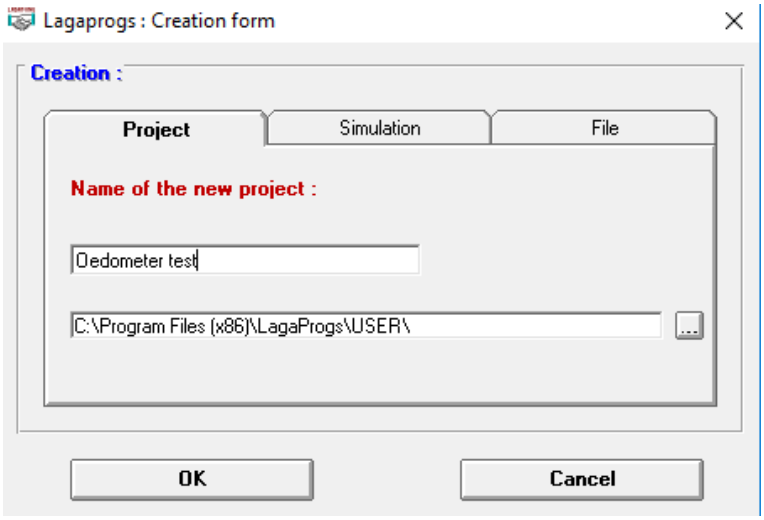

Figure 4.2

Following the same procedure, now create a simulation under the tab Simulation and write the name of the simulation, namely oedo_axisym. Click on OK.

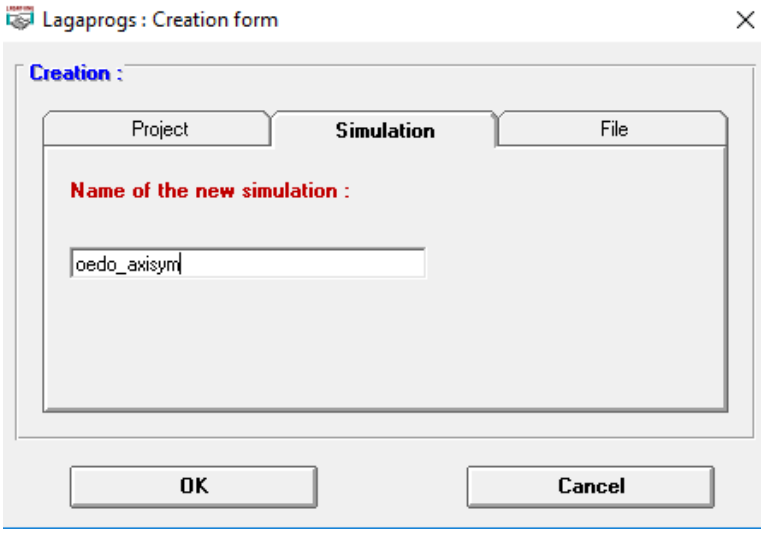

Figure 4.3

The tree view is shown in the upper right corner of the platform LAGAPROGS. The newly created folders are located in the directory `...\LagaProgs\Username\`.

These first operations also modify the file `...\LagaProgs\Username\Username.LST`, which list all the projects (with the corresponding number of simulations) of the user. This file is important as it enables to display the tree view in LAGAPROGS.

4.3 Creation of a .GML file

The .GML file contains all the information on the geometry of the problem. Click on Files then New from the LAGAPROGS toolbar and check the box GMAILL FILE from the tab File.

Write the name of the GMAILL FILE as oedometer, and click on OK.

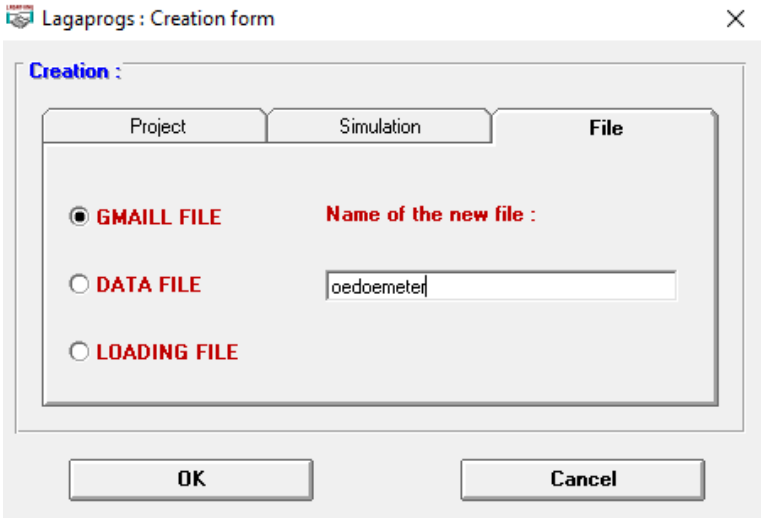

Figure 4.4

An empty .GML file is automatically generated and appears in LAGAPROGS under the section *PROJECT FILES*. Click on the file and then on the button SELECT FILE to assign it as the current working file. It is then also displayed in the section *PROJECT DATA*.

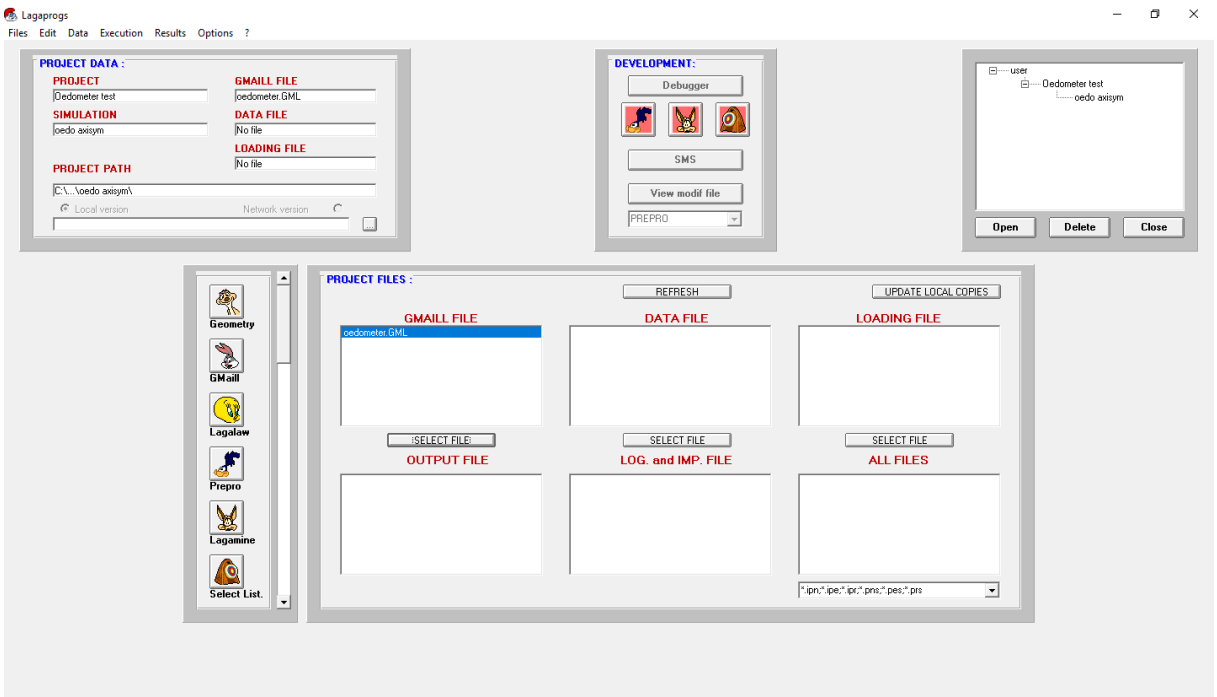

Figure 4.5

4.4 Definition of the type of problem

From the LAGAPROGS toolbar, click on Data and then on Analysis and problem type. A new window pops up where you can select the type of analysis and the type of problem. Our case can be simplified as an axisymmetric problem for a purely 2D mechanical analysis (no hydro-mechanical coupling is considered).

Select 2D MECHANICAL under *ANALYSIS TYPE* and AXISYMMETRIC under *PROBLEM TYPE*.

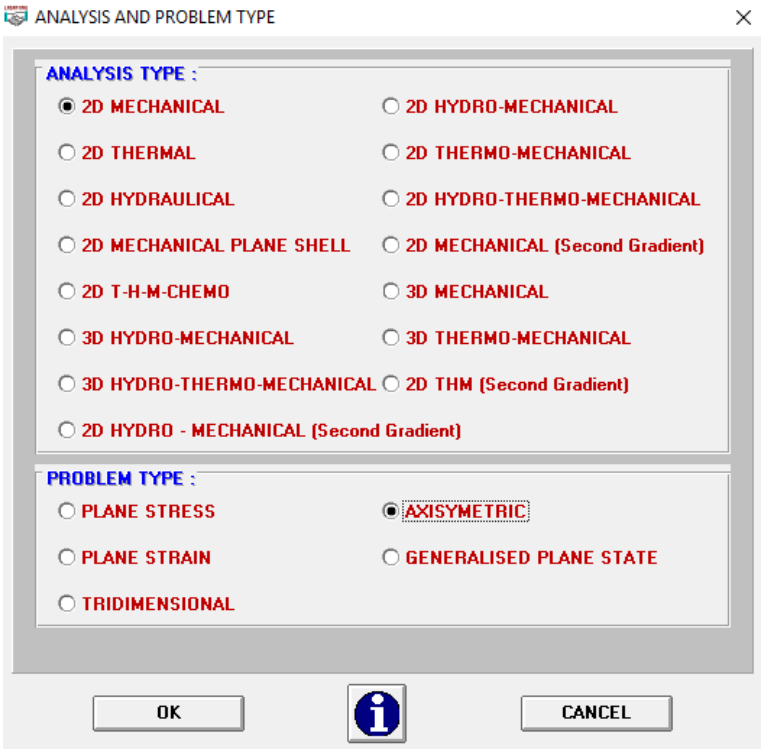

Figure 4.6

Advanced users will generally skip that step, and later modify the second line of the `.LAG` file.

4.5 Definition of the geometry, the finite elements and the boundary conditions

We now need to define the geometry of our problem by modifying the content of the `.GML` file, which is currently empty. The sub-program LAGAG will help us do so with an intuitive graphical interface.

To start LAGAG, click on the tab Data of the toolbar, and then click on Geometric and boundary conditions. Alternatively, one can directly click on the icon *Geometry* on the left of part of LAGAPROGS. Ignore the pop-up message appearing *'The GML file is empty. No data have been saved'* and click on OK. A new LAGAG window should open, and you might need to adjust the size of the window by double-clicking on the top bar.

Figure 4.7

4.5.1 Definition of points

We first need to define the coordinates of all the points of interest of the analysis. For our example, we only need to insert four points: ① (0,0), ② (0,0.02), ③ (0.03,0), and ④ (0.03,0.02) to mark out the limits of the problem.

The international system of units is used in LAGAPROGS, so that lengths are expressed in meters.

- From the toolbar, click on Edit and then on Characteristic point.
- Click the New button at the bottom of the window and enter the desired coordinates of the point.
- Click on the OK button to validate the point (or press Enter).
- Repeat the process for all points.

For an axisymmetric problem like ours, the y-axis is always the axis of symmetry.

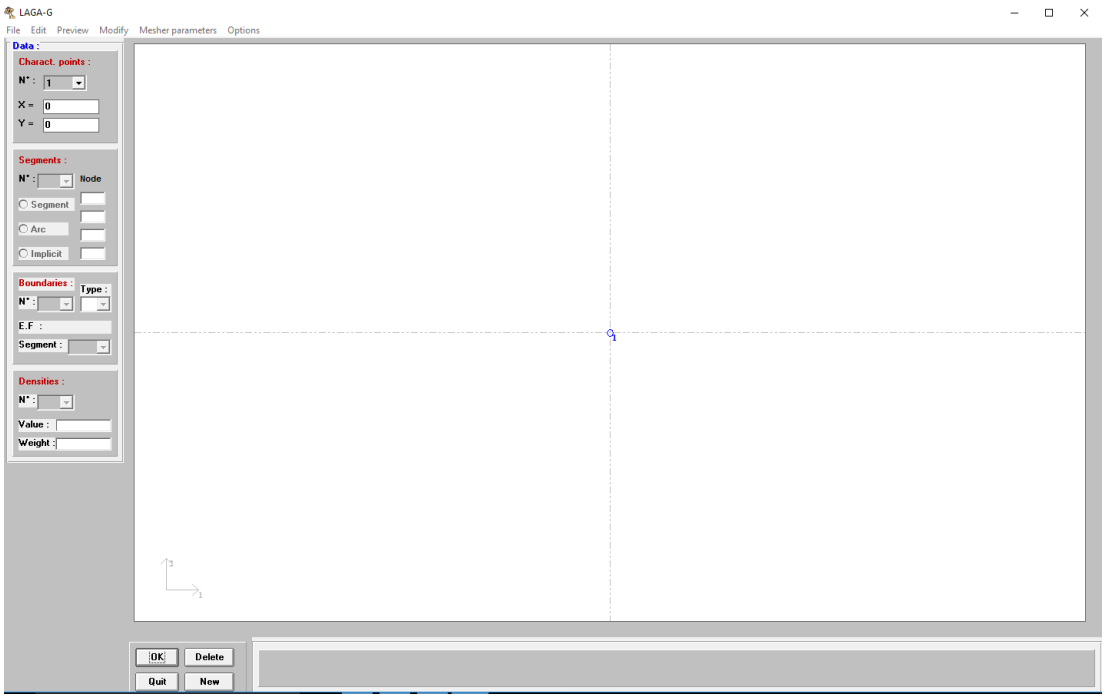

Figure 4.8

It is always possible to modify the coordinates of the points by selecting the corresponding point with the point drop-down menu and changing the values of X and Y. Do not forget to click on the button OK to validate your modification (or just press Enter).

Once all the points are defined, click the Quit button at the bottom left of the screen. It is possible to adjust the scale of drawing area so that all points are visible. From the toolbar, select Preview and then Fit to window.

Figure 4.9

4.5.2 Definition of segments

Before creating segments, select from the toolbar Options and then Division of segment in order to be able to divide later our segments into smaller elements for the mesh. Two types of segments are possible: a segment can either be a straight line between two points or an arc passing by three points (or passing by two points with a given center). For the oedometer test, we only need to link our points with straight lines to form a rectangle.

Click on Edit and then, Segment to define the geometry of the problem.

- Click on the New button to create the first segment.
- Left-click on the first point in the drawing area, and then on the second point.

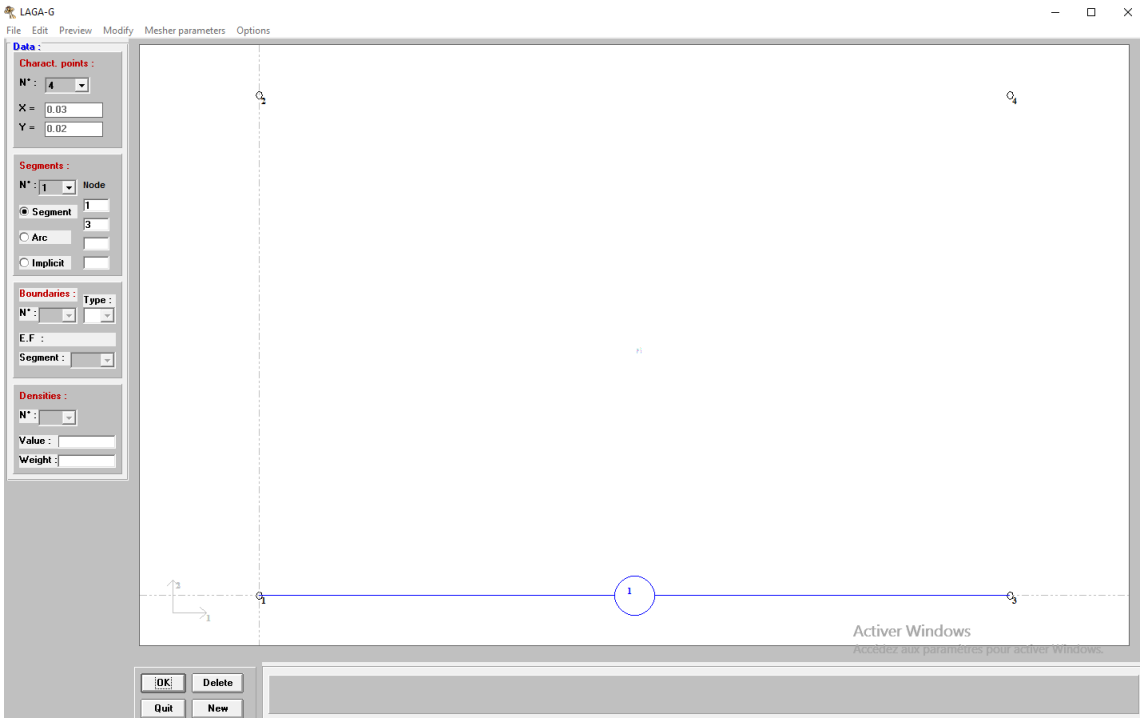

Figure 4.10

- Click on the OK button to validate the segment.
- Enter the number of subdivisions of the segment you have just created. This number will correspond to the number of finite elements on the corresponding border. Choose, for instance, 10 and click on OK.

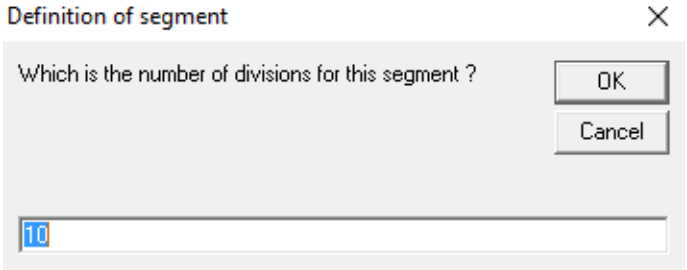

Figure 4.11

- Repeat this process for the other three segments.

- Click Quit , the drawing area should look like following figure.

