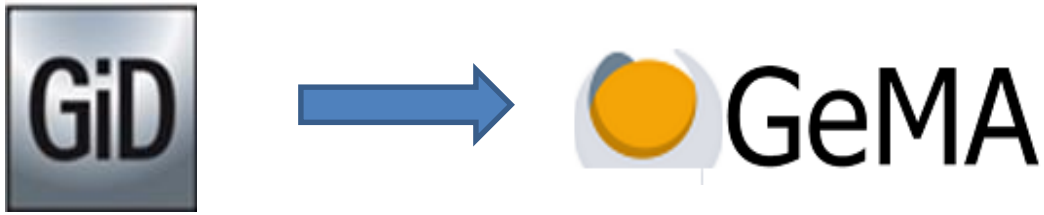
GiD-GeMA Plugin: Reference Tutorial

Tecgraf Institute – Modeling & Multiphysics Simulation Group – July 2018

Generalities

- *GiD* is an interactive user interface employed for definition, preparation and visualization of all the data associated to a numerical simulation. In this sense, it is necessary to define the geometry, materials, conditions and other parameters in order to generate a mesh suitable for several numerical methods such as Finite Element Method. In addition, *GiD* can be customized and configured by users for the generation of their own solver modules. Therefore, it is possible to set *GeMA* problem type into *GiD* interactive system.

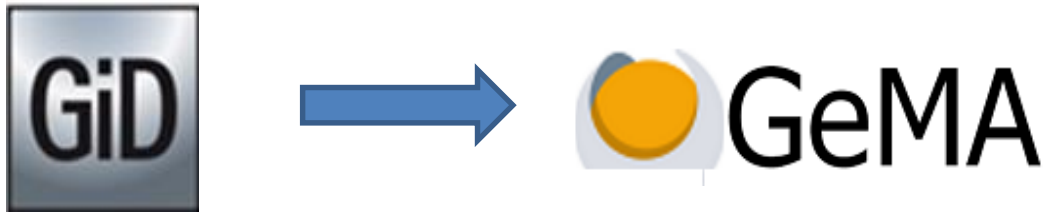
- *GiD-GeMA* is a plugin defined as a preprocessor for *GeMA* Multiphysics framework. Several functionalities can be performed by using this extension, including the creation and attribution of cohesive interface elements into a finite element mesh developed by *GiD*. In addition, the plugin supports three types of model configuration: mechanical, hydraulical e hydro-mechanical (2D and 3D examples). Therefore, it is possible to execute the finite element analysis, especially in cases involving fracture modeling, such as fault reactivation or hydraulic fracturing problems.



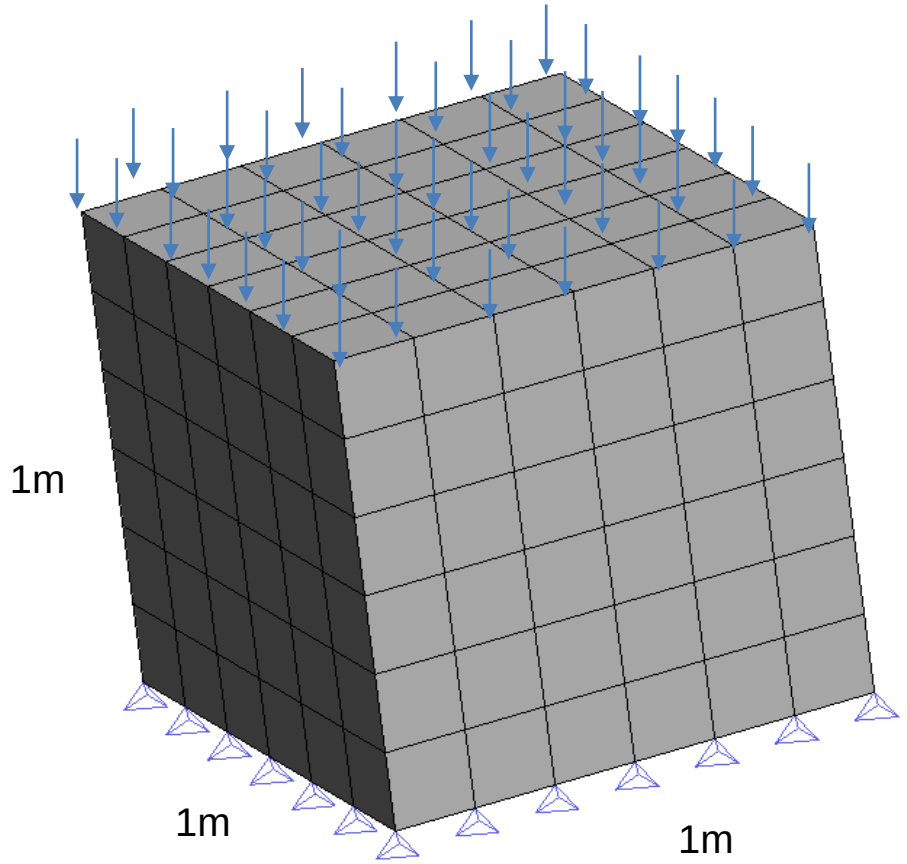
Example set purpose

- In this tutorial, there are described examples for execution of GiD-GeMA plugin. Some of the explanations of these tutorial are related to:

  - How to activate the GeMA problem type in GiD;
  - Understanding the interface platform of GiD-GeMA;
  - Setup material and boundary condition values in GiD-GeMA;
  - Generate mesh and calculate solutions with solver GeMA



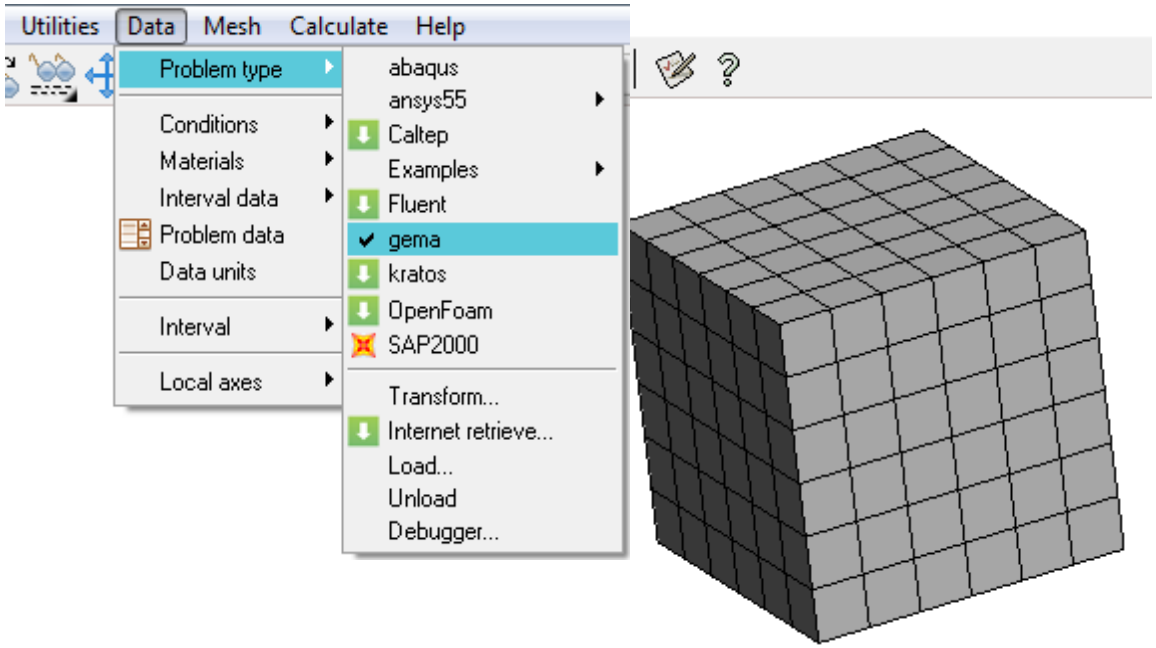
Example 1 – Mechanical Test – 3D



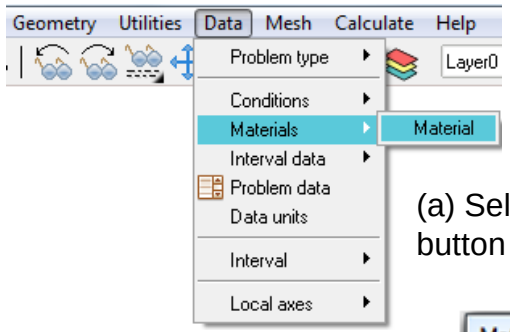
Parameters	Example 1
Young's modulus, E(kPa)	1.00E+06
Poisson's ratio, υ	0.25
Applied pressure load, P (kPa)	1000