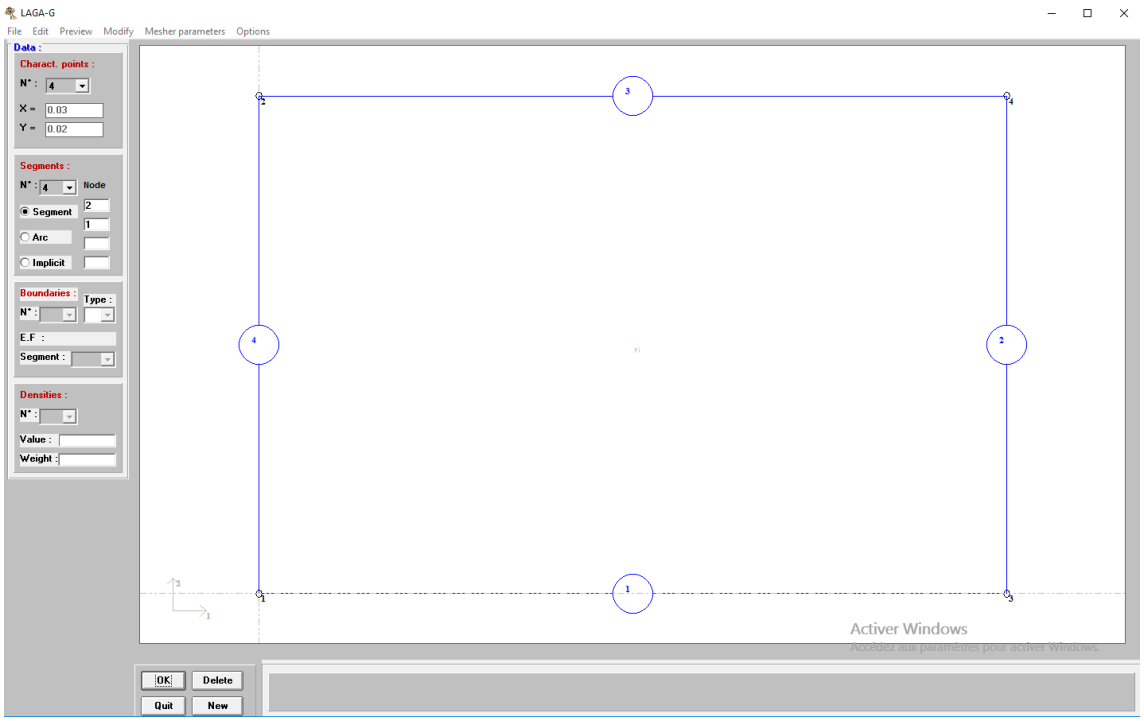

Figure 4.12

4.5.3 Definition of the boundaries

Now that the geometry of the problem has been defined, we need to select the type of mesh we want, assign law numbers to the elements and apply boundary conditions at the contour. To do so, select Edit and then Boundary . The available options are displayed below the drawing area and are assigned with a code number.

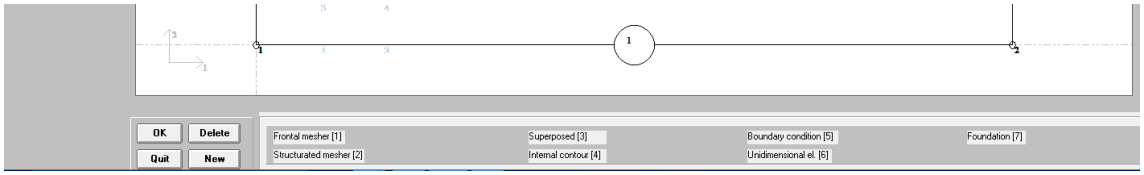

Figure 4.13

Solid elements There is only one type of material (namely the tested soil) and one group of elements forming the mesh for the oedometer simulation. Select New below the drawing area to create a new contour, and choose the desired type of contour on the left, as shown in the figure below. Naturally, the selected segments must form a closed loop.

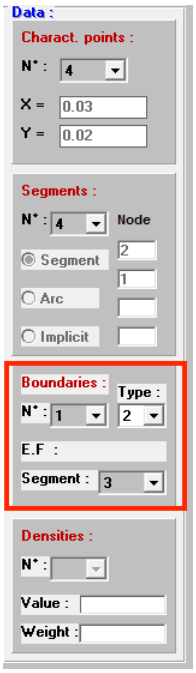

Figure 4.14

Select the *structurated mesher*, i.e. Type = 2, and then click on the four segments to mesh the soil sample.

The four segments should not be selected in any arbitrary order. While any segment can be selected first, the subsequent segments should be selected **counterclockwise**. If the domain is not properly meshed, errors will later arise when executing the sub-program PREPRO, see section 4.11.

One you selected the four segments, click on OK. A new window appears.

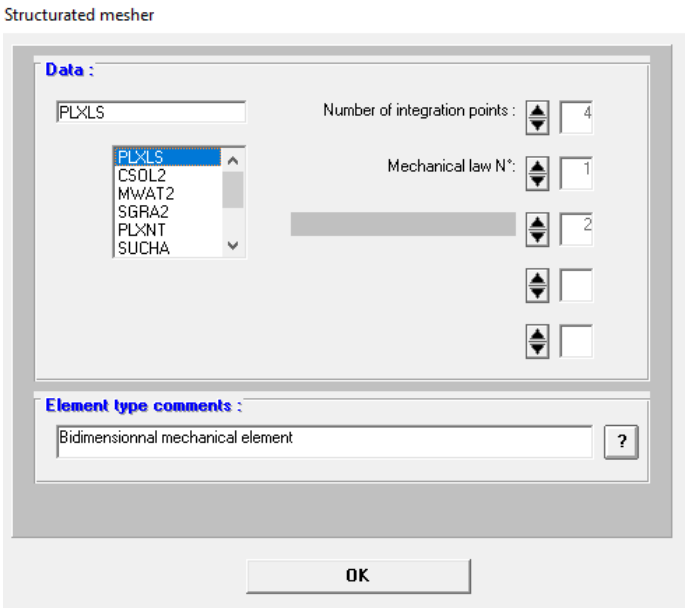

Figure 4.15

This window allows the user to choose the type of elements for the mesher. For further information about, please refer to the LAGAMINE manuals (especially Elmec, Elcou and Elthe).

For our example, we choose the PLXLS elements, which are 2D purely mechanical elements (with 8 nodes and 4 integration points by default). A number for the mechanical law must be assigned as 1. Since our problem is purely mechanical, the second number is dis-activated even if it is still displayed on the screen. Click on OK to close the window.

Boundary conditions We now define the boundary conditions of our problem. In the case of an axisymmetric oedometer test, the boundary conditions are:

- A **linearly distributed force** on top of the sample.
- **Fixed displacements** on the sides and at the bottom of the sample.

Therefore, we need to create 4 boundary conditions as illustrated in Figure 4.16.

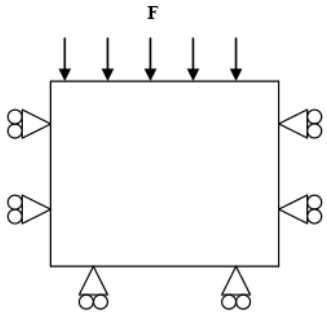

Figure 4.16

To define the **linearly distributed force**, click on New and select a Type = 5 boundary. Select the top segment on which the force will be applied and click on OK.

In LAGAMINE, a force is always applied on the left side of the selected segment for a defined direction (see Figure 4.17).

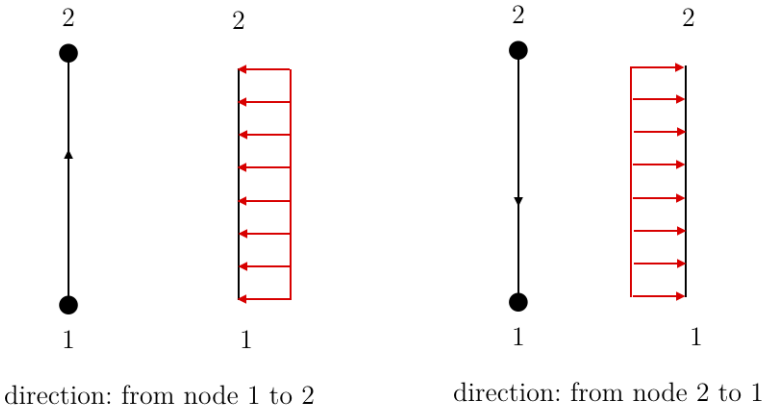

Figure 4.17 (Left) The direction of the segment is node 1 toward node 2, as a result the force is applied on the body to the left of the segment. (Right) Similarly, when the direction is switched, the force is applied to the body on the right of the segment.

A new window pops-up to define the direction of the force (see Figure 4.18). This question allows us to choose the direction of the segment.

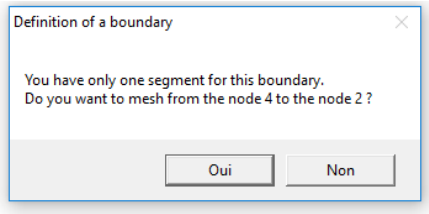

Figure 4.18

In our example (see Figure 4.19), to mesh from node 4 (respectively node 2 in Figure 4.17) to node 2 (respectively node 1 in Figure 4.17), the applied force will be downward. We then answer Oui (yes) to the question (see Figure 4.18).

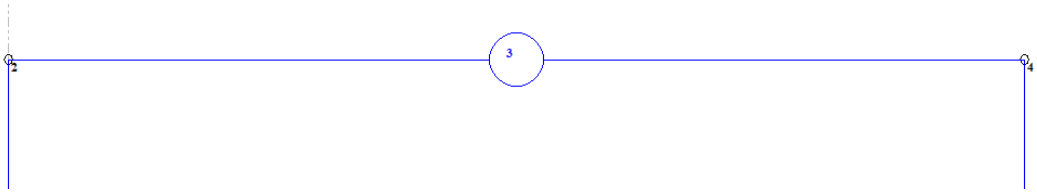

Figure 4.19

A new window appears where the type of element can be chosen for the contour. We choose here LICHA and we assign a new number to this law. The law number 1 was already assigned to the material mechanical behaviour of the sample, therefore this new mechanical law for the linearly distributed force will be defined by the law number 2. Click on OK to close the window.

For the **fixed displacements**, we must now fix the degrees of freedom to prevent any displacement from occurring at the border of the simulation. Select New and Type = 5 boundary condition, click on the bottom segment (Figure 4.20) and then click OK.

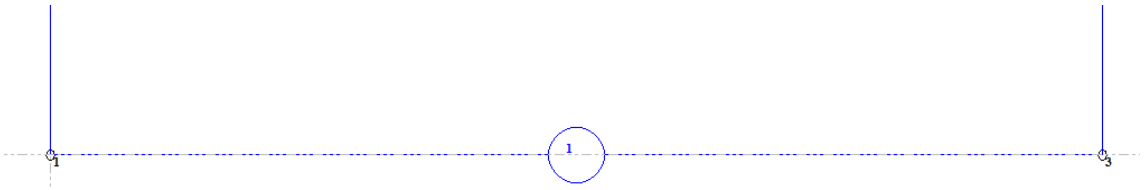

Figure 4.20

A message to choose the direction will pop-up once again (see Figure 4.21). Here, the direction does not matter as we will fix the vertical displacement. Click either on Oui (yes) or Non (no).

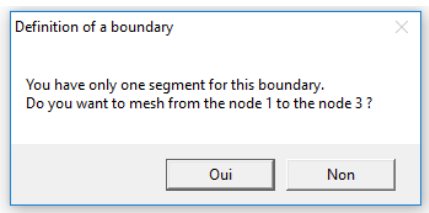

Figure 4.21

Table 4.1 summarizes the definition of the degrees of freedom in LAGAMINE.

DOF	2D	3D
x	1	1
y	2	2
z	-	3
p_w	3	4
p_g	4	5
T	5	6

Table 4.1 Number of the degrees of freedom (DOF) in 2D and 3D analyses.

In our 2D simulation, we want to fix displacement on the Y-axis at the bottom of the sample, so we need to select DOF FIXED and 2 as our number for the degree of freedom (DOF FIXED N°).

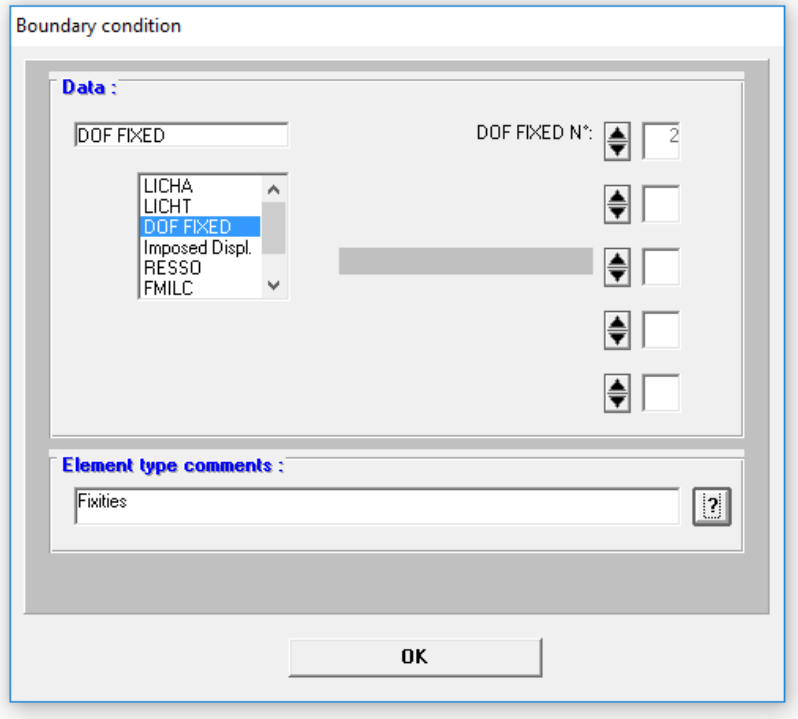

Figure 4.22

We repeat the same process for the two sides of the sample, but we now block the displacements on the X-axis, that is to say the degree of freedom number 1. We then obtain the following figure.

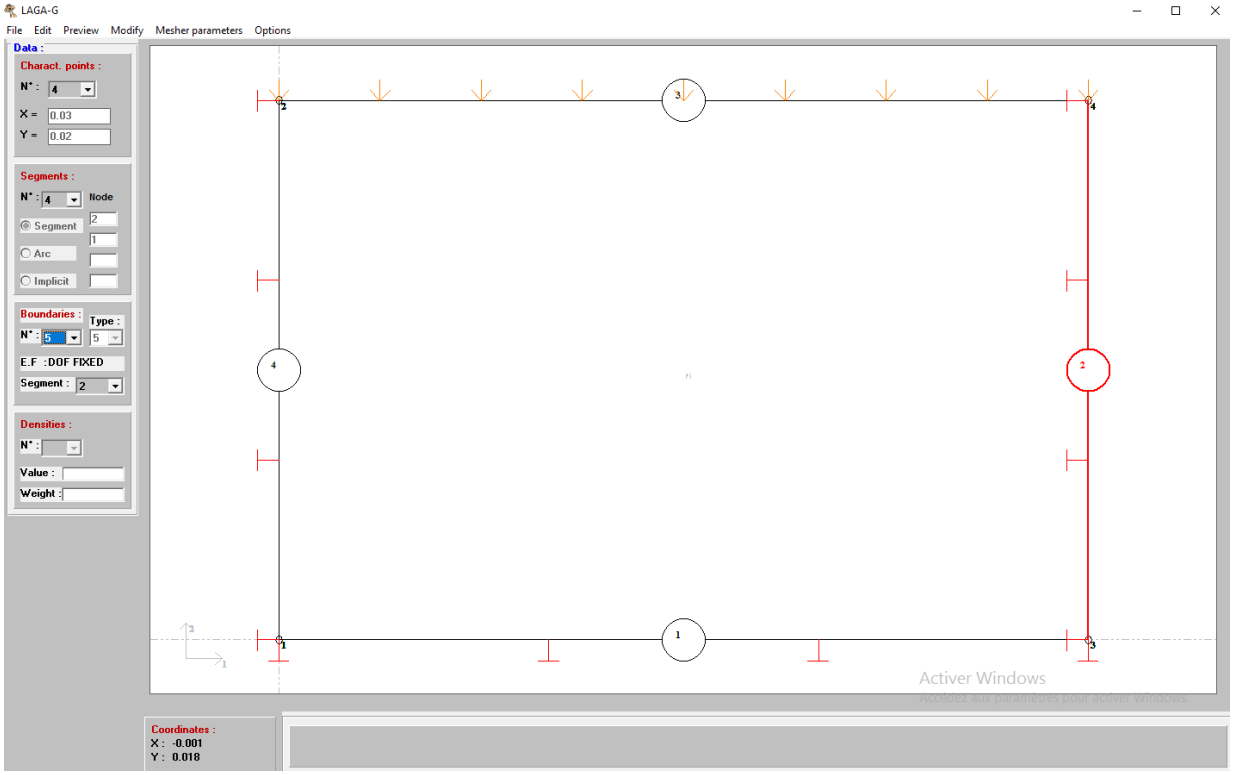

Figure 4.23

Click on Quit to leave the boundary options.

Finally, to save our data, select File and then Save data. A window might appear with a question asking you if you want to replace the existing file. Answer Oui (yes) and leave the LAGAG by clicking on File and then Quit.

Back to LAGAPROGS, if you double-click on the .GML file, we can see it has now been filled.

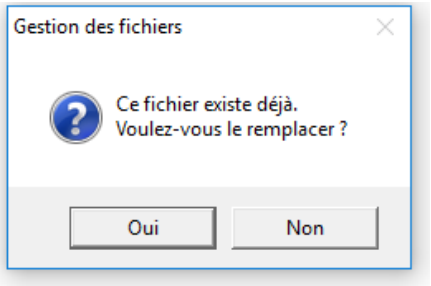

Figure 4.24

4.6 Execution of mesher GMAILL

From the toolbar, select Data and then GMAILL in order to execute the sub-program GMAILL that will build the mesh and create a first version of the data file oedometer .LAG. Alternatively, one can simply click on the GMAILL icon. If the execution has been successful, the following message will appear, click on OK.