Steps – Problem Type and Materials

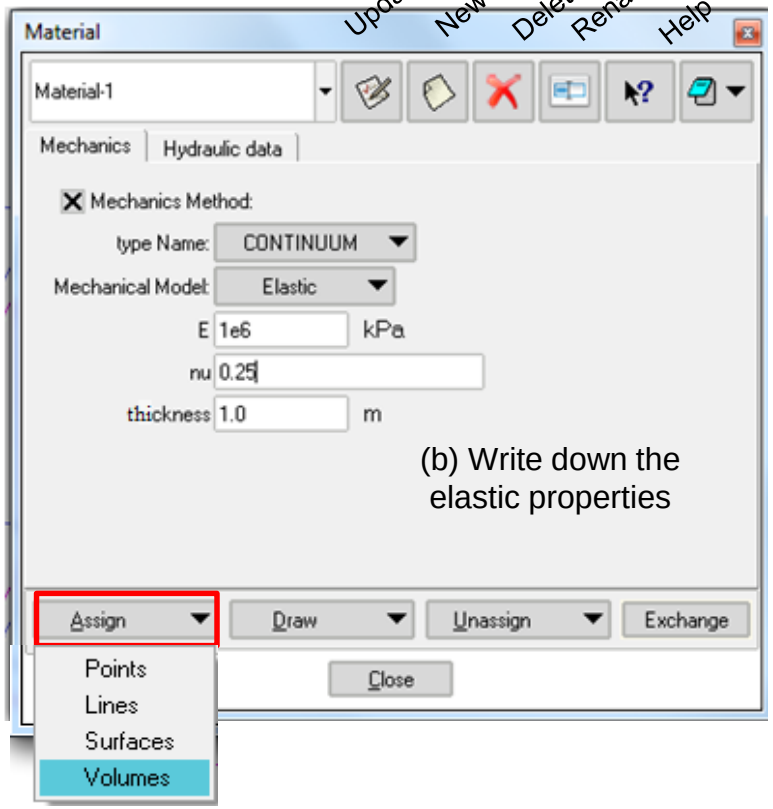
(1) **Problem type**



(2) **Generate Material**



(a) Select the material button in the menu bar

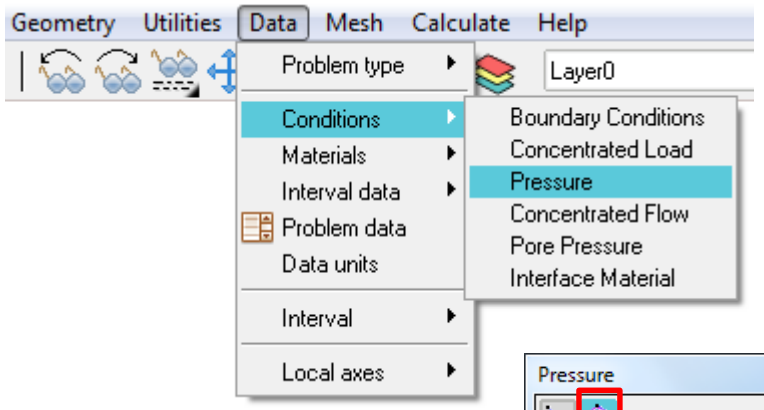


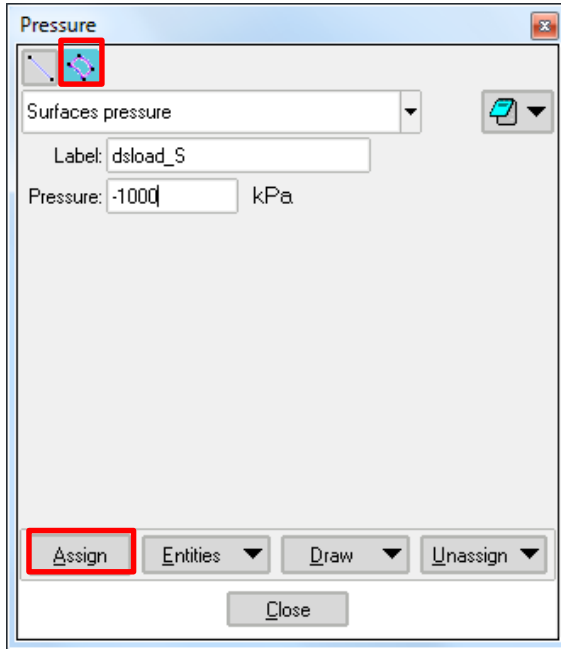
(b) Write down the elastic properties

(c) Click on Assign button and select "Volumes"

Steps – Load

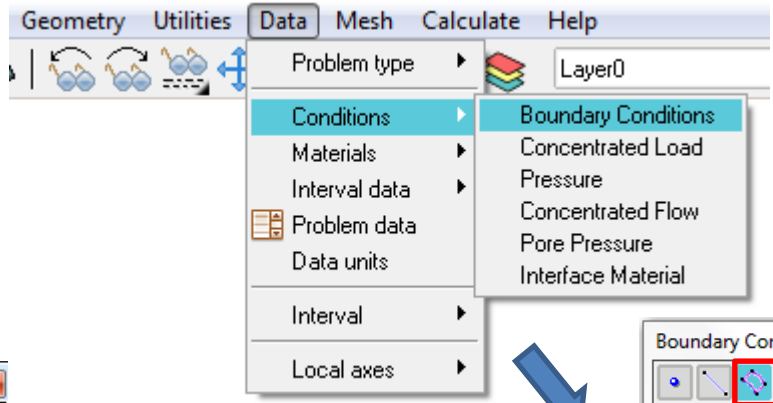
(3) **Pressure Load**

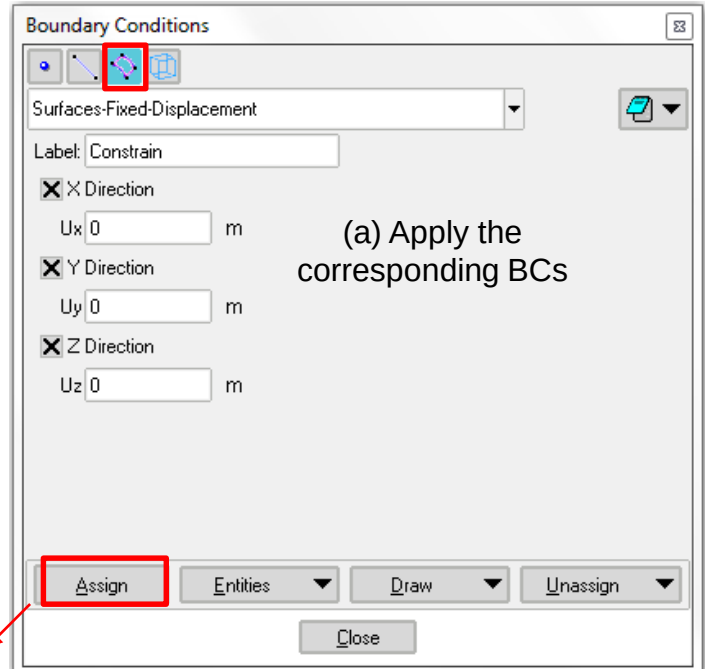




(a) Assign the pressure load in the top surface of the model, selecting the corresponding entities.