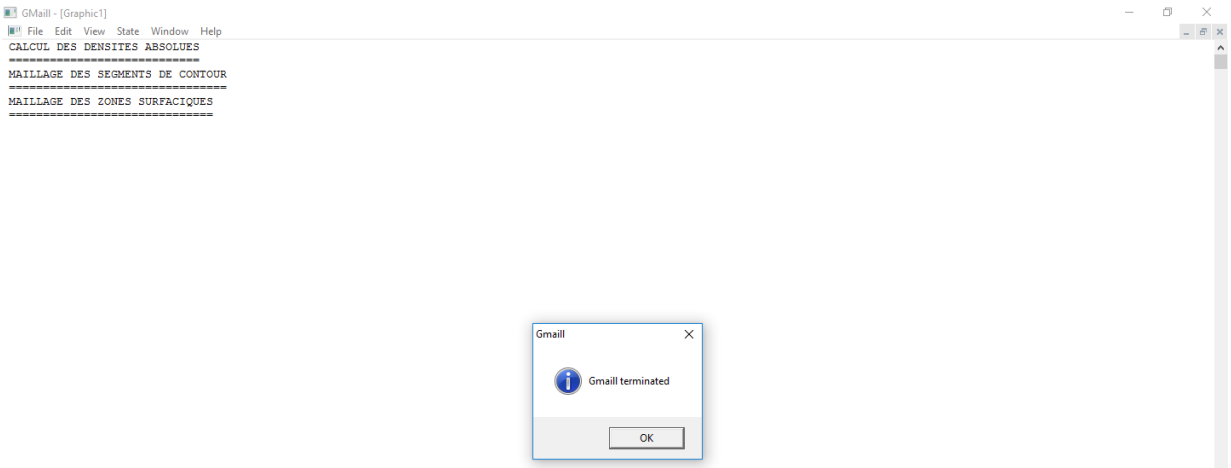

Figure 4.25

Click now on the Refresh button to display the newly generated file `oedometer.LAG`. Click once on this file and then on SELECT FILE below to add this file into the project data section.

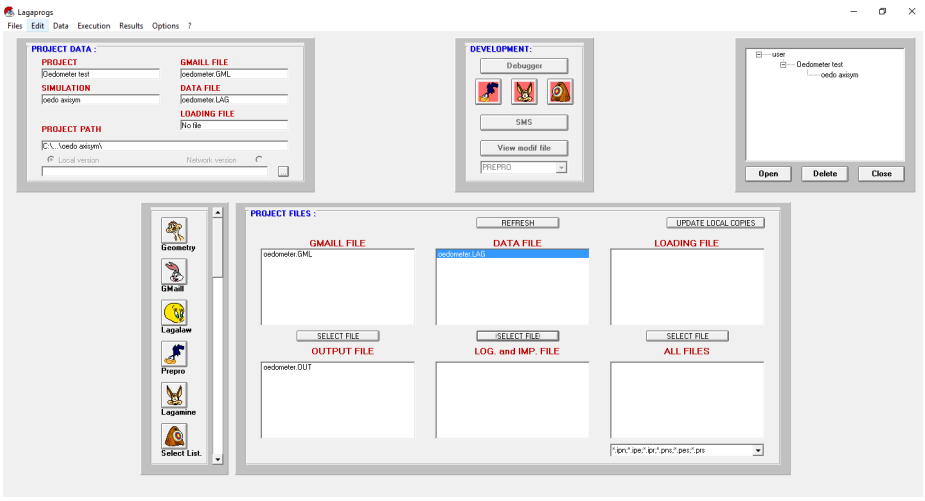

Figure 4.26

Every time GMAILL is being executed, a new `.LAG` file with the same name as the `.GML` file is created. That means that, if you have started filling the `.LAG` file and launch GMAILL, the file that you had previously filled will be lost.

4.7 Visualisation of the mesh with DESFIN

The software DESFIN available from the LAGAPROGS interface allows us to visualise the mesh. To do so, click on Results from the toolbar and then select DESFIN.

Depending in the version of DESFIN you are using, the following message can appear "0 ERREUR CATALOGUE: 1. LE CATALOGUE N'EXISTE PAS. DESFIN EN BATCH; [0] = NON, [1] = OUI". In that case, press *0* and then *Enter*.

.

The software is launched and the main menu is displayed (Figure 4.27).

The software DESFIN is in French by default, but is available is English. In order to switch to English:

1. Type 13 and then press Enter to access *OPTIONS GENERALES* (general options)
2. Type 1 and then press Enter to access *LANGUES DES MENUS* (menus language). Press 2 and then Enter to select English.
3. Type 2 and then press Enter to access *LANGUE DES DESSINS* (drawings language). Press 2 and then Enter to select English.
4. Type 99 to go back to the general menu.

Type 1 for initial structure and press *Enter* to confirm your choice.

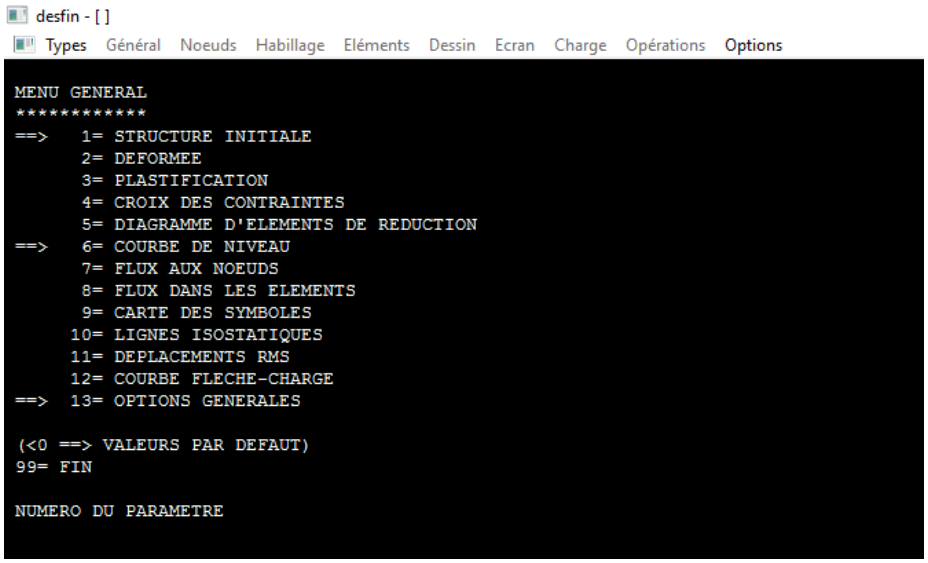

Figure 4.27

A new menu appears (Figure 4.28), type 0 to show a new drawing and press Enter again to confirm.

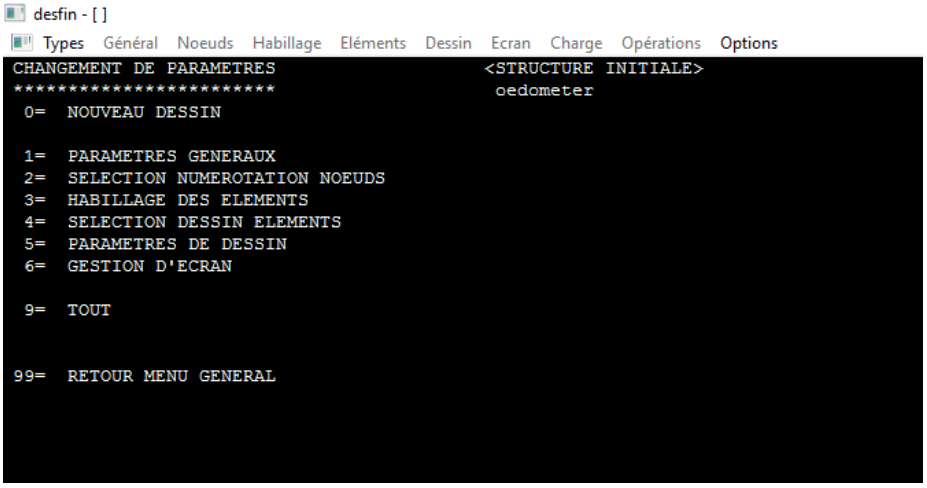

Figure 4.28

The mesh is then displayed (Figure 4.29). We have now confirmation that our sample has been correctly meshed with 10 elements in each direction. You can now quit DESFIN by selecting Types from the toolbar and then Retour Général to go back to the main menu and type 99 to exit the software. Answer Yes to the two messages popping-up.

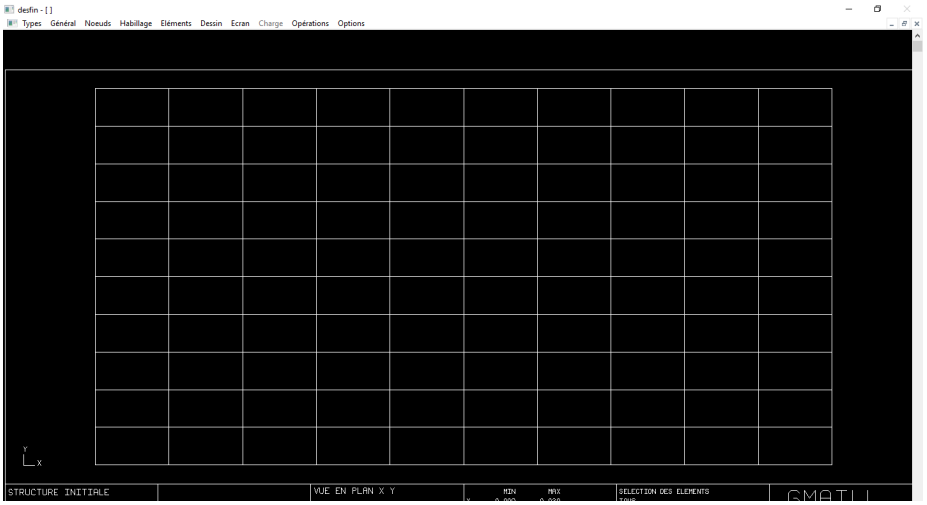

Figure 4.29

4.8 Constitutive laws and parameters (LAGALAW)

The sub-program LAGALAW allows us to define the constitutive laws and their associated parameters for the simulation. The numbers of the laws have been previously assigned while defining the contour. These can be found in the `.GML` file if you can't remember the numbering.
You can launch the program by clicking the on the icon LAGALAW or by selecting Data from the toolbar and then Lagalaw.

A new LAGALAW window opens (Figure 4.30). We first focus on the top section of this window which deals with the configuration of the different laws and their parameters.

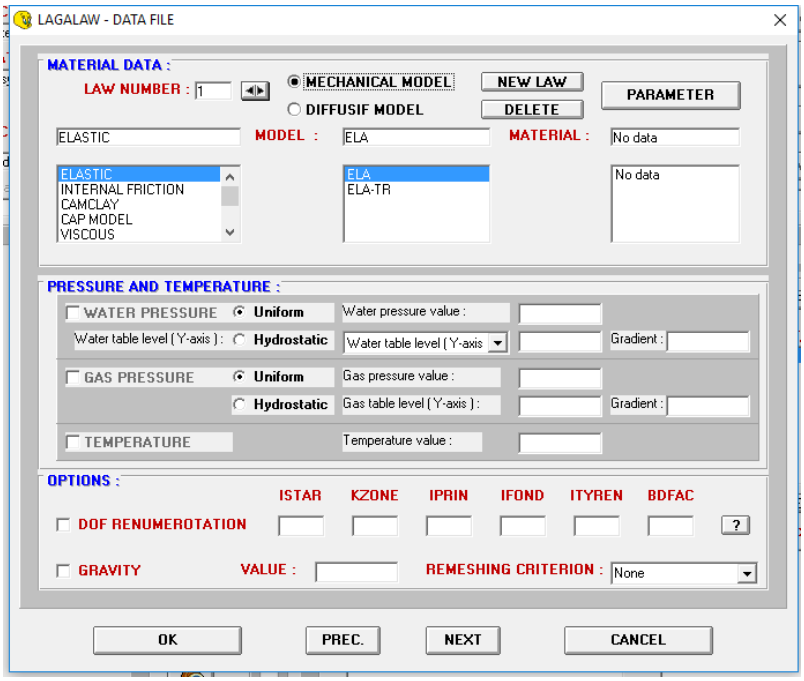

Figure 4.30

Law 1: Constitutive law of the sample The first law we assigned was attributed to the soil sample. For our case, we want an elasto-plastic behavior for our sample. Select *Internal friction* from the left scrolling menu and then the model *PLASOL*. Then click on the button Parameter.

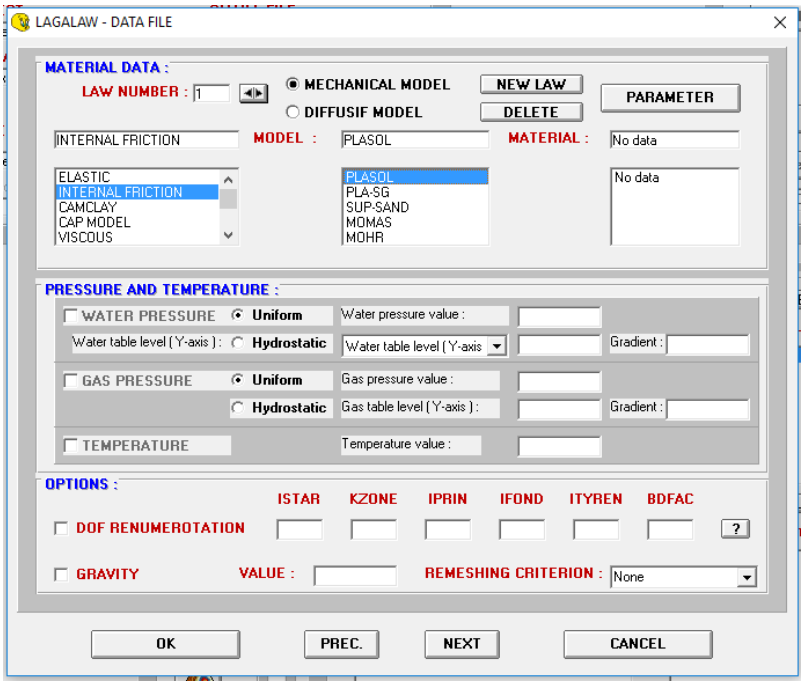

Figure 4.31

A new window appears where the parameters can be defined. Refer to the manuals for the meaning of each parameter. For our simulation, we need five parameters:

- Young modulus: $E = 500$ MPa

- Poisson's coefficient: $\nu = 0.2$
- Friction angle: $\phi = 20°$
- Cohesion: $c = 300$ kPa
- Dilatancy angle: $\psi = 0°$

The unit of reference for these parameters is the pascal Pa, fill the fields just like in Figure 4.32. The other parameters can remain at their initial value. The large right and left arrows at the bottom of the window give access to possible further windows. When all parameters have been filled, click on OK to go back on the main LAGALAW window.

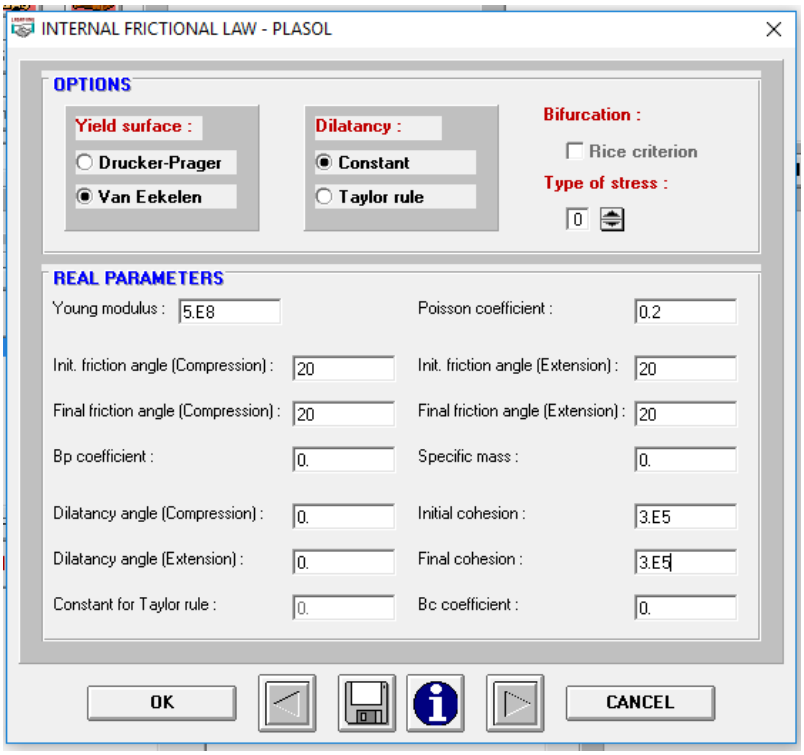

Figure 4.32

Law 2: linearly distributed load The second law assigned was the load density on top of the sample, namely a *LICHA* load. In LAGALAW main menu, click the button New law . A new law is generated and the law number should be switched to *2*. Select then *Boundary conditions* from the left scrolling menu and choose the model *LICHA*.

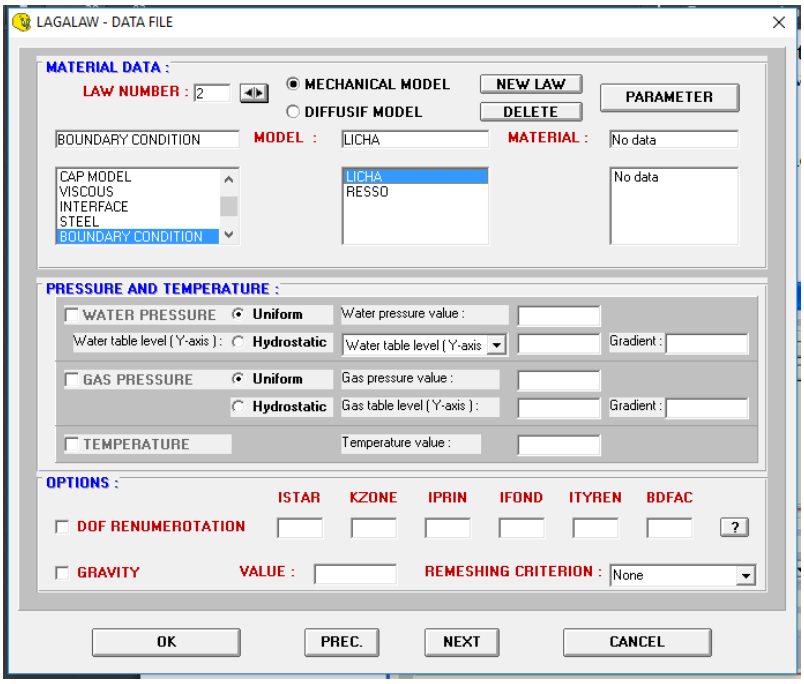

Figure 4.33

Click on Parameter, a new window opens where different models for *LICHA* can be defined (Figure 4.34).