(4) **Boundary Conditions**



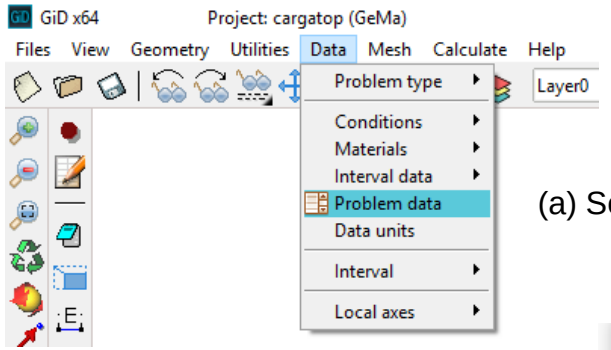


(a) Apply the corresponding BCs

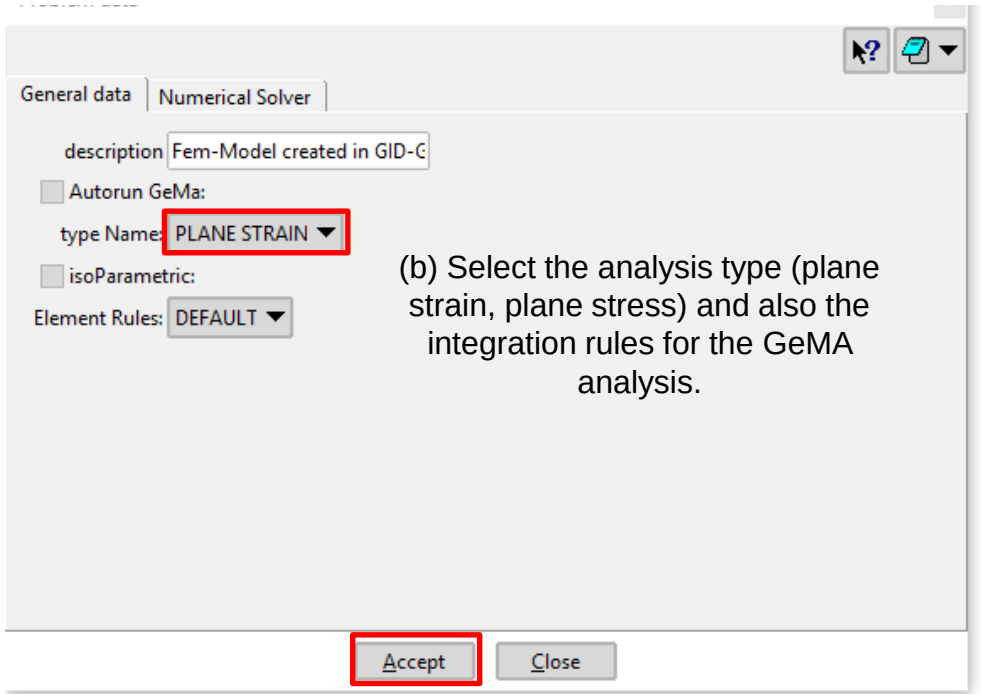
(b) Assign boundary conditions

Steps – General Problem Data

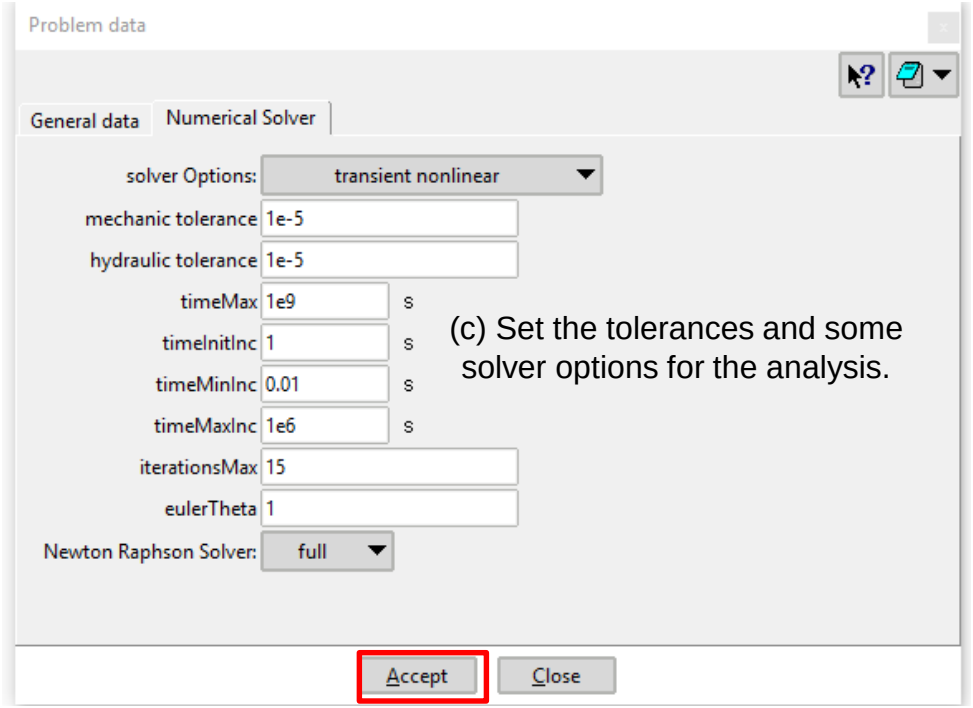
(5) **Set the general problem data**



(a) Select the problem data icon



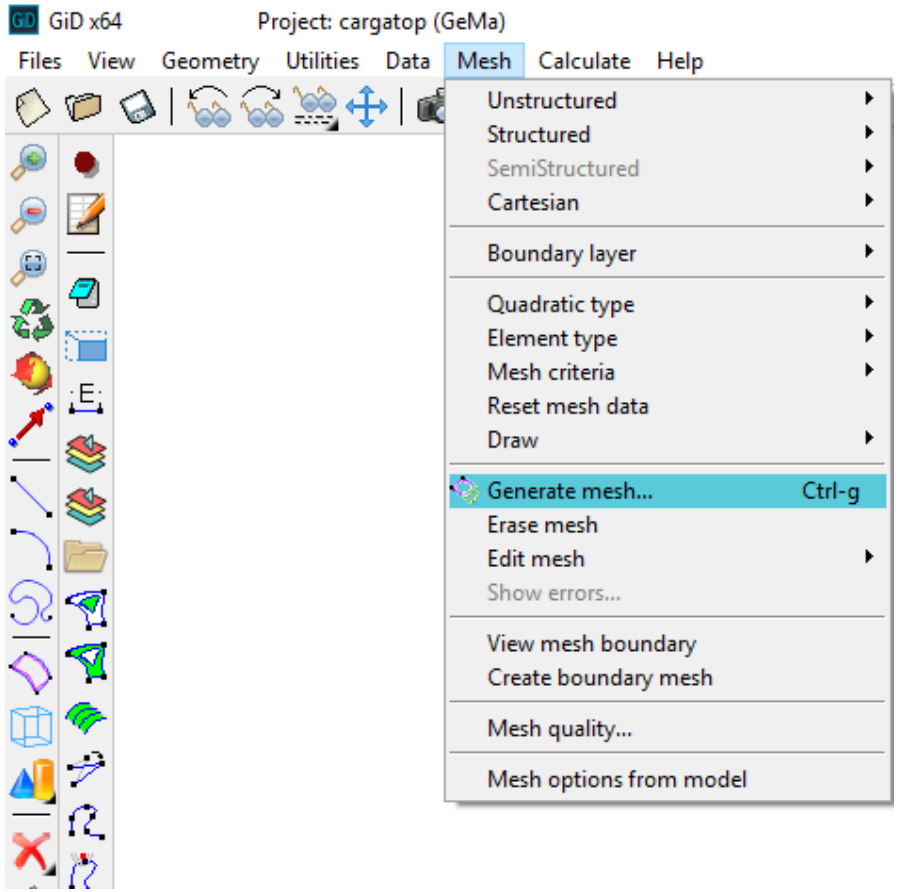
(b) Select the analysis type (plane strain, plane stress) and also the integration rules for the GeMA analysis.

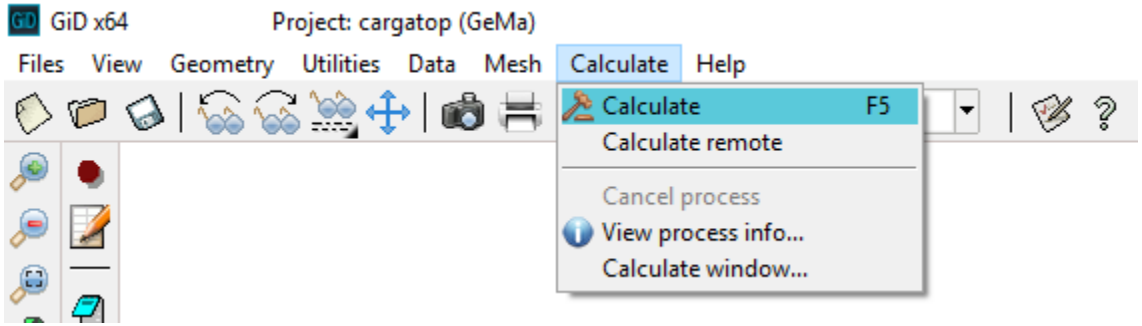


(c) Set the tolerances and some solver options for the analysis.

Steps – Mesh and GeMA input files

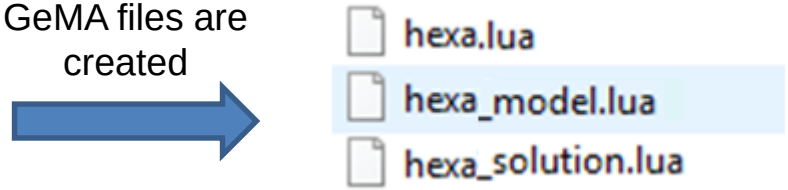
(6) **Generate mesh and GeMA input files**



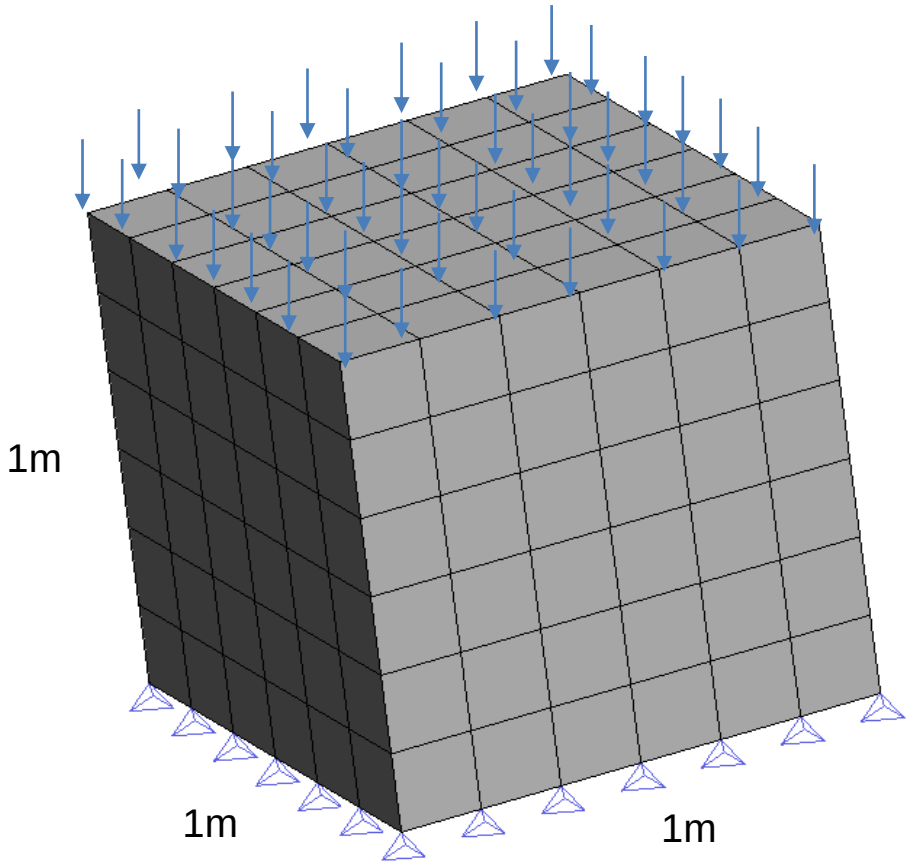


Click on 'Calculate' to generate GeMA input files

GeMA files are created



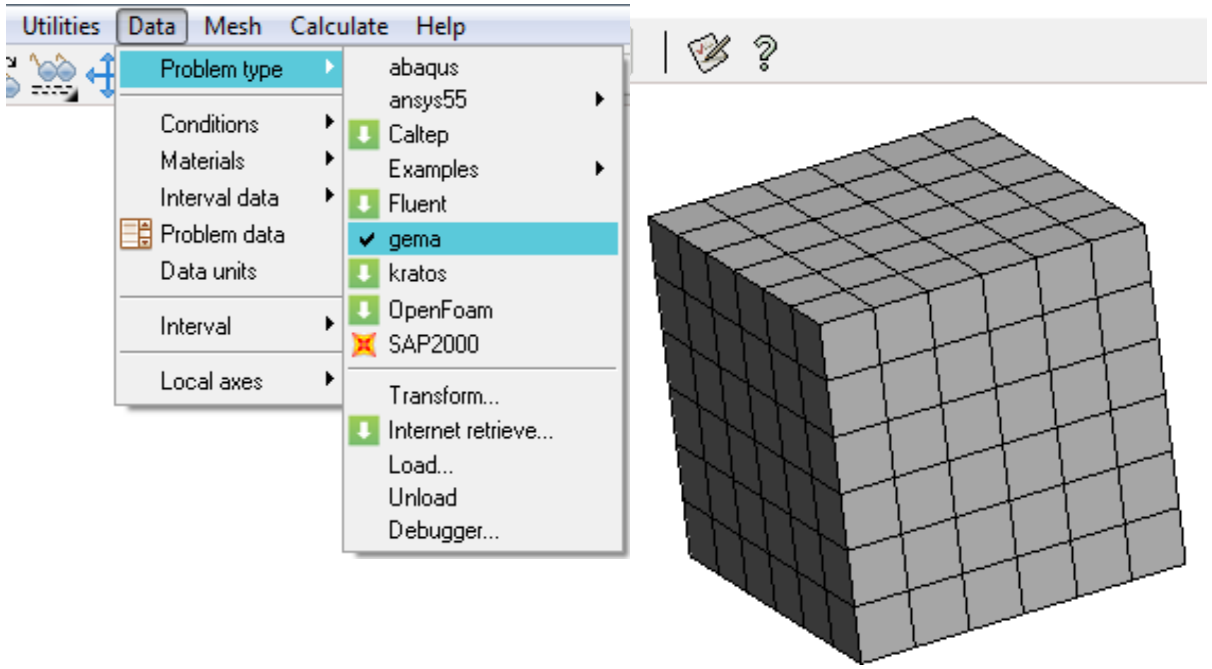
Example 2 – Hydraulical Test – 3D



Parameters	Example 2
Hydraulic Permeability in x, k (m/s)	1.16E-05
Specific weight of water, γ_w (kN/m³)	1.00E+01
Bulk modulus of water, K_{ww} (kPa)	2.20E+30
Porosity, Φ	2.00E-01
Distributed pressure load at top face, P (kPa)	1.00E+01
Pore pressure in the top surface, p (kPa)	1.50E+01

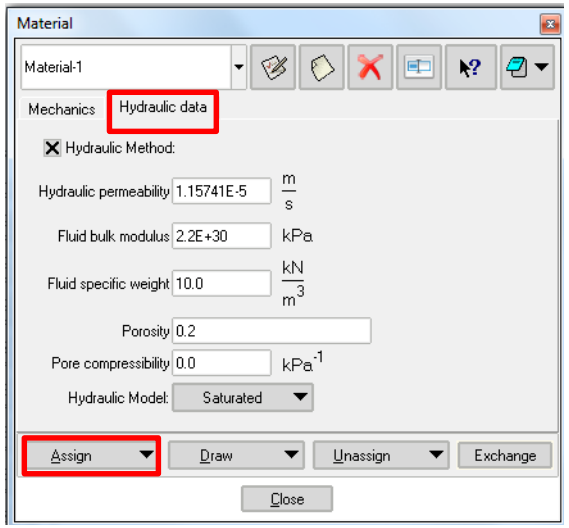
Steps – Problem Type and Materials

(1) **Problem type**



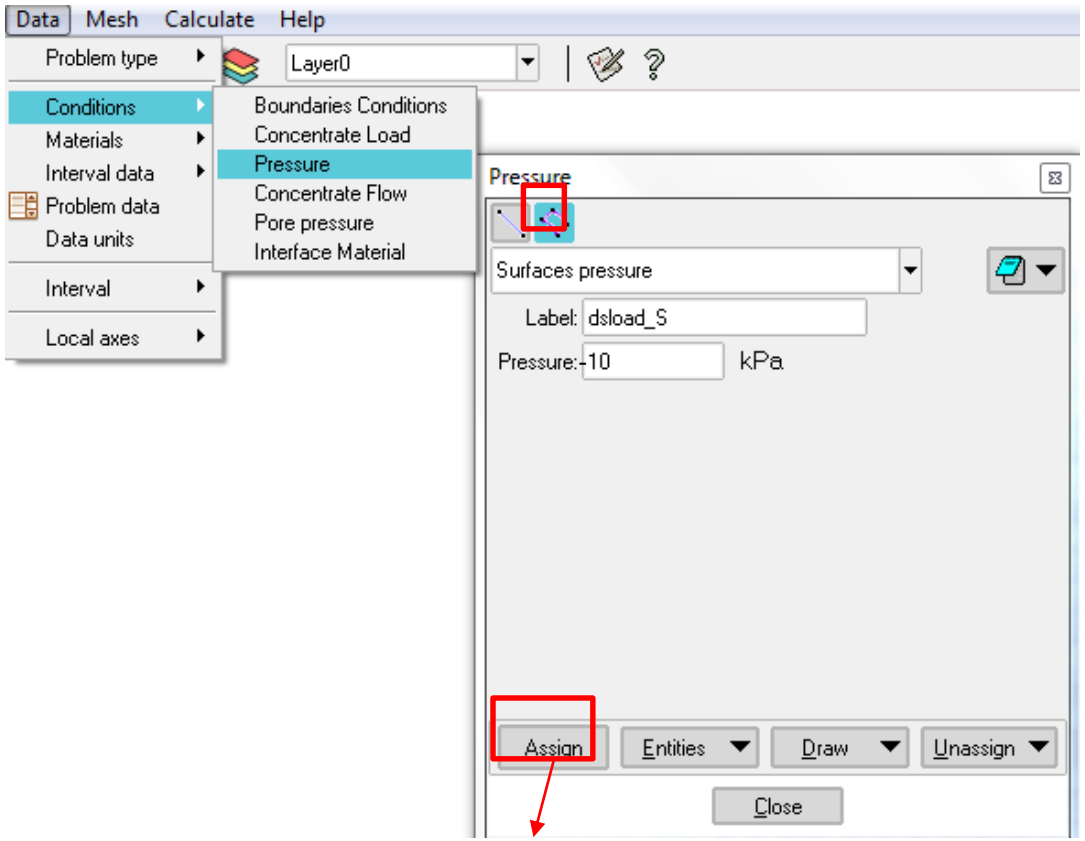
(2) **Generate Material**

(a) Assign hydraulic properties



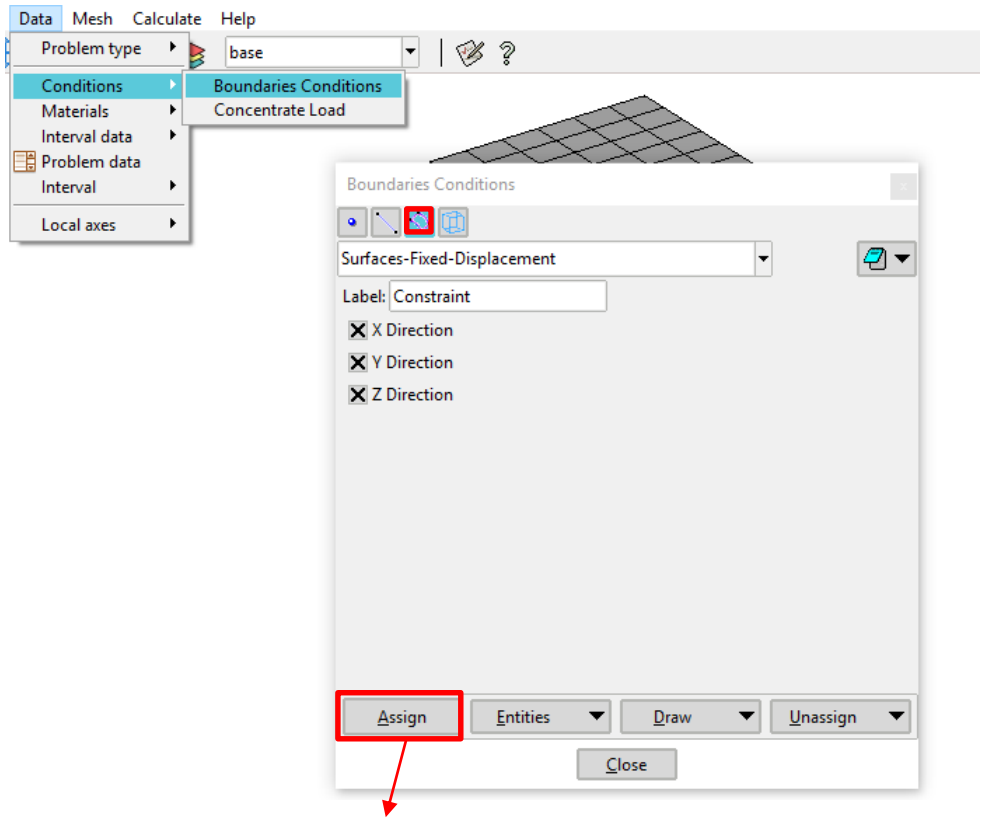
Steps – Pressure Load and Boundary Conditions

(3) **Pressure Load**



(a) Assign pressure load in the top surface

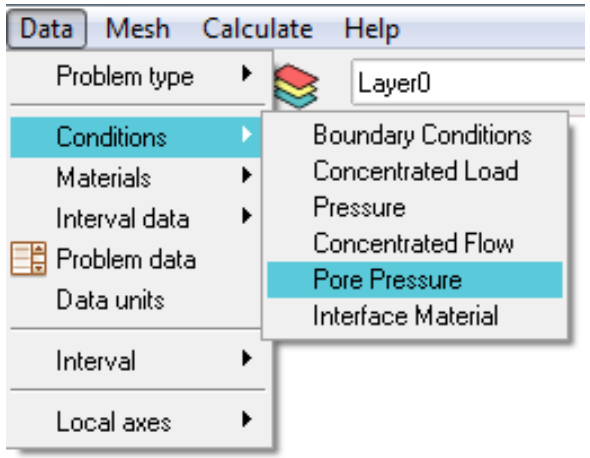
(4) **Boundary Conditions**



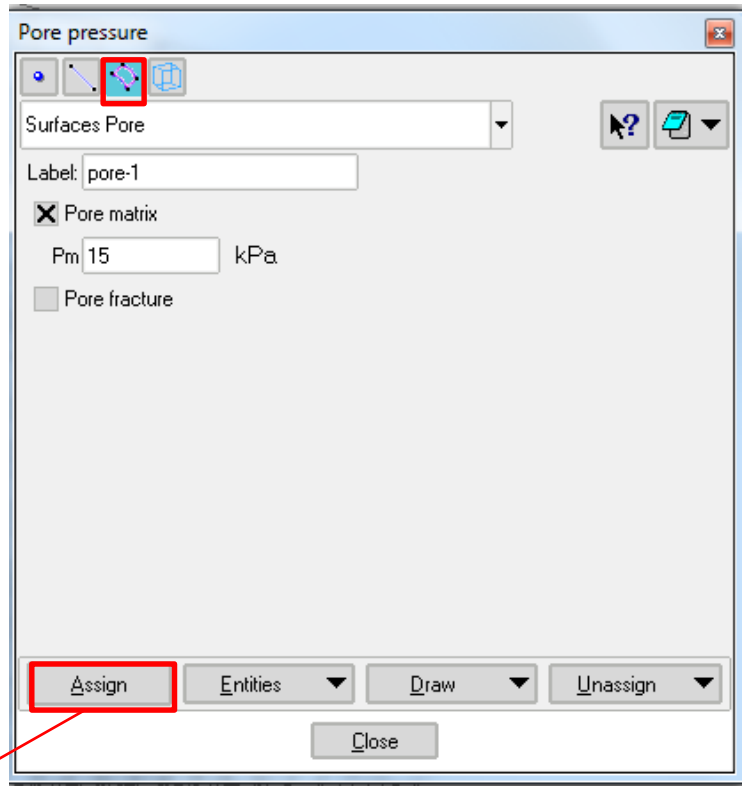
(a) Assign boundary conditions

Steps – Pore Pressure

(5) **Apply pore pressure in the top surface**



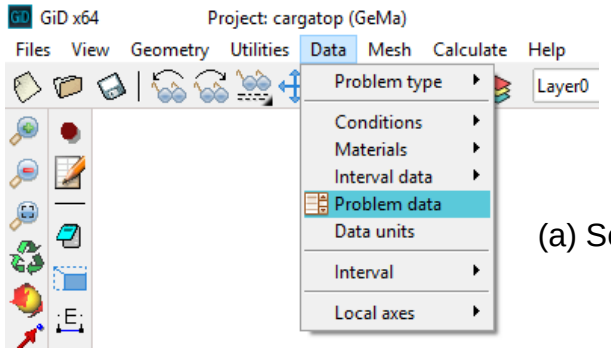
(a) Active the “Pore matrix” label and write down values of pore-pressure (in kPa)