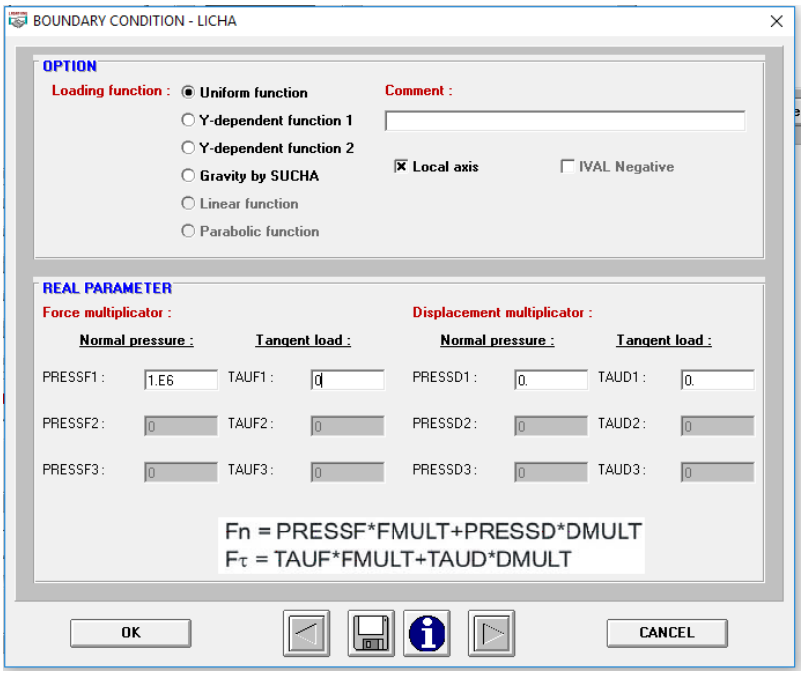

Figure 4.34

For the oedometer test, the applied force if uniformly distributed on top of the sample, but the intensity of this force increases over time. We apply a force multiplier *FMULT* to the force F to simulate this behavior. Therefore, the total force applied on the sample is $F \cdot FMULT$.

We will create later on in this tutorial an execution file whose goal will be to increase the force multiplier *FMULT* from 0 to 1. Thereby, the initial stress applied on the sample will be null, whereas

at this end of the simulation it will be equal to F.

This method implies that the force remains constant and proportional to the multiplier. This force must be normal to the top edge of the sample, and we must fill the maximum force F that will be applied at the end of the simulation, 1 MPa in our case (see Figure 4.34). Click on OK to go back to the main LAGALAW menu.

4.9 Initial pore pressure distribution (LAGALAW)

We can define in LAGALAW main menu the initial water pressures, gas pressures and the initial temperature if necessary. This part is only relevant for hydro-mechanical or thermo-hydro-mechanical coupled simulations. As our test is purely mechanical, we don't need to modify anything in this section.

Two points can be highlighted however:

- Gas and water pressures can be either defined from a uniform value or on the basis of a depth gradient and an initial value.
- The temperature of the whole model is defined from one constant value.

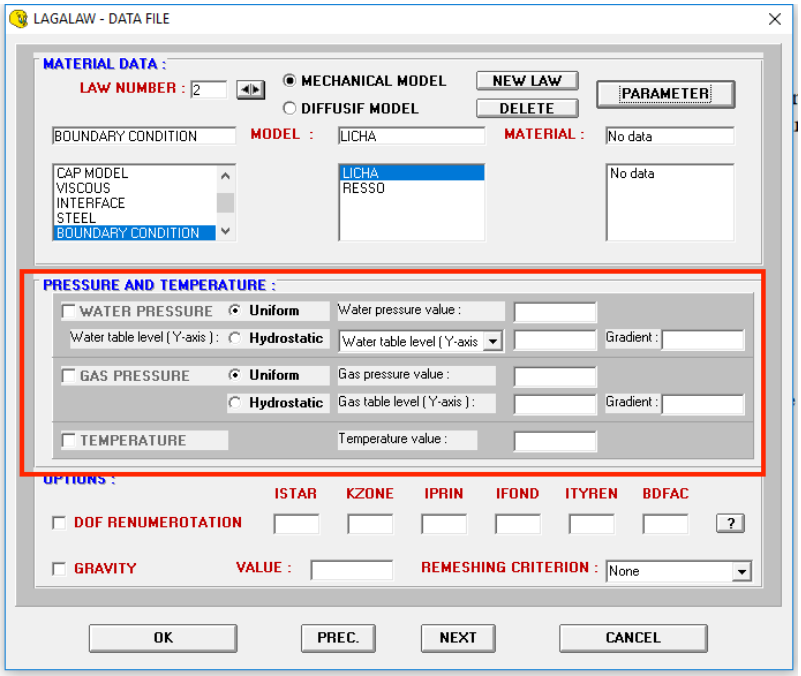

Figure 4.35

4.10 Renumbering (LAGALAW)

It is possible to renumber the degrees of freedom in order to acceleration the computation of the simulation. This option is to be configurated in the lower section of the main LAGALAW window (Figure 4.37).

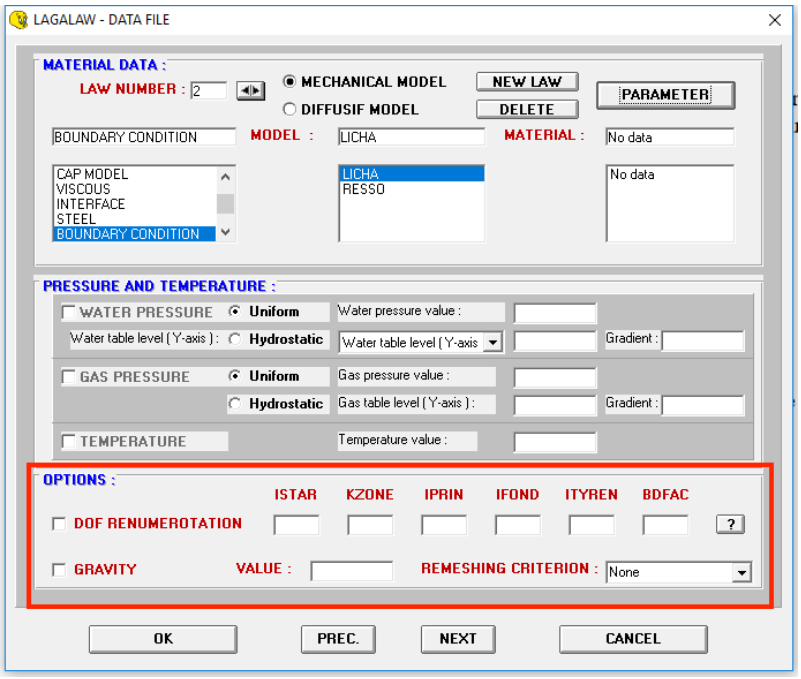

Figure 4.36

Two possibilities are available:

- **"Oil stain":** in this case we only need to check the box *DOF Renumerotation* and leave all the following fields blank.
- **Directional optimization:** (often more effective) The box *DOF Renumerotation* must also be checked the following fields must be filled as follows:
  - *ISTAR*: we need to fill a three-digit number 'abc'.
    * The digit 'a' corresponds to the direction for which the mesh has the highest number of nodes.
    * The digit 'b' corresponds to the intermediary direction.
    * The digit 'c' corresponds to the direction for which the mesh has the lowest number of nodes.

    The number 1 is the x-axis, 2 the y-axis, and 3 the z-axis. In 2D, the 'c' digit is always 0. Therefore, in our example, we should fill 210.
  - *ITYREN*: always write 2.
  - Leave the other fields blank.

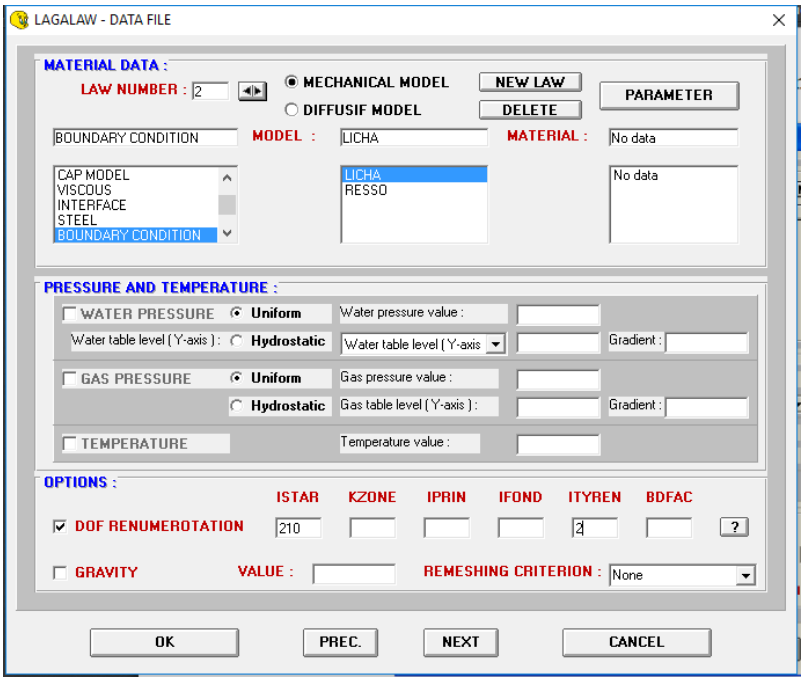

Figure 4.37

Click OK to confirm these modifications and exit LAGALAW. The .LAG file has been completed with these new information.

4.11 Execution of PREPRO

The sub-program PREPRO will update the .LAG file and pre-process all the data we have defined so far. Click on the PREPRO icon, or alternatively select Data and then Prepro.

A new window should pop-up. If the .LAG has been correctly filled previously, a message will inform you that the pre-processing is terminated (Figure 4.38), click Ok to close the program.

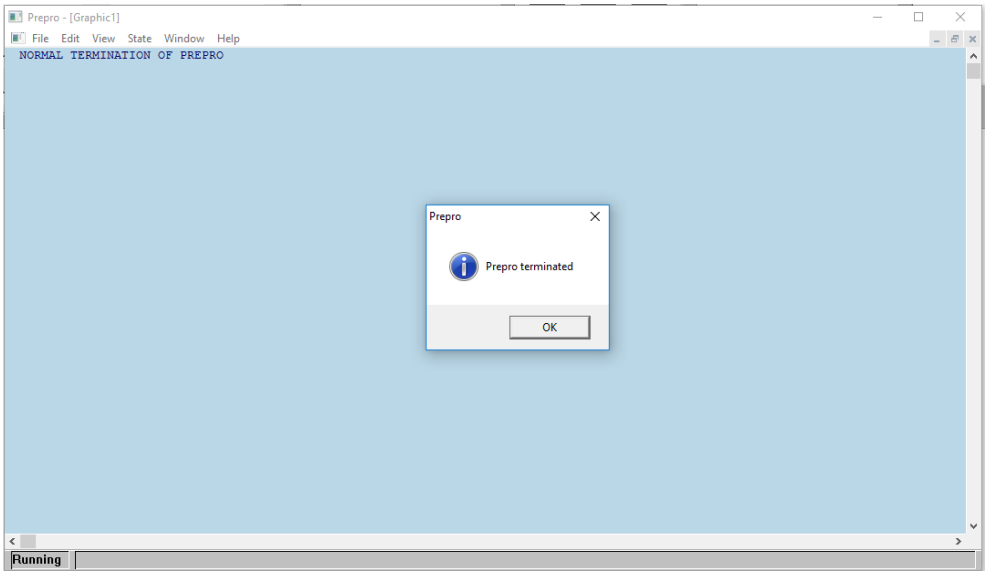

Figure 4.38

It is possible that a mistake has been made in the previous steps and that the file `.LAG` is incorrect. Executing PREPRO might then lead to an error message (Figure 4.39). In this case, you must carefully go through all the steps starting again from the geometry of the problem. For instance, common mistakes include selecting the elements in the wrong order while defining the contours (Section 4.5.3).

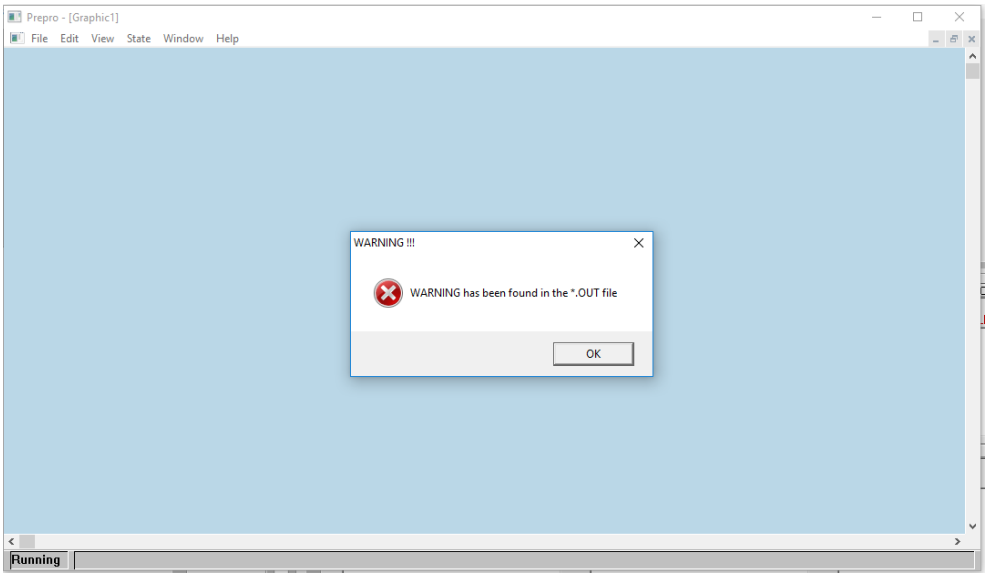

Figure 4.39

4.12 Definition of the initial stresses

To define the initial stress state, click on Data from LAGAPROGS toolbar and then select Initial stresses and Manual. A new window appears, we need to define for each group:

- The value for the initial vertical stress at $y = 0$.
- The vertical gradient for the vertical stresses.
- The value for the lateral earth pressure coefficient at rest K_0.
- The value of Biot's coefficient.

In our simulation, there is no initial stress for our soil sample as it is already in equilibrium. We must then fill the following values (see Figure 4.40):

- The value for the initial vertical stress at $y = 0$: **SIGY0 = 0**
- The vertical gradient for the vertical stresses: **DSIG =0**
- The value for the lateral earth pressure coefficient at rest: $K_0 = 1$
- It is not necessary to change the value of the Biot's coefficient as it is already by default equal to 1 (even if it is displayed 0 in the corresponding field).

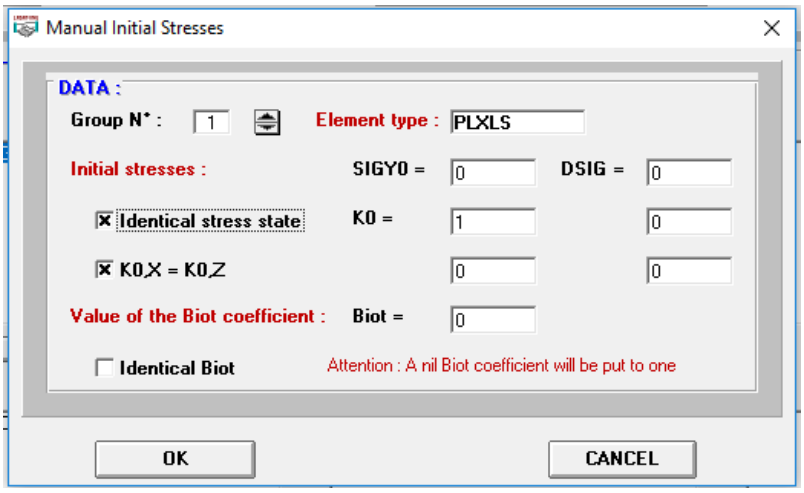

Figure 4.40

Those values are defined for the group 1 which corresponds to the PLXLS element (the soil sample). The other groups of elements are accessible with the arrow buttons. The initial conditions are identical for every group in our case. Checking the box *Identical stress state* will use the same parameters that the first group for all the others.

Click on Ok to confirm these modifications, the sub-program PREPRO will execute itself and the `.LAG` is being updated.

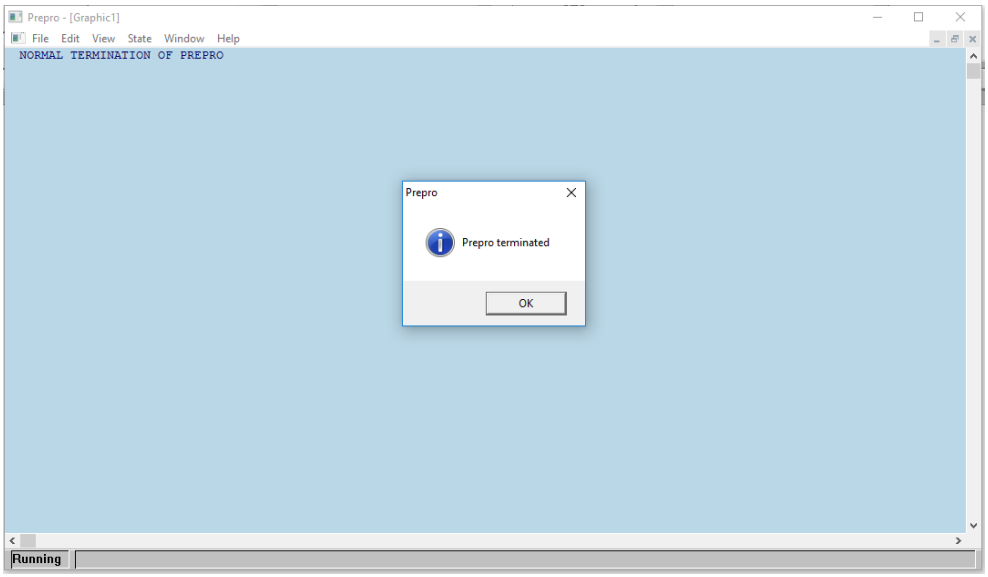

Figure 4.41

4.13 Creation of the loading file

The execution file `.DAT` deals with the evolution of the force and displacement multipliers with time. In LAGAPROGS, click on File and then New. Select the File tab and check the box Loading File. Name the file, loading_ex for instance, and click Ok.

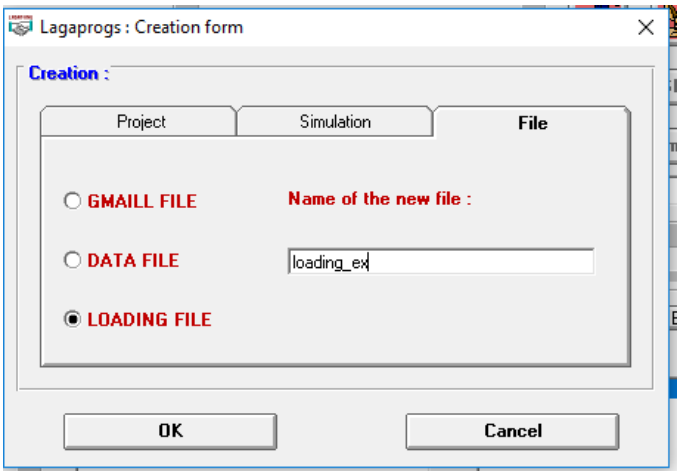

Figure 4.42

This new file shoud appear in the *Loading File* column in LAGAPROGS interface. Click on this file and on the button Select below to assign this file to the *Project Data* section.