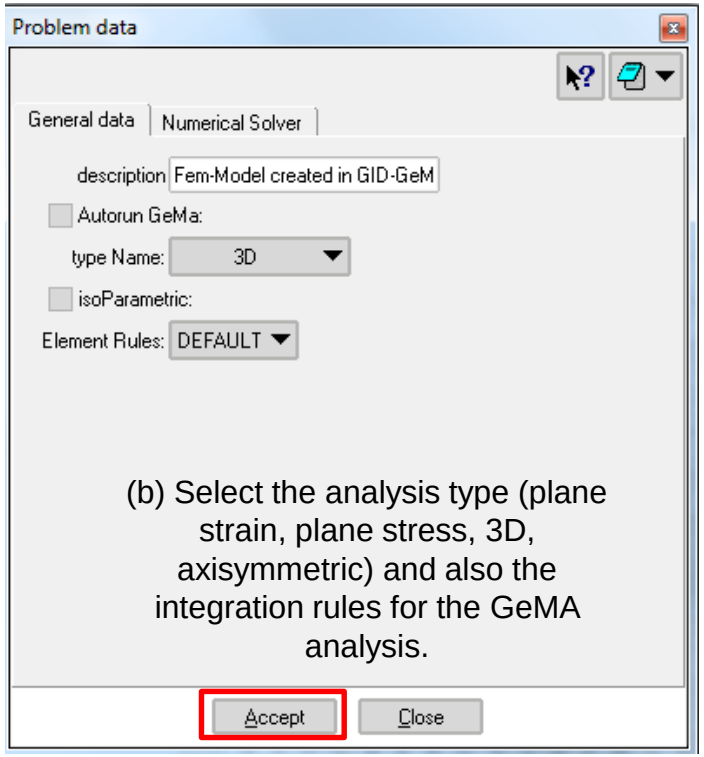
(b) Assign the pore pressure values in the correct region of the model, selecting the corresponding surface

Steps – General Problem Data

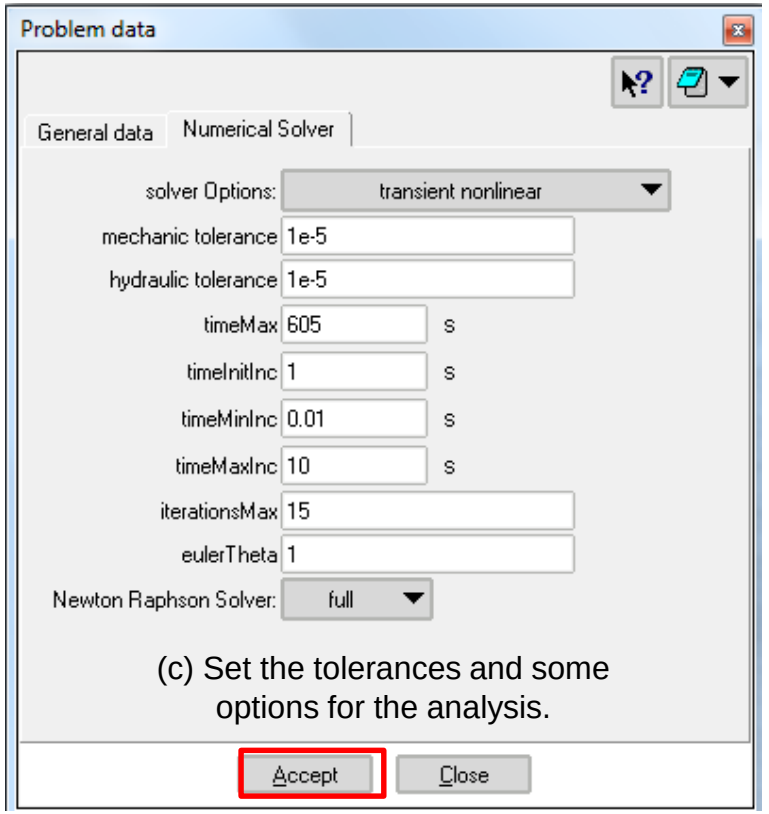
(6) **Set the general problem data**



(a) Select the problem data icon



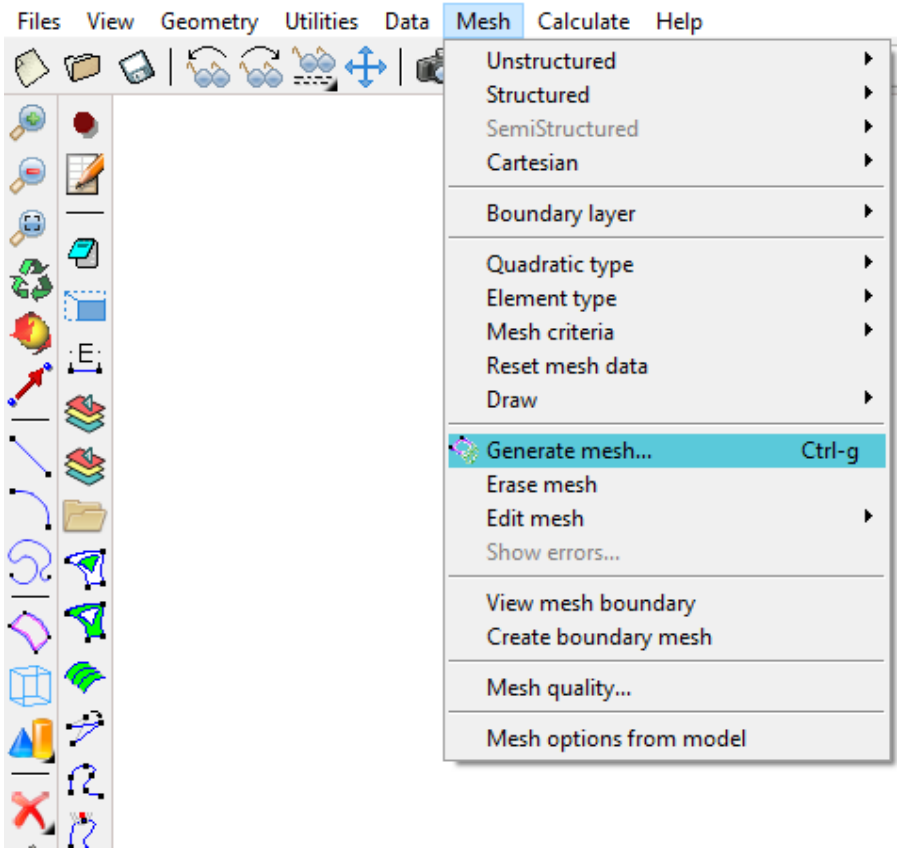
(b) Select the analysis type (plane strain, plane stress, 3D, axisymmetric) and also the integration rules for the GeMA analysis.

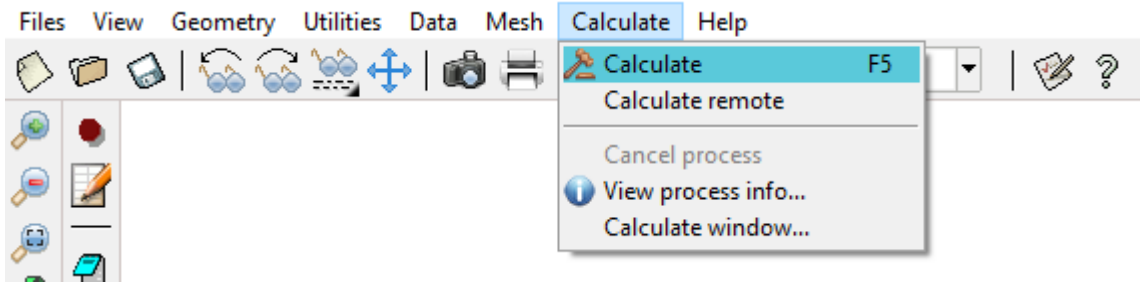


(c) Set the tolerances and some options for the analysis.

Steps – Mesh

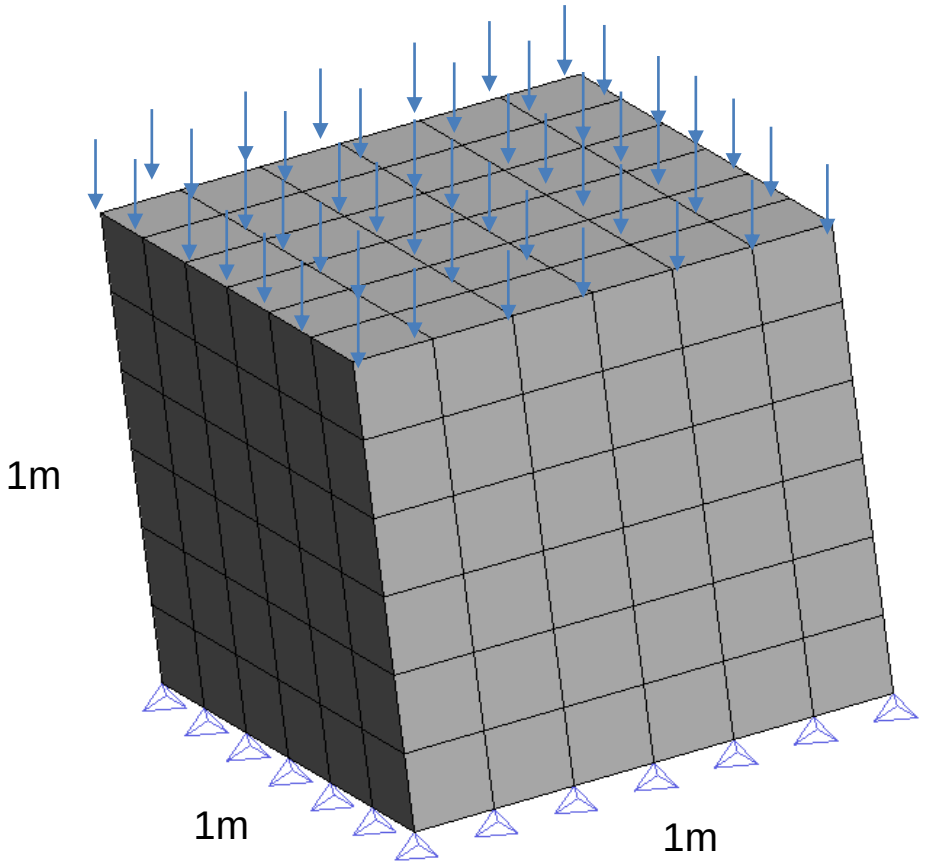
(7) **Generate mesh and GeMA input files**





Click on ‘Calculate’ to generate GeMA input files.