4.14 Definition of the loading file

The loading file `.DAT` is currently empty, we now want to define the strategy of our simulation while by the `.DAT` file. Select Execution from the toolbar and then Loading File. Two options are available: the Automatic strategy or the Manual strategy. Only the former strategy is elaborated in this tutorial.

The automatic strategy consists in defining general parameters for the strategy and the features of the first time step. The LAGAMINE code will determine itself the following time steps until convergence , it is impossible to predict in advance these steps. Once the automatic option is selected, a new window opens (Figure 4.43).

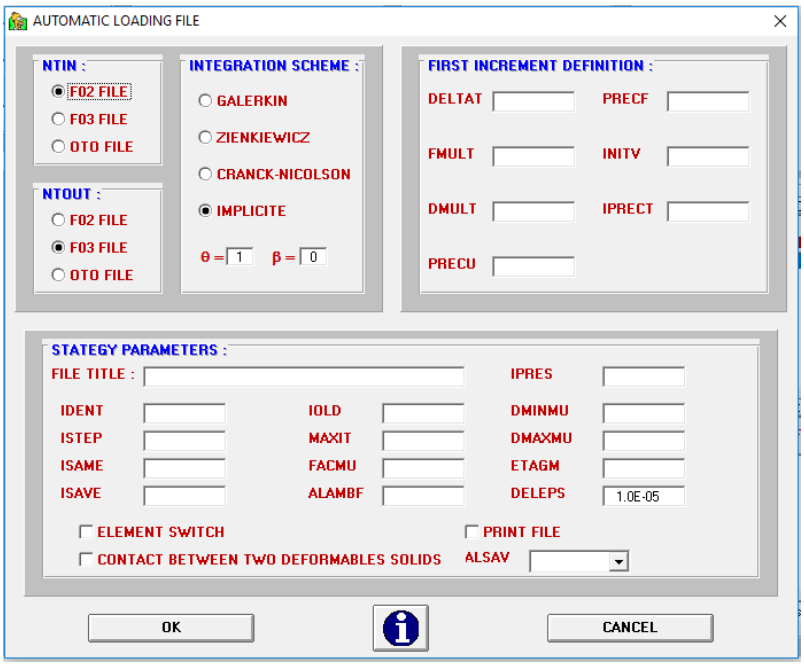

Figure 4.43

Several parameters must be filled to define the strategy:

- **DELTAT:** First time step in seconds.
- **FMULT:** value of the force multiplier FMULT for the first time step.
- **DMULT:** value of the force multiplier DMULT of the first time step.
- **PRECU:** precision of the displacements (usually 10^{-3}) for all time steps
- **PRECF:** precision of the forces (usually 10^{-3}) for all time steps
- **INITV:** possible re-initialization of convergence speed. Set at value at 1.
- **IPRECT:** Convergence criteria. Set value at 1 (=convergence when both criteria are met, see manual for further explanations)
- **IPRES:** Set value here at 0 for this example.
  - 0 by default
  - 1 if .DEP file is used
  - 8 if .LIC file is used
  - 18 if both files are used
- **IDENT:** 0 by default
- **ISTEP:** Number of this first timestep. Set value at 1.
- **ISAME:** Maximum number of iterations for the simulation. Set value at 9999.
- **ISAVE:** Set value at -1.
- **IOLD:** Number of the time-step for which the file NTIN should start. Set value at 1 here.
- **MAXIT:** Maximum number of iterations per timestep. Set value at 5.
- **FACMU:** Multiplicative factor (in case of convergence) or division factor (in case of divergence). Set value at 2.
- **ALAMBF:** End of the simulation in seconds. Set value at 0.1
- **DMINMU:** Minimum timestep in seconds. Set value at 10^{-6}.
- **DMAXMU:** Maximum timestep in seconds. Set value at 0.05
- **ETAGM:** 0 by default.
- **DELEPS:** 10^{-5} by default.
- **ALSAV:** Write in this box the time steps for which the simulation data will be saved in the .OTO file. Write here the following values (you need to press Enter to validate the entry for each value): 0.001, 0.005, 0.01, 0.05, 0.1.

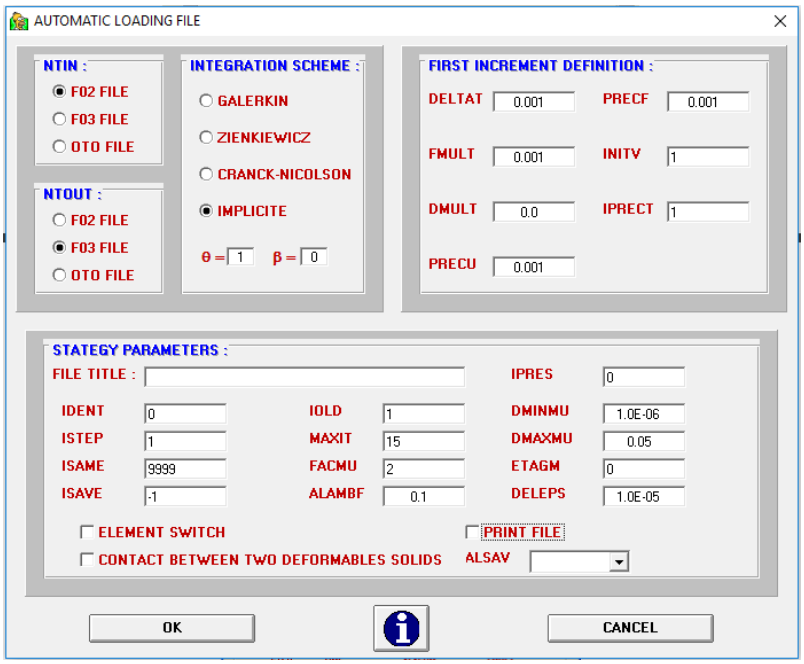

Figure 4.44

Check all the parameters, including the different time steps in **ALSAV**. Click on Ok to confirm these modifications.

4.15 Execution of LAGAMINE

Click on the LAGAMINE icon to run the code based on the loading and data files currently selected. A window will automatically open, displaying the evolution of the simulation for all time steps. If the simulation is run without any problems, the following window appears (Figure 4.45):

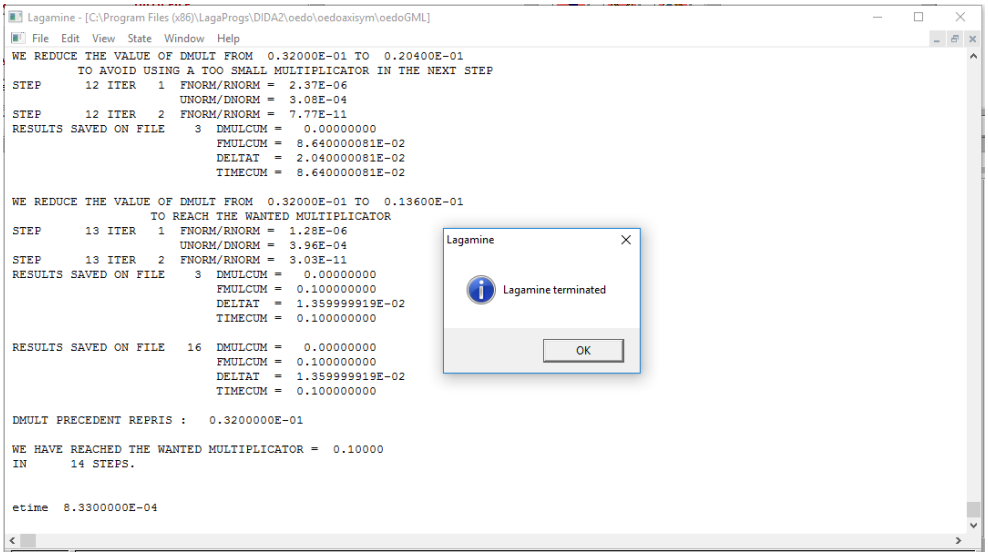

Figure 4.45

4.16 Selection of the variables to visualise with SELECT

In order to be able to draw and visual the updated mesh in DESFIN, we first need to select the results of the simulation with SELECT. From the toolbar, click on Results and then on SELECT and DESFIN file. The following window appears:

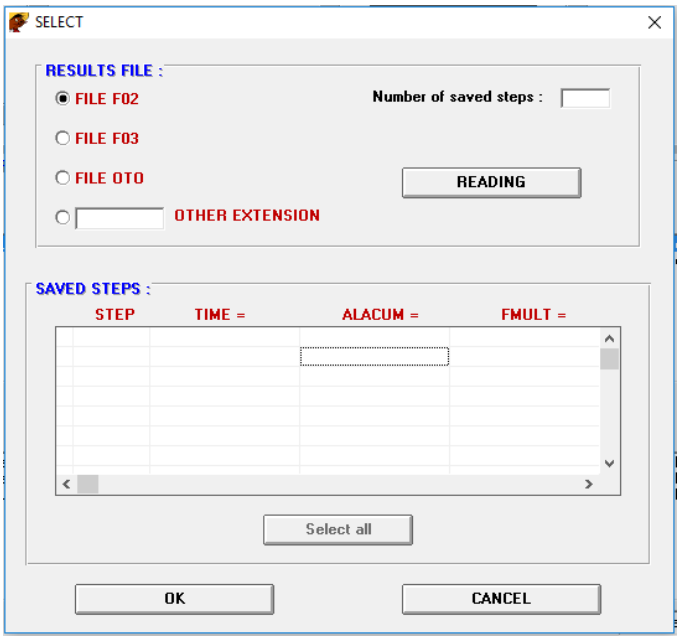

Figure 4.46

One can choose the type of file where the results have been saved (`.F02`, `.F03`, `.OTO`). Check the box File F03 and click on the button Reading. The time steps that have been saved appear in the lower part of the window (Figure 4.47). Here, only one step has been saved, but if the box File OTO is checked, all the time steps specified in ALSAV in the loading file are visible. Select one or several steps by clicking on the line or by clicking the button Select all. Click on Ok.

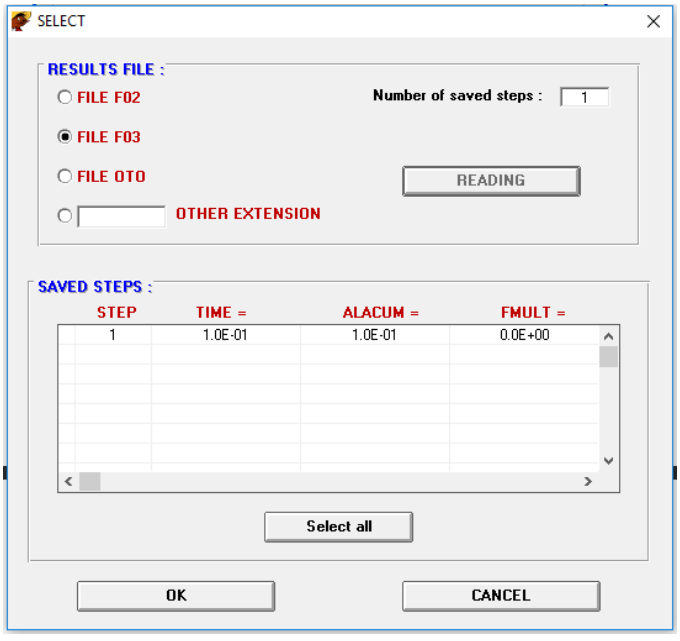

Figure 4.47 One time step has been saved

A new window appears with four different tabs:

- *Nodal values:*, it is possible to select the displacements, the updated coordinates, the reactions and the velocities at the nodes. Check here the first two boxes (Figure 4.48).

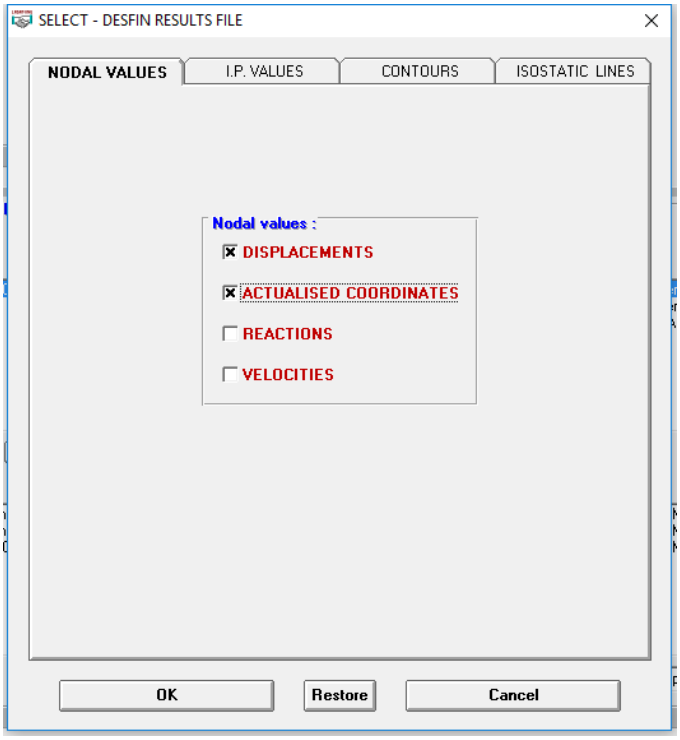

Figure 4.48

- *I.P values:* results at the integration points can be selected here, for example the pressure for the *LICHA* elements.
- *Contours:* results can be visualized as contours (stresses, state variables, deformations, etc.)

Once all the desired variables are selected, press the button OK to finalise the selection process, the window SELECT should close and a message appears saying that the .RES file has been created (Figure 4.49).

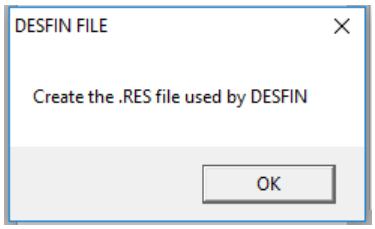

Figure 4.49

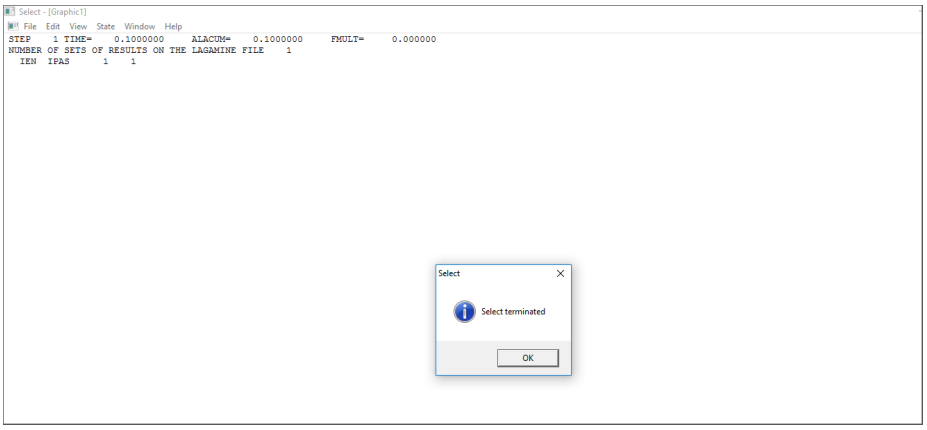

Figure 4.50

You can now go to the program DESFIN and select *2=DEFORMEE* and then *0=NOUVEAU DESSIN* to draw the updated mesh. The white solid lines in Figure 4.51 represent the initial structure and the dashed red lines represent the deformed mesh at the time step considered.

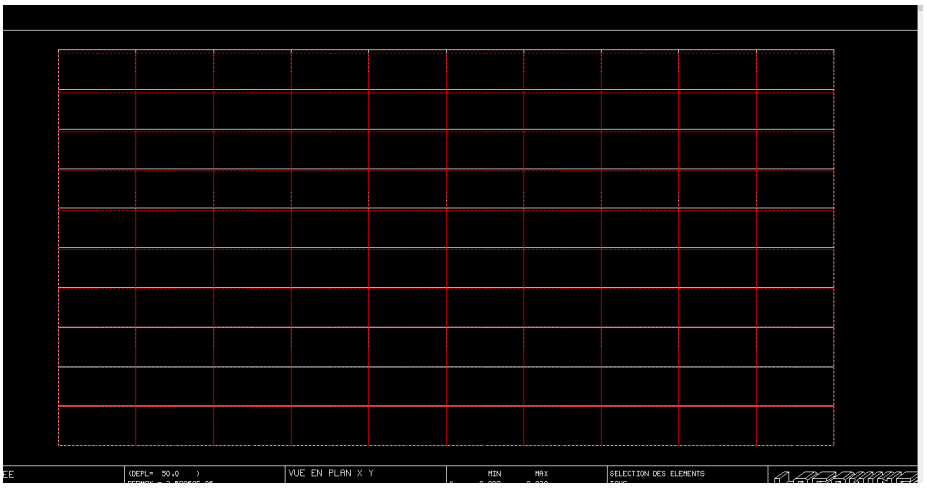

Figure 4.51

4.17 Export mesh in .PS format

It is possible to export the drawings from DESFIN into a postscript file. This is highly recommended when inserting drawings in a report instead of screenshot from DESFIN. This is done with the following steps:

- After the drawing of the mesh in DESFIN, select the Operations tab and then Save Graphic File. A message appears indicating that the figure has been saved (Figure 4.52).

Figure 4.52

- Exit and close the software DESFIN.
- In LAGAPROGS, select the tab Results, then PLOTTER and PS Maker.
- A window appears to select the drawings to be exported (4.53). When pressing OK, a .PS file is created in your simulation folder.