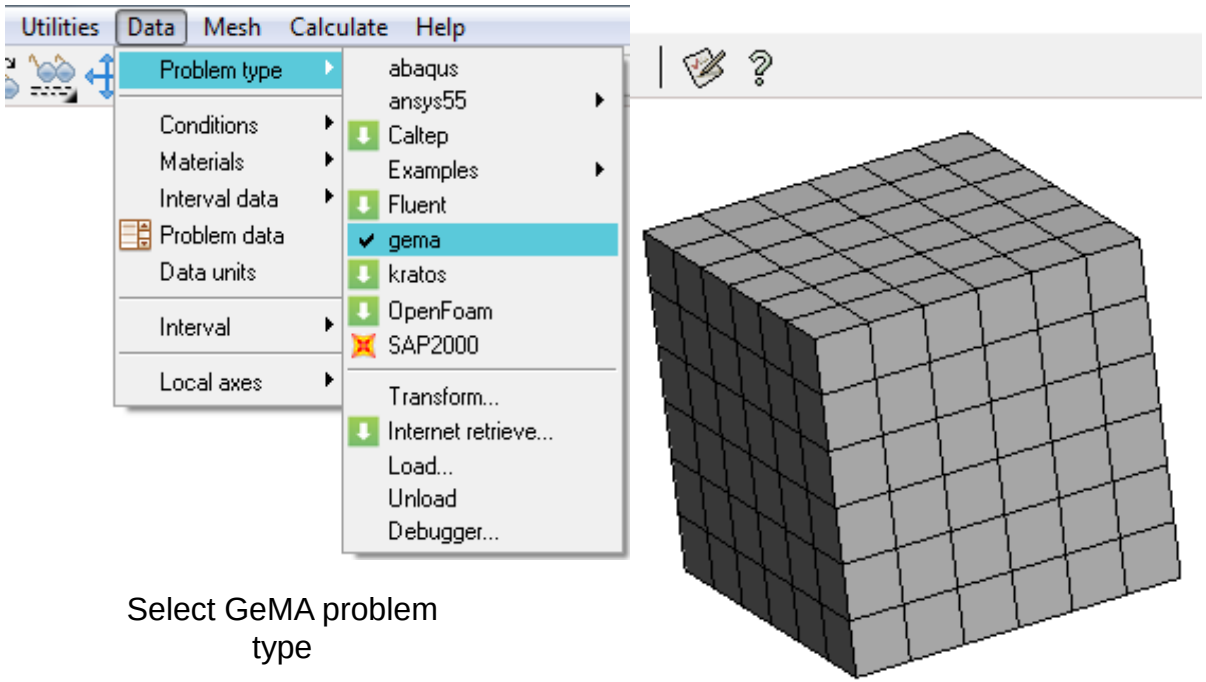
Example 3 – Hydro-Mechanical Test – 3D



Parameters	Example 3
Young's modulus, E(kPa)	1.00E+03
Poisson's ratio, υ	0.3
Hydraulic Permeability in x, k (m/s)	1.16E-05
Specific weight of water, γ_w (kN/m³)	1.00E+01
Bulk modulus of water, K_{ww} (kPa)	2.20E+30
Porosity, Φ	2.00E-01
Distributed pressure load at top face, P (kPa)	1.00E+01

Steps – Problem Type and Materials

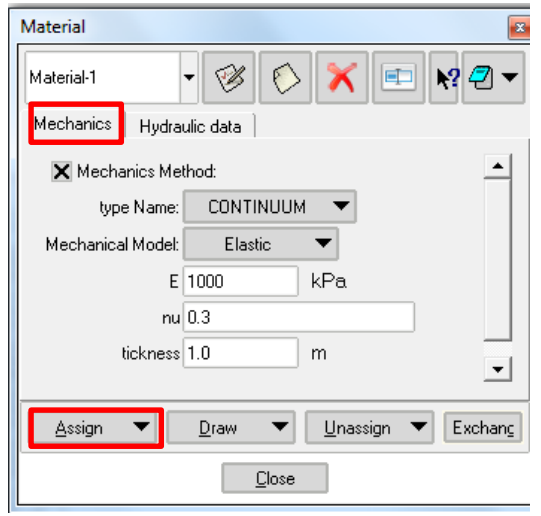
(1) **Problem type**



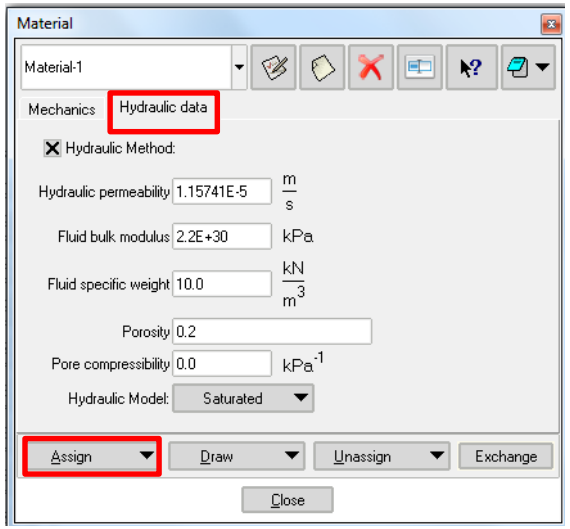
Select GeMA problem type

(2) **Generate Material**

(a) Assign mechanical properties

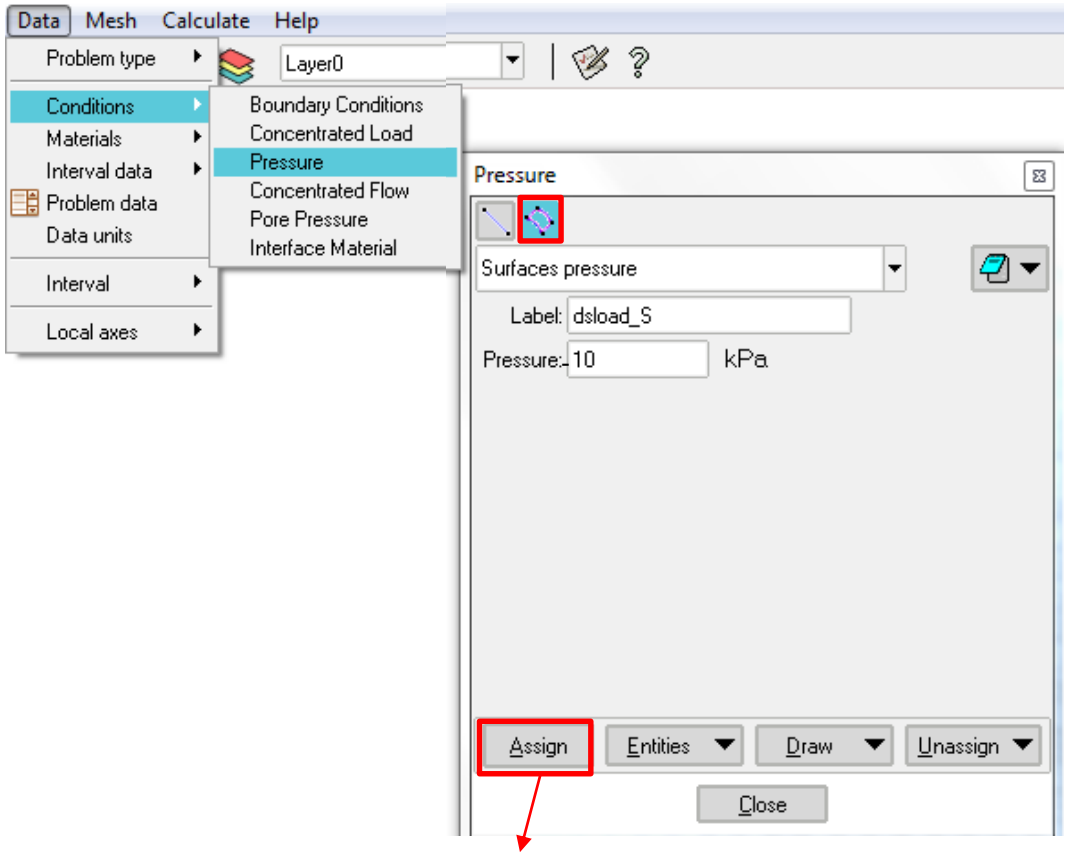


(b) Assign hydraulic properties



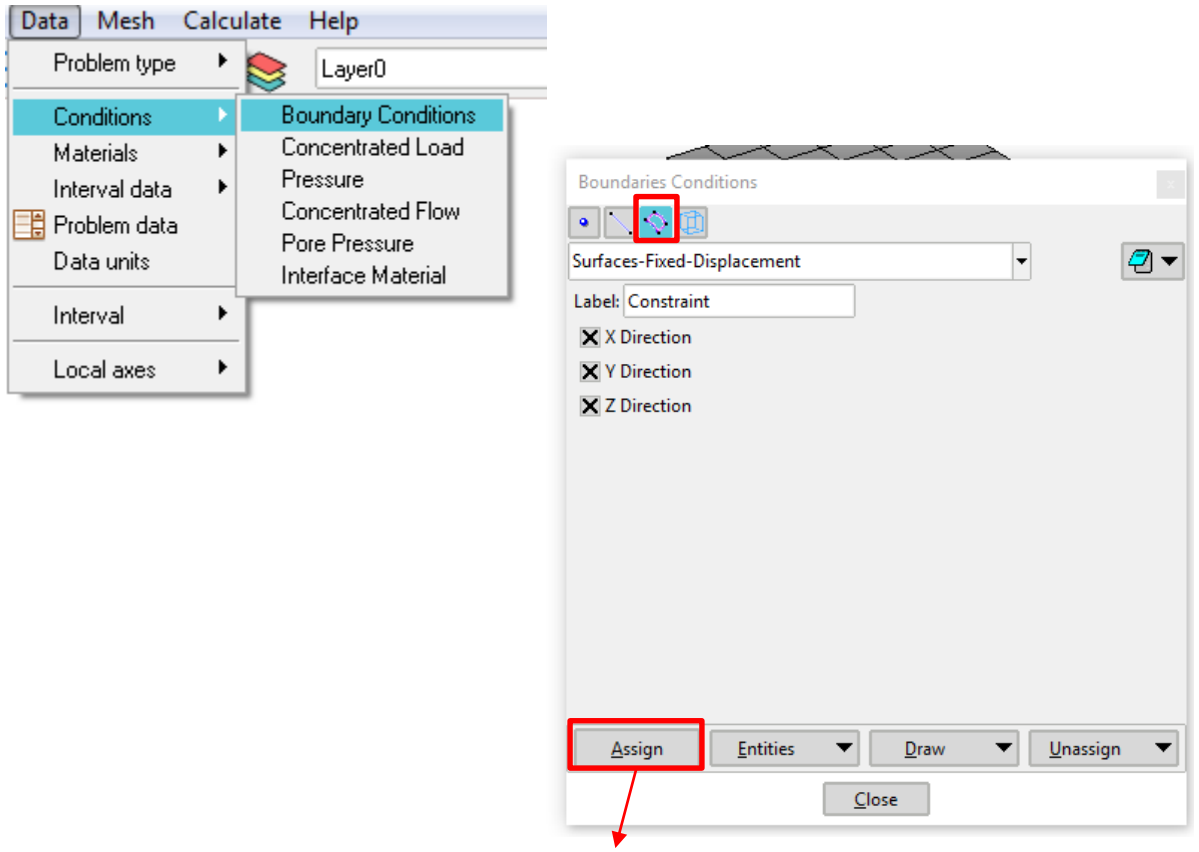
Steps – Pressure Load and Boundary Conditions

(3) **Pressure Load**



(a) Assign pressure load in the top surface

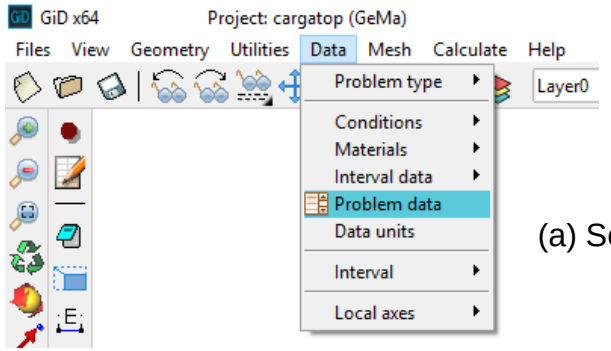
(4) **Boundary Conditions**



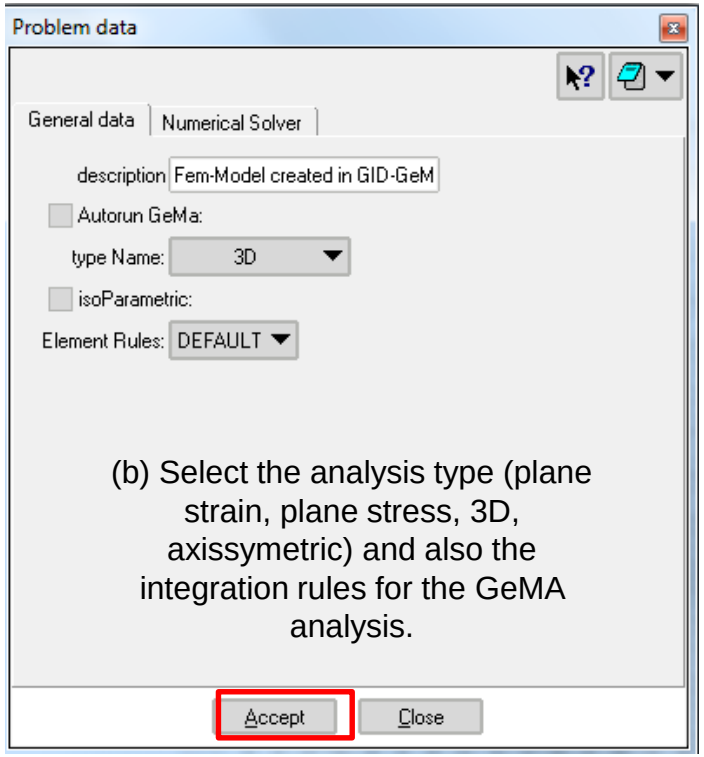
(a) Assign boundary conditions

Steps – General Problem Data

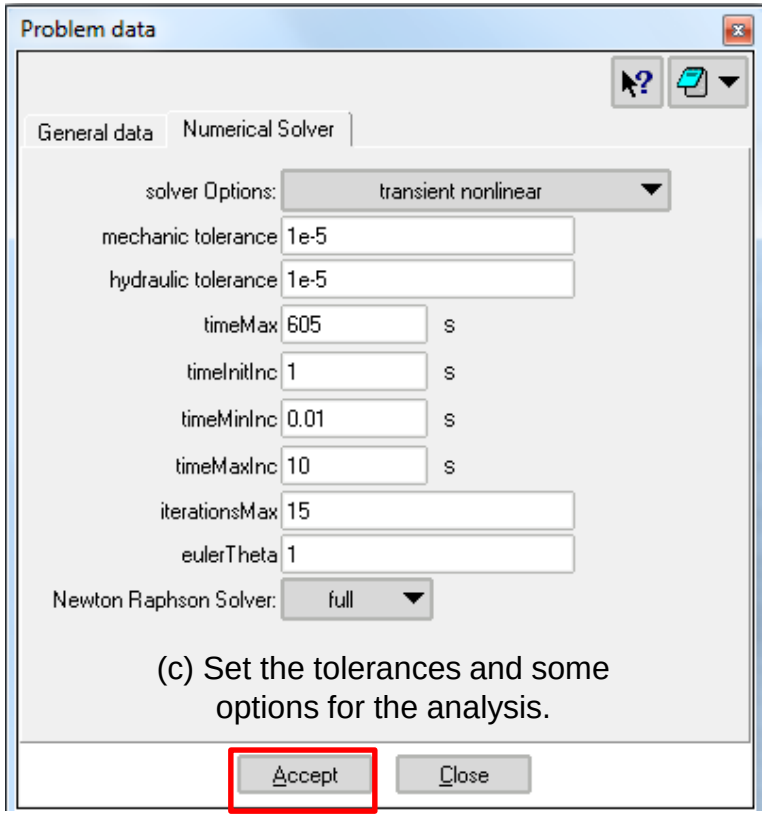
(5) **Set the general problem data**



(a) Select the problem data icon



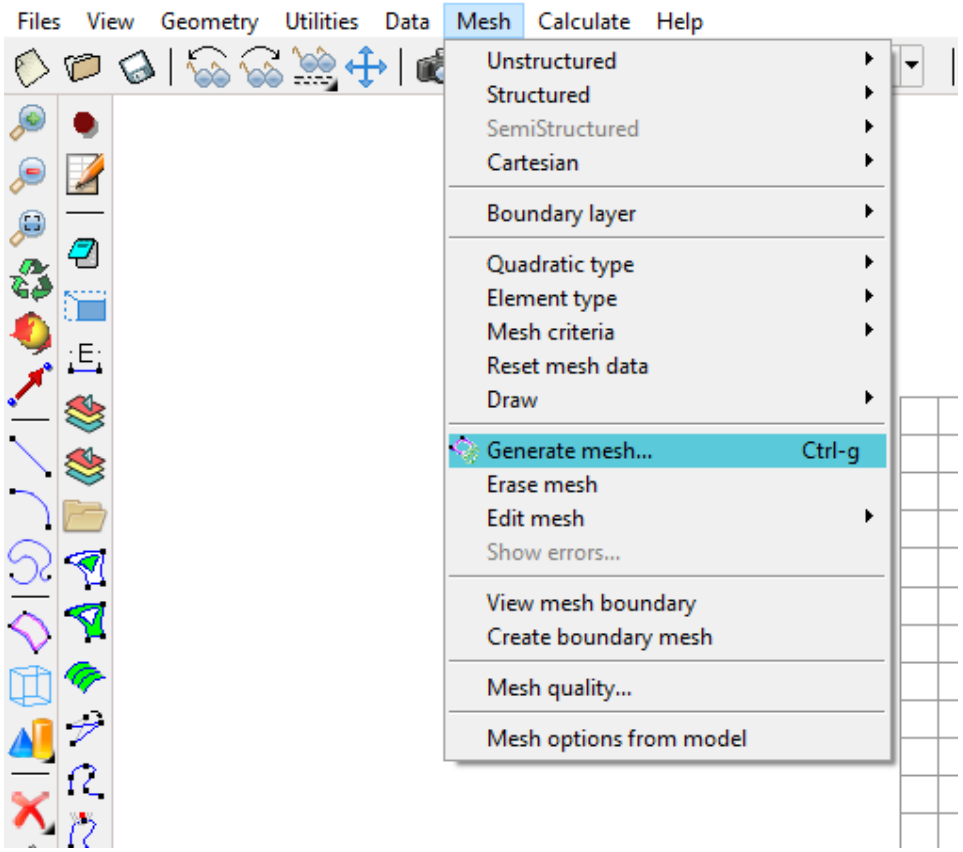
(b) Select the analysis type (plane strain, plane stress, 3D, axissymetric) and also the integration rules for the GeMA analysis.

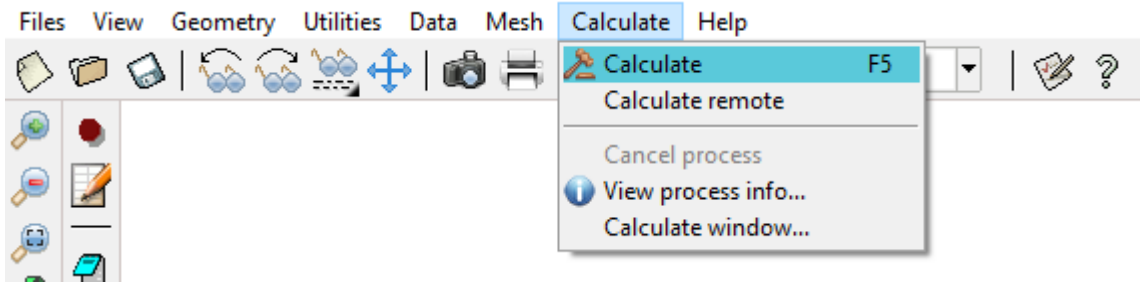


(c) Set the tolerances and some options for the analysis.