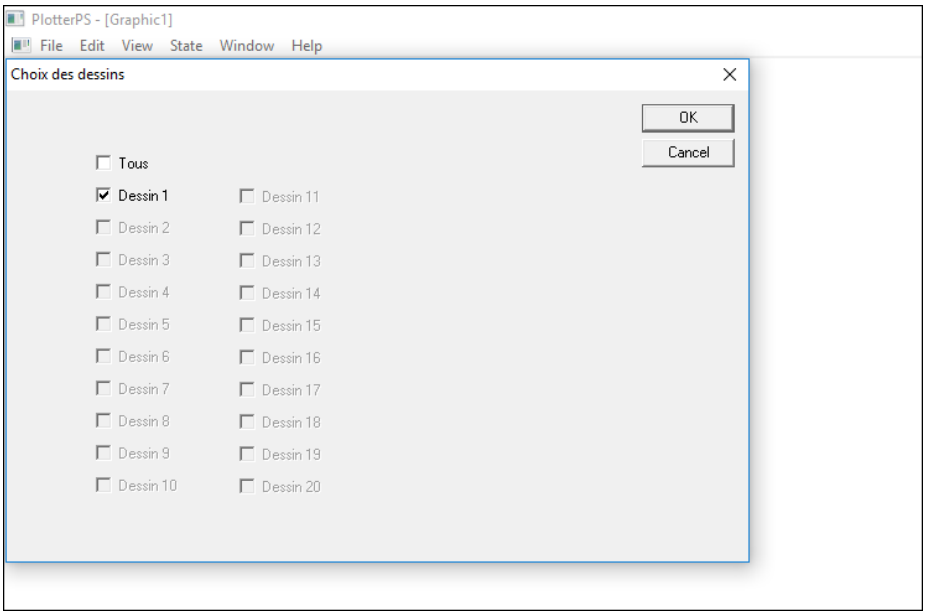

Figure 4.53 Select here which drawings you want to export in .PS format

- Open the .PS file with your favorite postscript software, you obtain something similar to Figure 4.54.

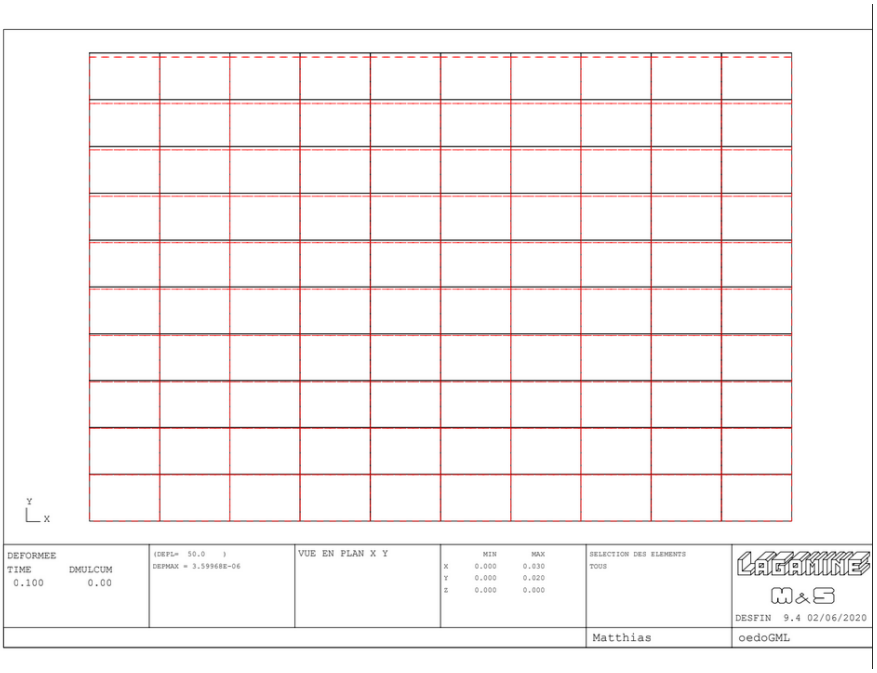

Figure 4.54 Initial and deformed mesh in post-script format.

Chapter 5

Triaxial test

5.1 Description of the problem

This section describes the simulation of a triaxial test in LAGAMINE using only the mechanical elements *PLXLS*. Two approaches with different boundary conditions are introduced:

- A stress-controlled approach
- A strain-controlled (more specifically, displacement-controlled) approach

Both cases share the same geometry and type of analysis, namely 2D axisymmetric, shown in Figure 5.1.

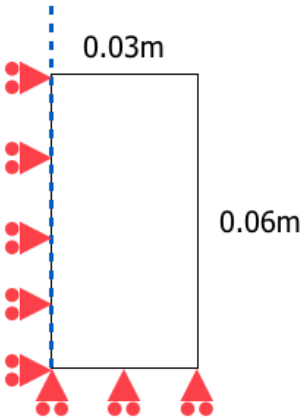

Figure 5.1 Geometry and initial boundary conditions for the triaxial test.

Analysis and problem type Select 2D MECHANICAL for *ANALYSIS TYPE* and AXISYMMETRIC for *PROBLEM TYPE*.

5.2 Stress-controlled drained

For the stress-controlled simulation, two different phases of the triaxial test are modeled (Figure 5.2). Starting from a situation with no initial stress, we define:

- Phase 1: isotropic loading compression from 0 to 100 kPa
- Phase 2: axial compression to 500 kPa

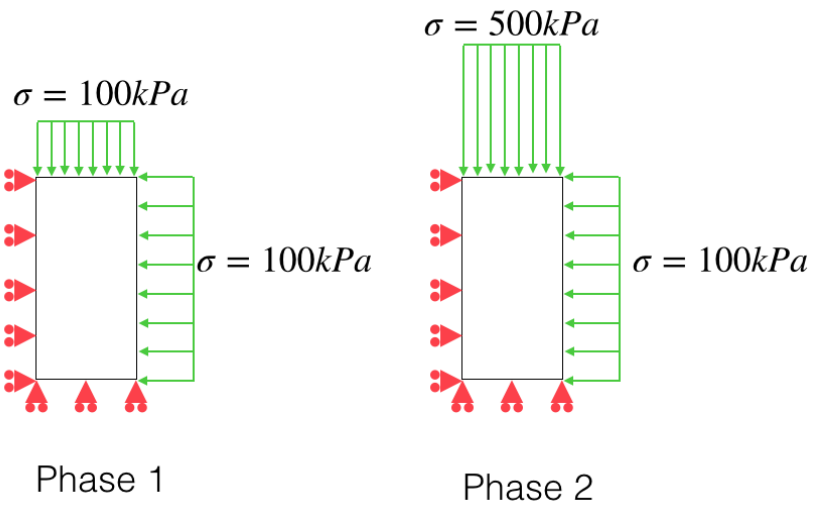

Figure 5.2 Phases definition for the triaxial test. Boundary conditions at the end of Phase 1 and Phase 2.

Geometry Build a 0.03 m x 0.06 m rectangular sample with only one *PLXLS* element. Set the subdivision of segments to 1 to obtain a mesh with a single element. The boundary condition at the bottom of the sample should be fixed in the y-direction (DOF number 2) and the the boundary condition at the left of the sample should be fixed in the x-direction to simulate the symmetry of the problem (DOF number 1). Two *LICHA* elements should be imposed on the top (LICHA2) and right (LICHA1) sides of the sample to simulate the loading (Figure 5.3). Detailed steps are:

1. Go to Data –> Geometry and boundary condition .
2. Edit –> Charactrerize point . Then define the points. Then Quit –> Preview –> Fit to window .
3. Options –> Division of segment –> Edit –> segment . Click New to create the segments one by one. Then Quit .
4. Now, define the boundary by Edit –> Boundary . New –>*select Type 2*–>*select all the four boundary and click OK*. In the new window, select *PLXLS* (note that the 1st mechanical law is given to the soil here). Click OK.
5. Now, set the boundary condition. Click New –>*select Type 5*. Click the bottom one then click OK, select DOF FIXED with 2 (y direction), and click OK. Click New , click the left one then click OK, select DOF FIXED with 1 (x direction), and click OK. New , click the right one then click OK, select LICHA and change mechanical law N to 2, and click OK. New , click the top one then click OK, select LICHA and change mechanical law N to 3, and click OK. Because we need to control the top boundary and right boundary separately, we need to assign two LICHA to them. Finally, Quit . File –> Save Data –> Yes .
6. Execution GMAIL by clicking the GMAIL logo–>Click OK–> Refresh –>click the LAG file–>click Select .

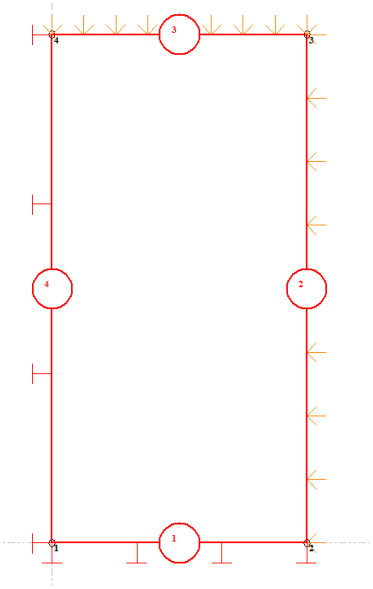

Figure 5.3 Geometry of the stress-controlled triaxial test in LAGA-G

LAGALAW Now we define the laws.

1. Data –> Lagalaw –> Interal Friction –>*PLASOL* model for the mechanical law of the *PLXLS* elements. Enter the parameters by clicking Parameter then filling the table as shown in Figure 5.4. Click OK.
2. Click New Law , now the Law Number is 2 which is for the right-side boundary. Select Boundary condition –> LICHA –> Parameter , set PRESSF1 to 100e3 and others as 0 (see explanation of the paragraph below), then click OK.
3. Click New Law , now the Law Number is 3 which is for the top-side boundary. Select Boundary condition –> LICHA –> Parameter , set PRESSD1 to 500e3 and others as 0 (see explanation of the paragraph below), then click OK.
4. Click OK to complete the definition of LagaLaw.

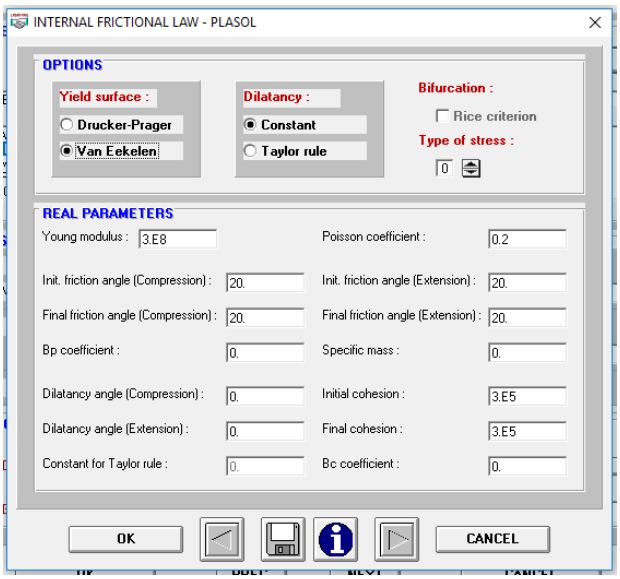

Figure 5.4 Parameter of the law *PLASOL* in LAGALAW

In LAGAMINE, the evolution of two *LICHA* elements can be simultaneously controlled using two different multipliers *FMULT* and *DMULT*. For the *LICHA* element applied on the right side of the sample, we will use the *FMULT* multiplier with a final normal pressure *PRESSF1* equal to 100 kPa. For the *LICHA* element applied on top of the sample, we will use the *DMULT* multiplier with a final normal pressure *PRESSD1* equal to 500 kPa.

`.LIC` **file** Referring to appendix C to create the LIC file. The two multipliers follow different evolution through time in such a way that the sample remains in isotropic compression during the first phase (see Figure 5.5).

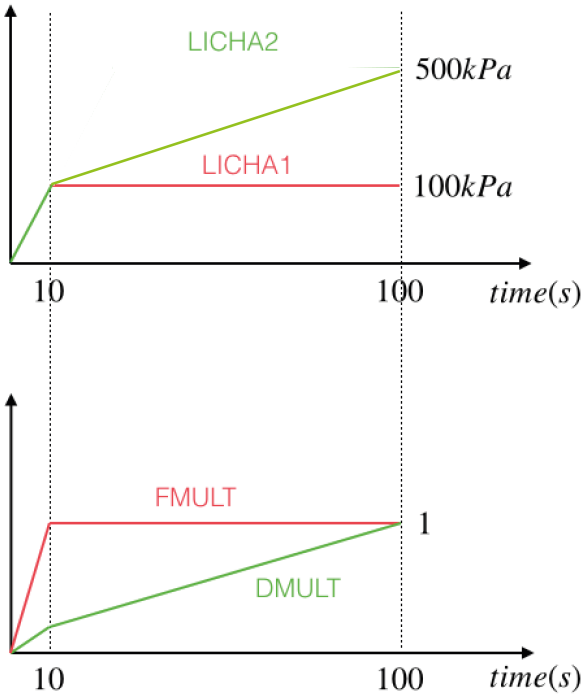

Figure 5.5 Evolution of the *LICHA* elements and *FMULT* and *DMULT* multipliers in time.

The evolution of the multipliers is controlled with a `.LIC` file. Refer to section C for background information with the handling of `.LIC` files. Three moments are defined at $t = 0$ s, $t = 10$ s and $t = 100$ s. The values of *FMULT* and *DMULT* at these specific moments are given by the user (Figure 5.6). The values at intermediate times are linearly interpolated.

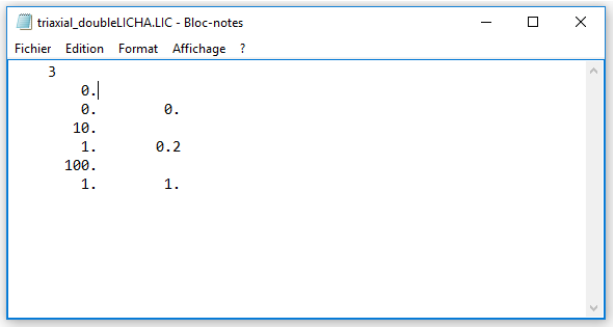

Figure 5.6 Definition of the `.LIC` file for the triaxial test.

Execution of Prepro click Prepro logo (or Data –> Prepro), then click OK.

Loading file Remarks: The parameters of the loading file for an automatic strategy are given in Figure 5.7. It is important to set *FMULT* and *DMULT* at 0 because these are monitored by the `.LIC` file (*IPRES* = 8). Failing to do so will result in a error message when launching LAGAMINE: "Stop because automatic strategy is impossible with both imposed displacement and forces". The parameter *ALAMBF* indicates the time moment for which the simulation ends, it is set at 100 s in this case.

Detailed steps are as follows:

1. Creation of the loading file: File –> New –> File button, select Loading file. –> Click select file in the loading File panel.
2. Execution –> Loading file –> Automatic strategy

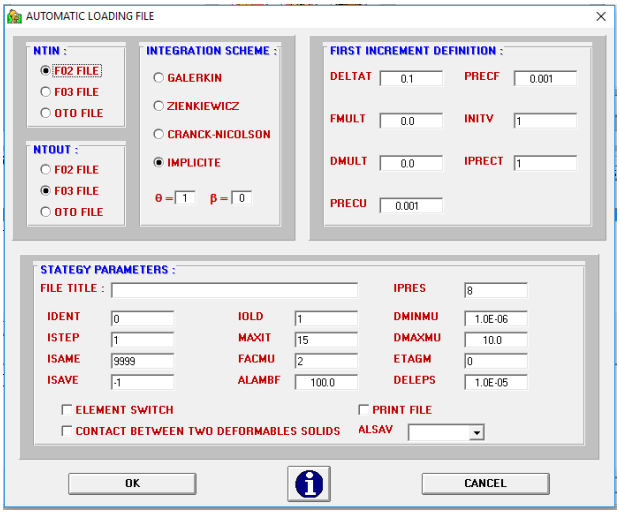

Figure 5.7 Definition of the loading file for the triaxial test.

Remarks: all the files (such as .LIC and .DEP file) should be ready before the execution of PrePro, except the loading file. The loading file can be set just before the execution of Lagamine. Namely: Get all files ready –> execute prepro –> set loading file –> execute Lagamine.

Execution of Lagamine Click on the Lagamine icon to run the code.

Output Following these steps should provide a result similar to Figure 5.8. Steps could be:

1. Results –> SELECT –> Desfin. And select files that you want to output.
2. Results –> Desfin to go to Desfin and visualise the mesh.

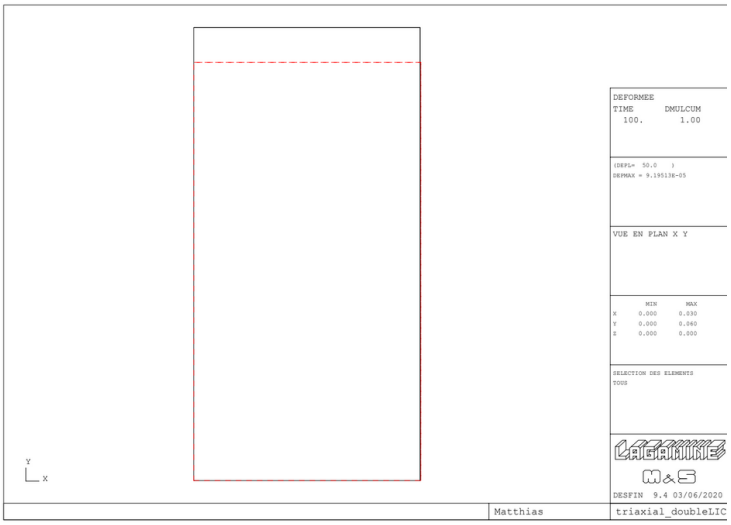

Figure 5.8 Deformed mesh for the stress-controlled simulation.

5.3 Displacement-controlled drained

For the displacement control type of drained triaxial compression demonstrated herein. The consolidation phase is skipped and the simulation directly starts from a consolidated state at 100 kPa. The top wall is controlled by displacement, the right-lateral wall boundary is controlled by LICHA (applying a constant confining pressure).

1. Creation of the project, simulation and Gmail.
2. From the LagaProgs toolbar, click on Data and then on Analysis and problem type. A new window pops up where you can select the type of analysis and the type of problem. Our case can be simplified as an axisymmetric problem for a purely 2D mechanical analysis (no hydro-mechanical coupling is considered). Select 2D MECHANICAL under *ANALYSIS TYPE* and AXISYMMETRIC under *PROBLEM TYPE*.

Geometry Remarks: Apply *LICHA* elements on the right side (stress control for constant confining pressure). Since we will monitor the position of the top nodes in the y-direction, it is required to fix the corresponding degree of freedom at the top of the sample (DOF number 2). See (Figure 5.9).

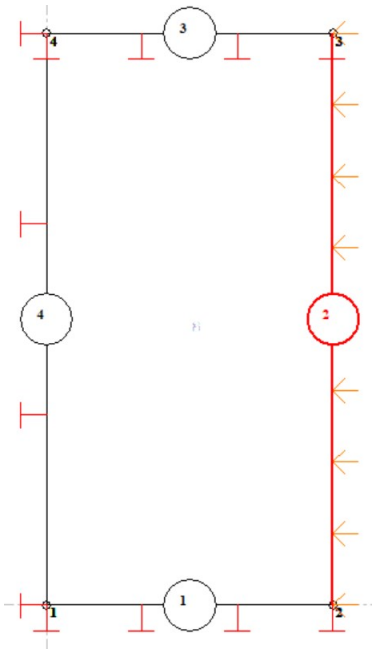

Figure 5.9 Geometry of the displacement-controlled simulation.

Detailed steps are:

1. Go to Data –> Geometry and boundary condition .
2. Edit –> Charactrerize point . Then define the points. Then Quit –> Preview –> Fit to window .
3. Options –> Division of segment –> Edit –> segment . Click New to create the segments one by one. Then Quit .
4. Now, define the boundary by Edit –> Boundary . New –>*select Type 2*–>*select all the four boundary and click OK*. In the new window, select *PLXLS* (note that the 1st mechanical law is given to the soil here). Click OK.
5. Now, set the boundary condition. Click New –>*select Type 5*. Click the bottom one then click OK, select DOF FIXED with 2 (y direction), and click OK. Click New , click the left one then click OK, select DOF FIXED with 1 (x direction), and click OK. New , click the top one then click OK, select DOF FIXED with 2 (y direction), and click OK. New , click the right one then click OK, select LICHA and change mechanical law to 2, and click OK. Finally, Quit . File –> Save Data –> Yes .
6. Execution GMAIL by clicking the GMAIL logo–>Click OK–> Refresh –>click the LAG file–>click Select .

LAGALAW Now we define the laws.

1. Use a *PLASOL* model for the mechanical law of the *PLXLS* elements. Enter the parameters as shown in Figure 5.10. Namely, Data –> Lagalaw –> Interal Friction –>*PLASOL* model for the mechanical law of the *PLXLS* elements. Enter the parameters by clicking Parameter then filling the table as shown in Figure 5.10. Click OK.
2. Click New Law , now the Law Number is 2 which is for the right-side boundary. Select Boundary condition –> LICHA –> Parameter , set PRESSF1 to 100e3 and others as 0,

then click OK. Define a LIC file is needed. The LIC file of this example is shown in Fig. 5.11. Note that, in this example, we don not need the consolidation process, hence, in the LIC file, the FMULT is directly set as 1 at the initial time 0.

3. Click OK to complete the definition of LagaLaw.

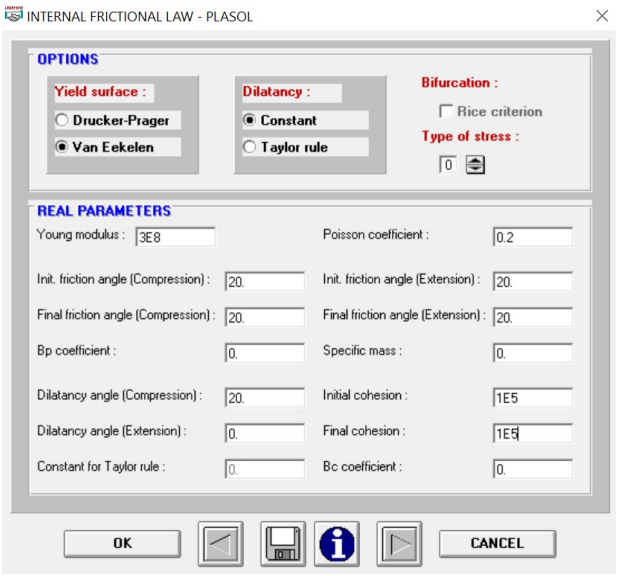

Figure 5.10 Parameter of the law *PLASOL* in LAGALAW

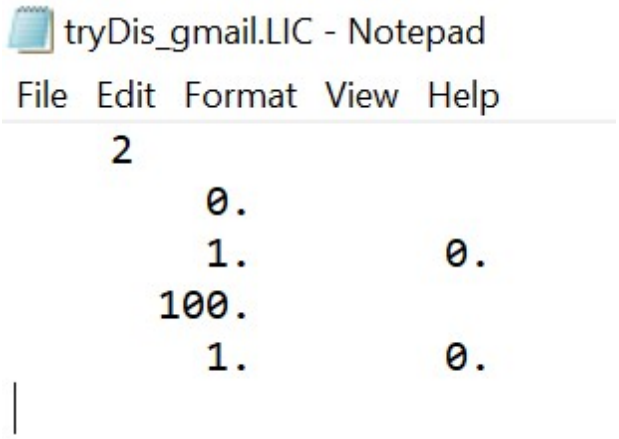

Figure 5.11 LIC file in this example.

`.DEP` **file** The position of every *fixed* node can controlled using a `.DEP` file. Refer to Section D for the handling of `.DEP` files. In this example, the domain is 10 × 10 elements (when setting the segment as 10). Go to DESFIN, input 1 (then press enter,same for following)-> 1 -> 3 -> 1 -> 0 . You will see the ID of the nodes. As we control the displacement of the top boundary, we need to take down the ID of the nodes at the top. (Or we can Check your `.LAG` file to find which nodes to fix as it depends on the creation order of the points in LAGA-G.) After that, define the DEP file according to Appendix D. The DEP file is shown in Fig. 5.12. It demonstrates 20% of axial strain.

tryDis_gmail.DEP - Notepad

File Edit Format View Help

```
  2   21    0
      0.
  3    2     0.060
  4    2     0.060
 43    2     0.060
 44    2     0.060
 45    2     0.060
 46    2     0.060
 47    2     0.060
 48    2     0.060
 49    2     0.060
 50    2     0.060
 51    2     0.060
 52    2     0.060
 53    2     0.060
 54    2     0.060
 55    2     0.060
 56    2     0.060
 57    2     0.060
 58    2     0.060
 59    2     0.060
 60    2     0.060
 61    2     0.060
    100.
  3    2     0.048
  4    2     0.048
 43    2     0.048
 44    2     0.048
 45    2     0.048
 46    2     0.048
 47    2     0.048
 48    2     0.048
 49    2     0.048
 50    2     0.048
 51    2     0.048
 52    2     0.048
 53    2     0.048
 54    2     0.048
 55    2     0.048
 56    2     0.048
 57    2     0.048
 58    2     0.048
 59    2     0.048
 60    2     0.048
 61    2     0.048
```

Figure 5.12 DEP file in this example.

Initial stresses The initial situation simulates the isotropic compression of the sample. In the window "Manual Initial Stresses" (Figure 5.13), assign an initial vertical stress σ_{y0} of -100 kPa (compressive stresses are negative). Assign a lateral earth coefficient K_0 of 1 to emulate an isotropic situation, and check the box "Identical stress state". Operations are: Data –> Initial stresses –> Manual .

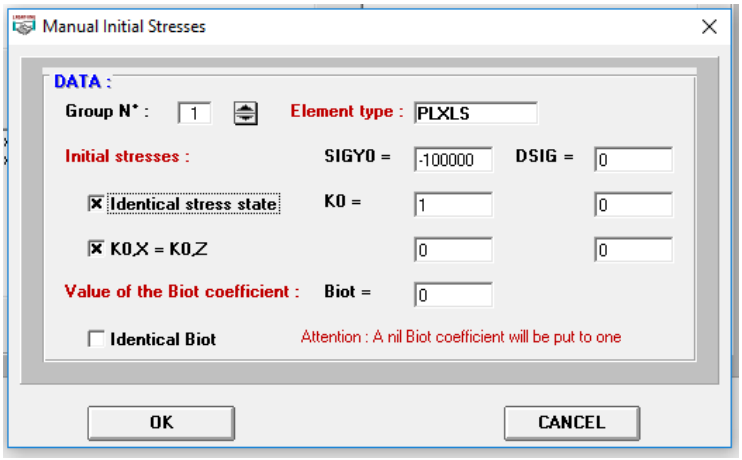

Figure 5.13 Manual Initial stress window for the displacement-controlled simulation.

Execution of Prepro click Prepro logo (or Data –> Prepro), then click OK.

Loading file Figure 5.14 shows the configuration of the automatic strategy for the displacement-controlled simulation. The difference with the stress-controlled simulation is that *IPRES* is set to 18 for using both `.LIC` and `.DEP` files. Special note that, since in the LIC file and DEP file, the final time is set at 100 s, therefore, in Figure 5.14, ALAMBF should be set to 100 to be consistent. DMAXMU can be set to 1 to speed up.

Detailed steps are:

1. Creation of the loading file: File –> New –> File button, select Loading file. –> Click select file in the loading File panel.
2. Execution –> Loading file –> Automatic strategy

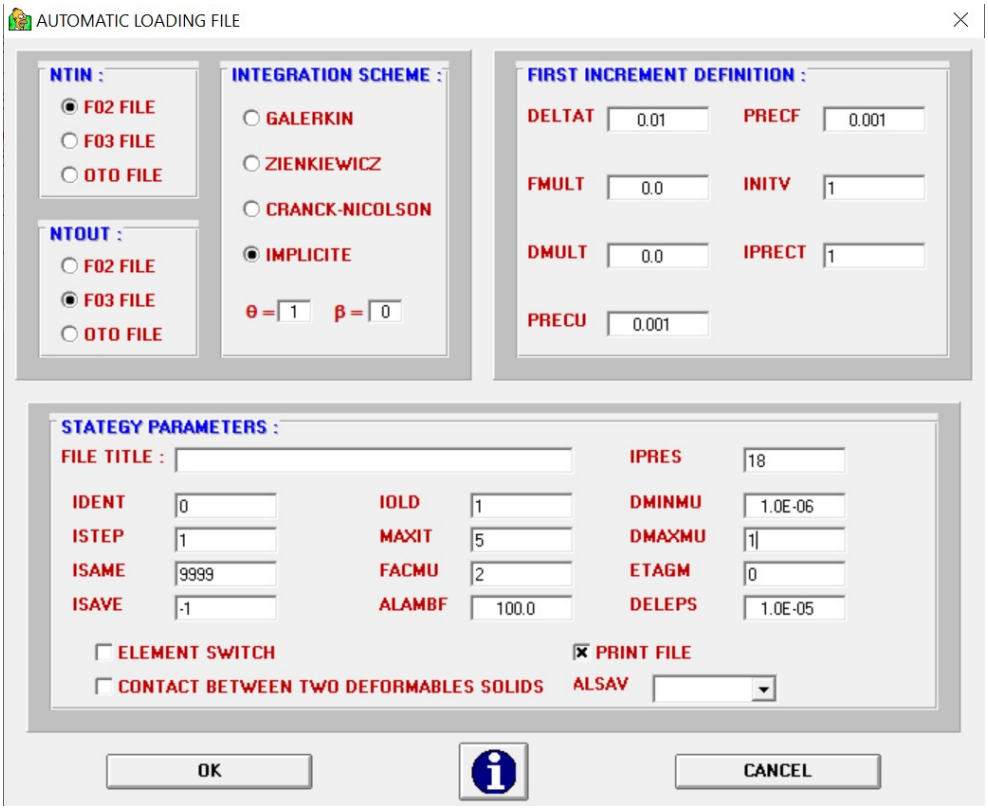

Figure 5.14 Loading settings in this example.

Note that in figure 5.14, the PRINT FILE is checked. If there is no PRI file defined, then execution of Lagamine here will give an error. You can run Lagamine without checking PRINT FILE to see whether the program works well. Otherwise, before the next step (Execution of Lagamine), you need to define a PRI file.

Execution of Lagamine Click on the Lagamine icon to run the code.

Results - effect of confining pressure Here, it demonstrates the triaxial test (by displacement control) under 100 to 300 kPa of confinement as shown in Fig. 5.15. The two places we need to change with the confining pressure are the Lagalaw and initial stress:

1. Click Lagalaw –>Go to LICHA–> Parameter, set PRESSF1 to 200e3 and others as 0, then click OK.

2. Data –> Initial stresses –> Manual . Assign an initial vertical stress σ_{y0} of -200 kPa (compressive stresses are negative).

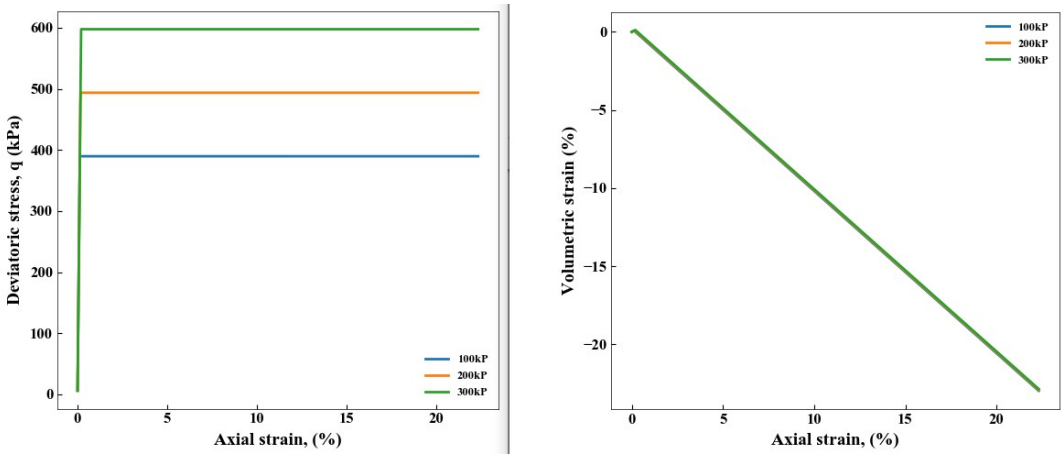

Figure 5.15 Loading settings in this example.

5.4 Displacement-controlled Undrained

The previous two sections demonstrate the triaxial compression (strain control and displacement control) in the drained condition. In this section, the displacement control undrained triaxial compression test is presented.

For the displacement control type of undrained drained triaxial compression demonstrated herein. It is basically the same as the displacement-controlled drained triaxial test shown in the last section. The boundary condition (Gmail) is the same (no pressure is introduced in this case). However, the problem of analysis and the constitutive law are different.

1. Creation of the project, simulation and Gmail.
2. From the LAGAPROGS toolbar, click on Data and then on Analysis and problem type . A new window pops up where you can select the type of analysis and the type of problem. Our case can be simplified as an axisymmetric problem for a purely 2D mechanical analysis (no hydro-mechanical coupling is considered). Select 2D Hydro-MECHANICAL under *ANALYSIS TYPE* and AXISYMMETRIC under *PROBLEM TYPE*.

Geometry Remarks: Apply *LICHA* elements on the right side (stress control for constant confining pressure). Since we will monitor the position of the top nodes in the y-direction, it is required to fix the corresponding degree of freedom at the top of the sample (DOF number 2). See (Figure 5.9).

Detailed steps are:

1. Go to Data –> Geometry and boundary condition .
2. Edit –> Charactrerize point . Then define the points. Then Quit –> Preview –> Fit to window .
3. Options –> Division of segment –> Edit –> segment . Click New to create the segments one by one. Then Quit .
4. Now, define the boundary by Edit –> Boundary . New –>*select Type 2–>select all the four boundary and click OK*. In the new window, select *CSOL2* (note that the 1st mechanical law

is given to the soil here, and the law number 2 is given to the flow law). Click OK. (see Fig. 5.16)

5. Now, set the boundary condition. Click New –>*select Type 5*. Click the bottom one then click OK, select DOF FIXED with 2 (y direction), and click OK. Click New, click the left one then click OK, select DOF FIXED with 1 (x direction), and click OK. New, click the top one then click OK, select DOF FIXED with 2 (y direction), and click OK. New, click the right one then click OK, select LICHA and change mechanical law to 3, and click OK. Finally, Quit. File –> Save Data –> Yes.
6. Execution GMAIL by clicking the GMAIL logo–>Click OK–> Refresh –>click the LAG file–>click Select.

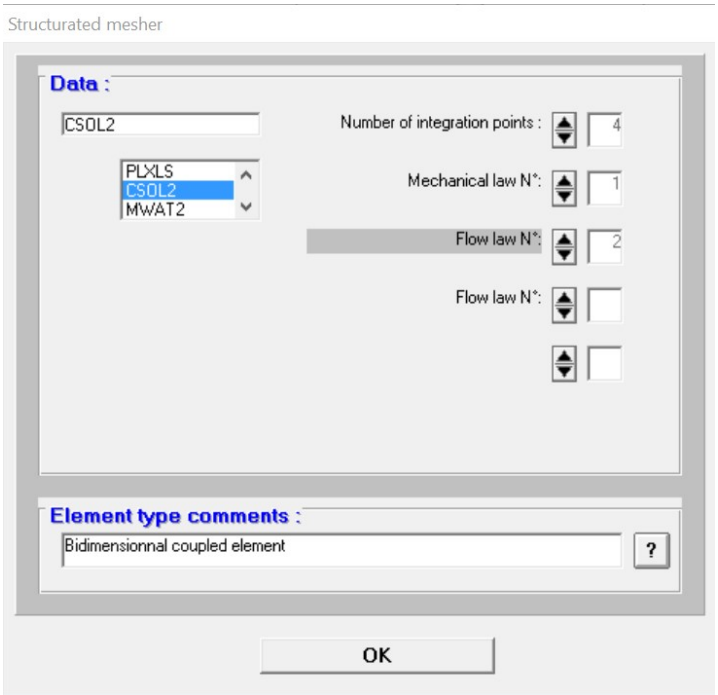

Figure 5.16 Loading settings in this example.

LAGALAW Now we define the laws.

1. Use a *PLASOL* model for the mechanical law of the *PLXLS* elements. Enter the parameters as shown in Figure 5.10. Namely, Data –> Lagalaw –> Interal Friction –>*PLASOL* model for the mechanical law of the *PLXLS* elements. Enter the parameters by clicking Parameter then filling the table as shown in Figure 5.17. Click OK.
2. Click New Law, now the Law Number is 2 which is for the flow boundary. Select Diffusif model –> FLUID DIFFUSION –> ECOU-S –> Parameter, set parameters as shown in Fig. 5.18.
3. Click New Law, now the Law Number is 3 which is for the right-side boundary. Select Boundary condition –> LICHA –> Parameter, set PRESSF1 to 100e3 and others as 0, then click OK. Define a LIC file is needed. The LIC file of this example is shown in Fig. 5.11. Note that, in this example, we don not need the consolidation process, hence, in the LIC file, the FMULT is directly set as 1 at the initial time 0. (the same as the drained case except the law number)
4. Click OK to complete the definition of LagaLaw.

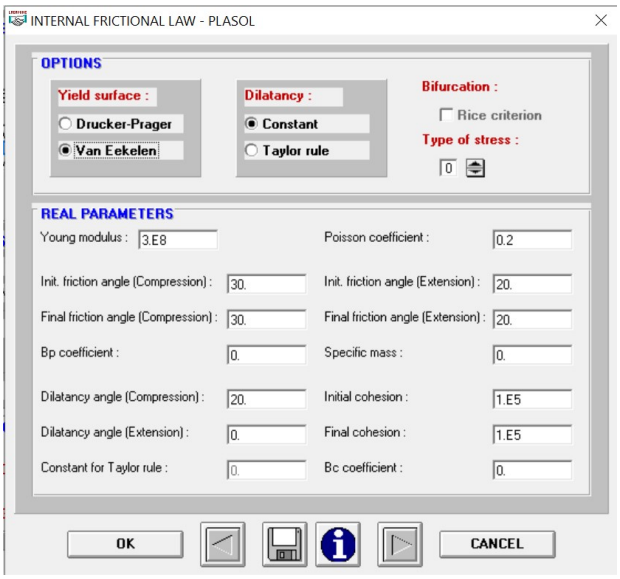

Figure 5.17 Parameter of the law *PLASOL* in LAGALAW

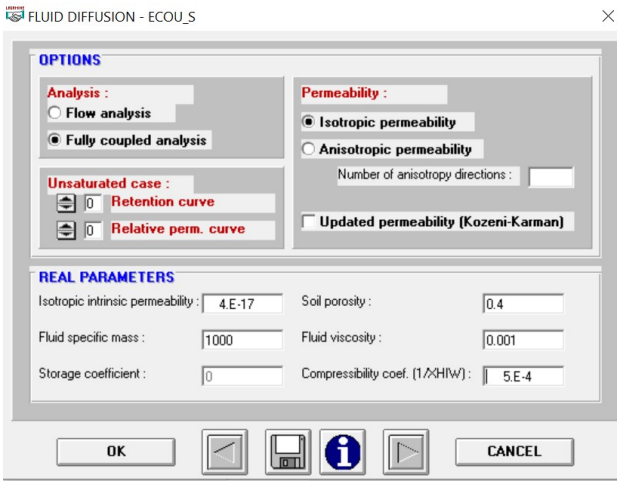

Figure 5.18 Parameter of the law *ECOU-S* in LAGALAW

`.DEP` **file** The position of every *fixed* node can controlled using a `.DEP` file. Refer to Section D for the handling of `.DEP` files. In this example, the domain is 10 $\times$ 10 elements (when setting the segment as 10). Go to DESFIN, input 1 (then press enter,same for following)-> 1 -> 3 -> 1 -> 0. You will see the ID of the nodes. As we control the displacement of the top boundary, we need to take down the ID of the nodes at the top. (Or we can Check your `.LAG` file to find which nodes to fix as it depends on the creation order of the points in LAGA-G.) After that, define the DEP file according to Appendix D. The DEP file is shown in Fig. 5.12. It demonstrates 20% of axial strain. (the same as the drained case)

Initial stresses The initial situation simulates the isotropic compression of the sample. In the window "Manual Initial Stresses" (Figure 5.13), assign an initial vertical stress σ_{y0} of -100 kPa (compressive stresses are negative). Assign a lateral earth coefficient K_0 of 1 to emulate an isotropic situation, and check the box "Identical stress state". Operations are: Data –> Initial stresses –> Manual. (The same as the drained case)

Execution of Prepro click Prepro logo (or Data –> Prepro), then click OK.

Loading file Figure 5.14 shows the configuration of the automatic strategy for the displacement-controlled simulation. The difference with the stress-controlled simulation is that *IPRES* is set to 18 for using both `.LIC` and `.DEP` files. Special note that, since in the LIC file and DEP file, the final time is set at 100 s, therefore, in Figure 5.14, ALAMBF should be set to 100 to be consistent. DMAXMU can be set to 1 to speed up. (The same as the drained case)

Detailed steps are:

1. Creation of the loading file: File –> New –> File button, select Loading file. –> Click select file in the loading File panel.
2. Execution –> Loading file –> Automatic strategy

Note that in figure 5.14, the PRINT FILE is checked. If there is no PRI file defined, then execution of Lagamine here will give an error. You can run Lagamine without checking PRINT FILE to see whether the program works well. Otherwise, before the next step (Execution of Lagamine), you need to define a PRI file.

Execution of Lagamine Click on the Lagamine icon to run the code.

Appendices

Appendix A

File description: format and structure

LAGAMINE and the related programs deal with different working files. They may take these files as input but, they may also fill them in or modify them. Therefore, these files are formatted and structured in such a way that it is readable for the programs, mostly written in Fortran. The global structure of the simulation files obeys specific rules that must be learned if the user want to modify the files directly.

Most of the files of your simulation can be read with your favorite text editor. This section doe not aim at explaining the meaning of every value written in the files as this is already elaborated in the manuals and DOKUWIKI. However this section emphasizes how to read those manuals and the rules that must be adopted when writing in the simulation's files.

It is recommended to use the free text editor NOTEPAD++, which enables different tabs in a same window, updates files and contains several useful macros.

A.1 Format

The simulation's files are made of several lines, each of them containing specific information. Each line can be seen as a collection of numbers and letters or words. Let us consider that the document is a giant table where each cell is either left empty or filled with a single digit or a single letter.

The format for each line is given in the manuals and follows the same pattern: one letter flanked with two numbers.

- The central letter indicates the type of object. It can be either an integer (I), a floating (G), or a string (A).
- The left number indicates how many objects are in the line considered for the given type.
- The right number indicates how much space these objects take or how many cells are allocated for each one of these objects.

The symbol "/" is used to indicate that a new line should start.

These formats describe the maximum number of items that can be read on the same line (or on several lines when the symbol "/" is used). If some items are not present, the program will consider them as equal to zero.

Example 1. A line with the format "3I5" consists of 3 integers, each integer being allocated with 5 cells. Typically, the digits are written starting from the rightmost cell (as shown below for 1 and 20), but this is not compulsory. In the example below, each blank or empty cell corresponds to one space.

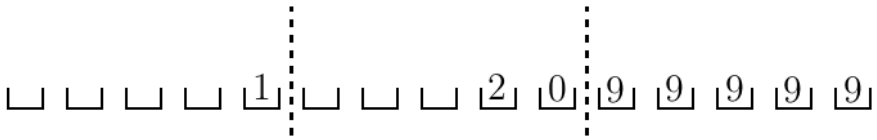

Figure A.1

Example 2. Let us suppose we want to write the two floating numbers 0.2 and $5 \cdot 10^8$ on a line whose format is "2G8.0" (2 floats, 8 cells allocated to each floating number). The decimal point and the exponent both take up one cell, as shown below.

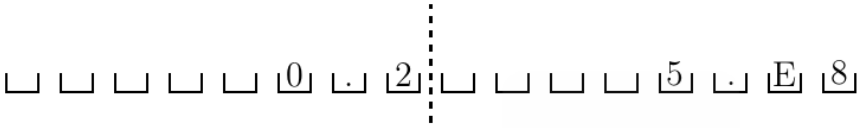

Figure A.2

Example 3. A last example is provided below, with the format on the left-hand side and the cell numbering at the top and bottom.

```
                    5   10   15   20   25   30   35   40   45   50   55
                ....|....|....|....|....|....|....|....|....|....|....|..
                ┌──────────────────────────────────────────────────────────┐
14I5            │   -2    4 2493    1 1752    0    0    0    0    1       │
14I5            │    0    0    0   30   60   18   10                      │
14I5            │    0                                                    │
14I5            │    0                                                    │
A5              │NODES                                                    │
I5,8G10.0       │    1  0.000000  0.000000  0.000000  0.000000            │
I5,8G10.0       │    2  0.000000  0.000000  0.400000  0.000000            │
I5,8G10.0       │    3  0.000000  0.000000  0.800000  0.000000            │
I5,8G10.0       │    4  0.000000  0.000000  1.200000  0.000000            │
I5,8G10.0       │    5 15.000000  0.000000  0.000000  0.000000            │
I5,8G10.0       │    6 15.000000  0.000000  0.400000  0.000000            │
…               │…                                                        │
                └──────────────────────────────────────────────────────────┘
                ....|....|....|....|....|....|....|....|....|....|....|..
                    5   10   15   20   25   30   35   40   45   50   55
```

Figure A.3

A.2 Structure

Each file obeys to a very specific structure that is thoroughly described in the manuals. It is primordial to respect the structure; otherwise, the sub-programs will not be able to run.

Appendix B

Manuals

The finite element code LAGAMINE, and its associated programs, are documented in a series of manuals and the DOKUWIKI.

- `Lagamine.pdf`: general description of the code
- `Intro.pdf`: structure of the data file `NAMDAT.LAG`
- `Lagamex.pdf`: structure of the loading file `NAMEX.DAT`
- `Annexes.pdf`: description of some specific sections
- `Elmec.pdf`: description of the mechanical elements
- `Elthe.pdf`: description of the thermo-mechanical elements
- `Elcou.pdf`: description of the coupled elements
- `Lawmec.pdf`: description of the mechanical modems
- `Lawther.pdf`: description of the flow or thermal models
- `Lawcou.pdf`: description of the coupled constitutive models

Appendix C

Control of the loading history (.LIC file)

The loading history (i.e., the evolution of external forces or fluxes) can be controlled using a .LIC file. This .LIC file is used to impose the evolution over time of both the force multiplier *FMULT* and the displacement multiplier *DMULT*. These multipliers will multiply constant values defined in the constitutive laws (i.e., in LICHA or LICHT).

In order to use a .LIC file, the user must use ***IPRES* = 8** in the automatic loading strategy window. In the case of both .LIC and .DEP files, ***IPRES* = 18** should be used.

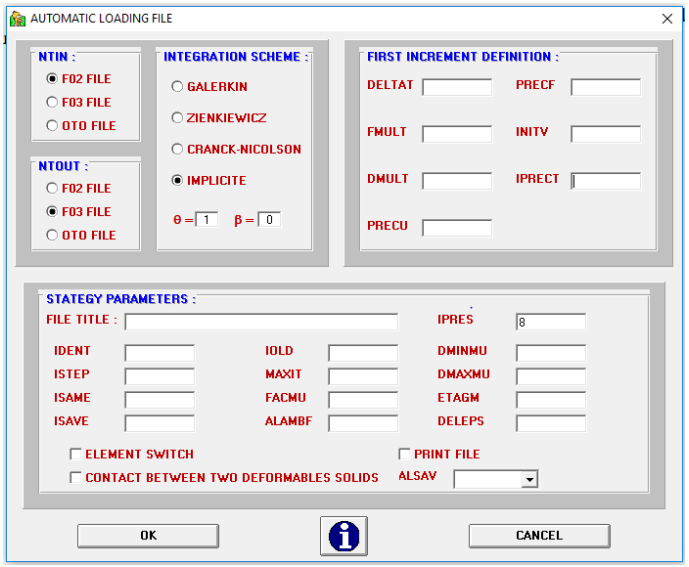

Figure C.1

To create a .LIC file, go to the toolbar in LagaProgs and select Options, then Tools and LIC file. A new text document opens, the NAMDAT.LIC file, which has the same name as the NAMDAT.LAG file.

There is no user-friendly program to fill in the .LIC file, which needs to be completed manually. The format[1] is the following:

[1] Refer to Chapter A to understand the format.

- Line 1 (1I5): number of time moments of the imposed values
- Line 2 (1G10.0): definition of the first time moment t_1
- Line 3 (2G10.0): imposed value of *FMULT* at t_1 and imposed value of *DMULT* at t_1
- Line 4 (1G10.0): definition of the second time moment t_2
- Line 5 (2G10.0): imposed value of *FMULT* at t_2 and imposed value of *DMULT* at t_2
- and so on for all time moments.

Note that the linear interpolation is performed between two consecutive time moments t_i and t_{i+1}.

Example We want to impose the following evolution of *FMULT* and *DMULT*.

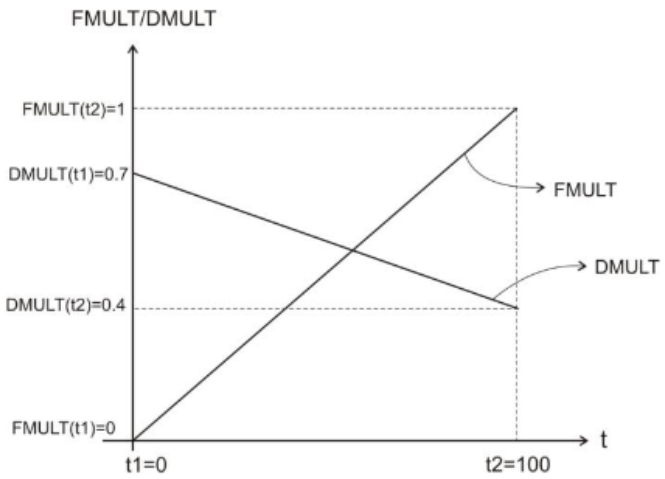

Figure C.2

The .LIC file should then be written as shown below.

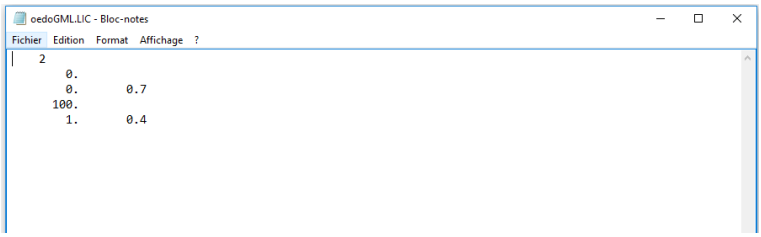

Figure C.3

The international system of units is used in LAGAPROGS so that times are expressed in seconds.

More information on the .LIC file can be found in the DOKUWIKI.

Appendix D

Control of the evolution of prescribed degrees of freedom (.DEP file)

It is possible to control the evolution of prescribed degrees of freedom through the whole simulation using a .DEP file.

All degrees of freedom that are controlled using a .DEP file must be **fixed** in the corresponding .LAG file.

In order to use a .DEP file, the user must use ***IPRES* = 1** in the automatic loading strategy window. In case of both .LIC and .DEP files, ***IPRES* = 18** should be used.

To create a .DEP file, go to the toolbar in LAGAPROGS and select Options, then Tools and DEP file. A new text document opens, the NAMDAT.DEP file, which has the same name as the NAMDAT.LAG file.

There is no user-friendly program to fill in the .DEP file, which needs to be completed manually. An example of a .DEP file is given below to control the position of the top nodes in a strain-controlled triaxial test.

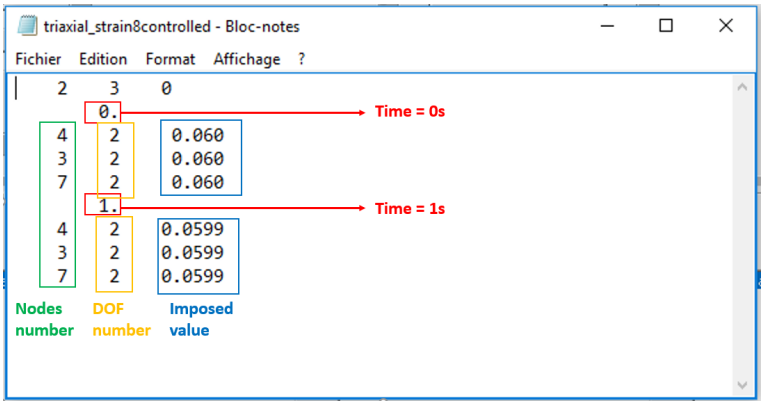

Figure D.1

The first line made of three integers (3I5) is formatted as follows:

- Number of time moments for which the values are imposed (format I5)
- Number of definitions at each time moment (format I5)
- Set to 0 by default (format I5)

In this example, the second degree of freedom is imposed for three nodes (3, 4, and 7) at two different time moments: 0 s and 1 s.

More information on the .LIC file can be found in the DokuWiki.

Appendix E

Printing output values (`.PRI` file)

It is possible to configure the printing options in a `.PRI` file in order to obtain the following outputs:

- nodal values (x, y, z, p_w, p_g and T) for specific nodes, given in a `.IPN` file.
- particular values at integration points (stresses and state variables) for certain elements, given in a `.IPE` file.
- values of reaction for specific nodes, given in a `.IPR` file.

When using a `.PRI` file, the user must check the box *PRINT FILE* in the loading file window.

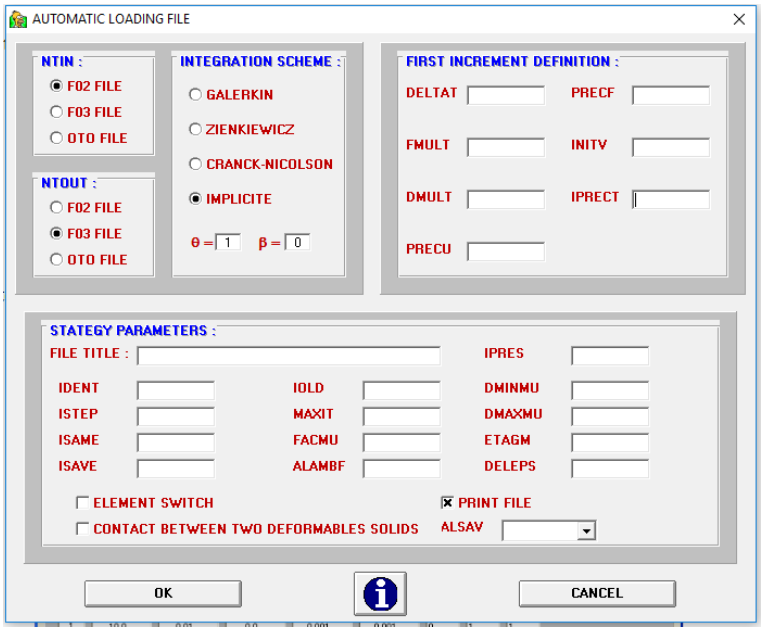

Figure E.1

To create a `.PRI` file, go to the toolbar in LAGRAPROGS and select Options, then Tools and PRI file. A new blank document opens, the `NAMDAT.PRI` file, which has the same name as the `NAMDAT.LAG` file. There is no friendly-user sub-program to fill the `.PRI` file, which needs to be completed manually. Explanations on how to fill the file can be found in the DOKUWIKI. Detailed introduction on PRI file can be found in the tutorial given by Sophie.