Steps – Mesh

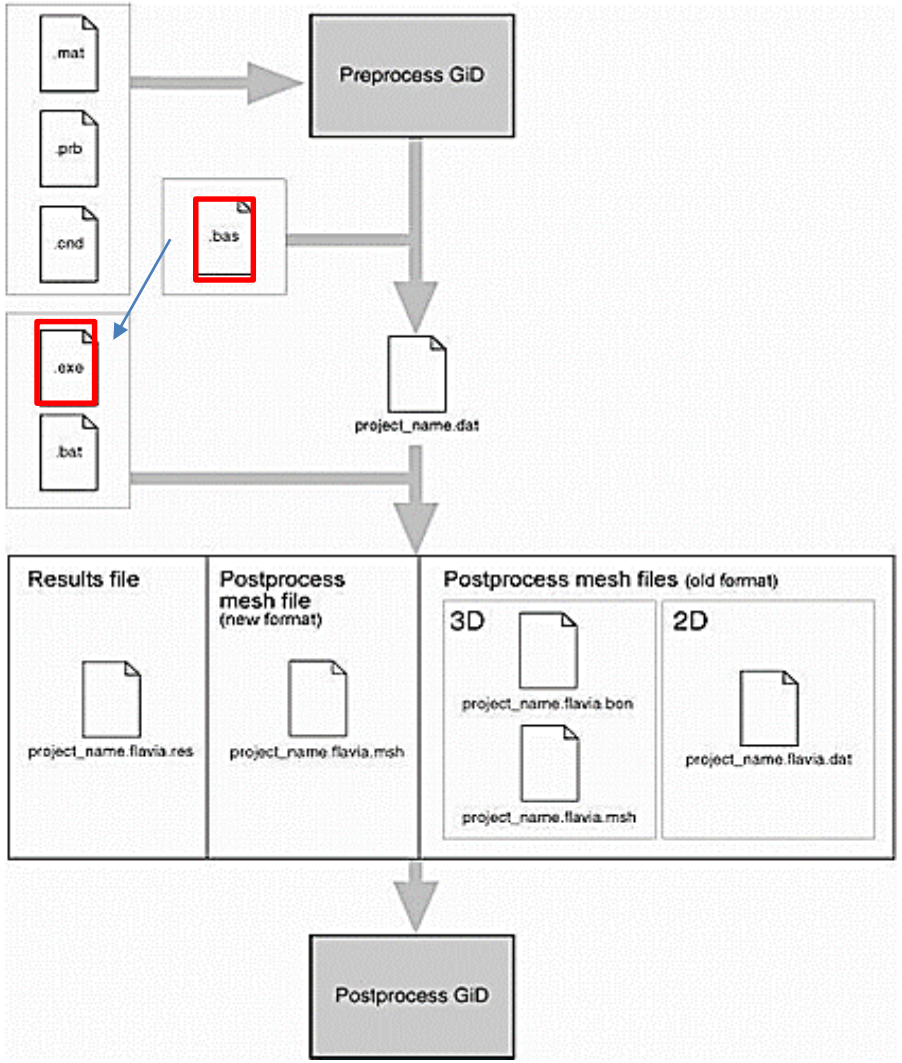
(6) **Generate mesh and GeMA input files**





Click on ‘Calculate’ to run the analysis.

Gid-GeMa including interface elements



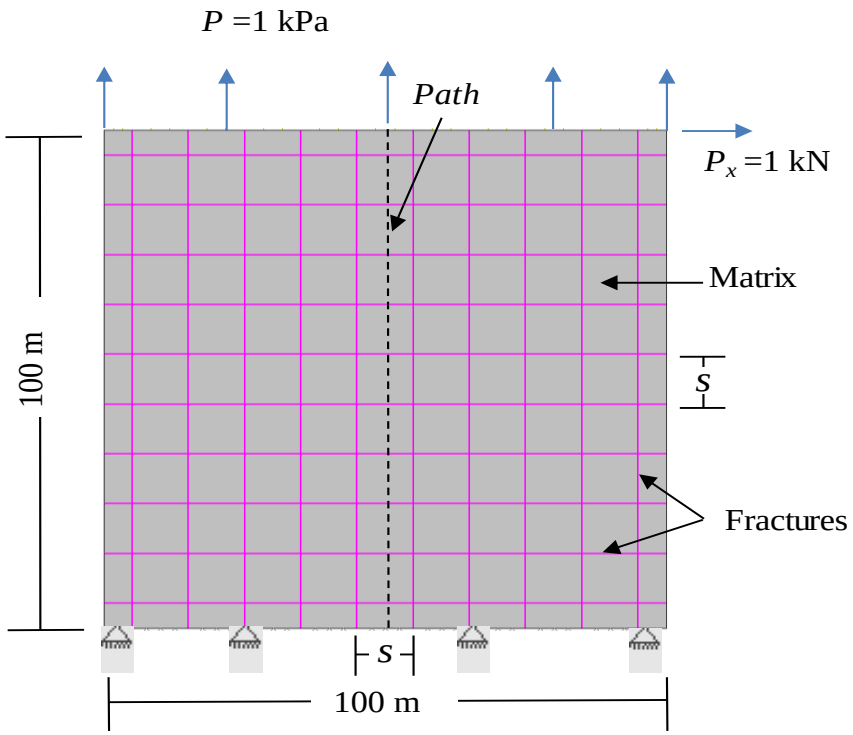

```
%GENERAL_DATA
   NPOIN    NELEM    NMATS    NDIME  NNOMAX  NELINT
    1086      362        3        2       3     515

%FACE_DATA
     iface    Lmat Lcodeh Lcodem Lcodet nnoface nnofacec ngauss    no(1)     no(2)
         1       3       1       4       1       2       2       2      594       604


%NODES
        ID                  X                   Y                   Z
         1           -5.243260           -2.394710            0.000000

%ELEMENTS
    6400       8
   Ielem     Conectivities        Lmat  Nnom Nnof
       1   16724   16770   16772   16741   16749   16774   16775   16761     1     8     8
```

```
-------------------------------------------------------------------------------
|                                                                             |
|                    TECGRAF/COMPUTATIONAL GEOMECHANICS GROUP                 |
|                                  GID2INT                                    |
|                Code for creation of interface elements in GID               |
|                                                                             |
-------------------------------------------------------------------------------
please enter the input file name *.int (include extension):
```

IntMesh.lua

File with the created interface elements

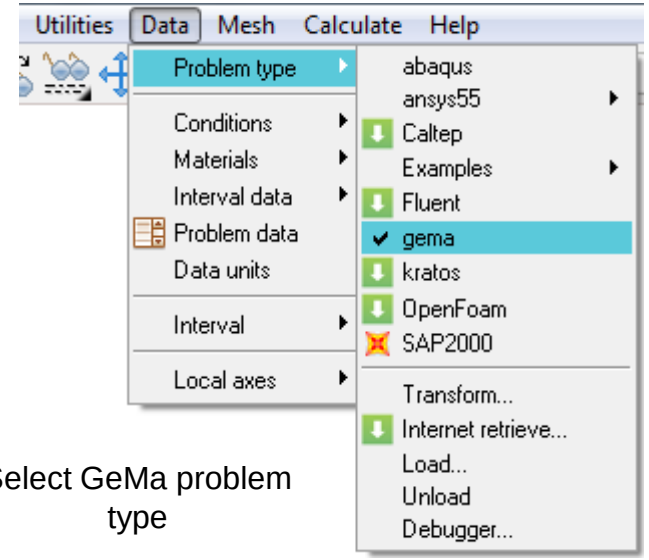
Example 4 – Mechanical Test – 2D



Parameters	Example 4
Young's modulus, E(kPa)	1.00E+06
Poisson's ratio, υ	0.2
Normal stiffness of fracture, k_n (kPa/m)	1.00E+05
Tangential stiffness of fracture, k_s(kPa/m)	1.00E+05
Fracture spacing, s(m)	10.0

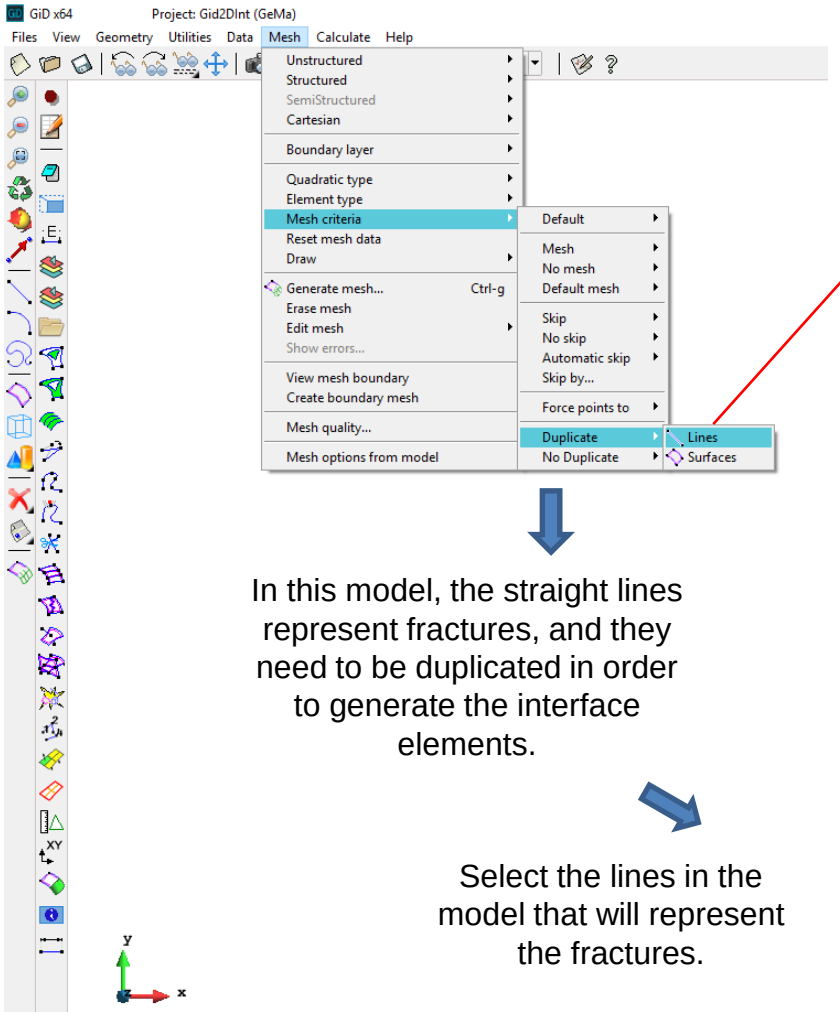
Steps – Problem Type

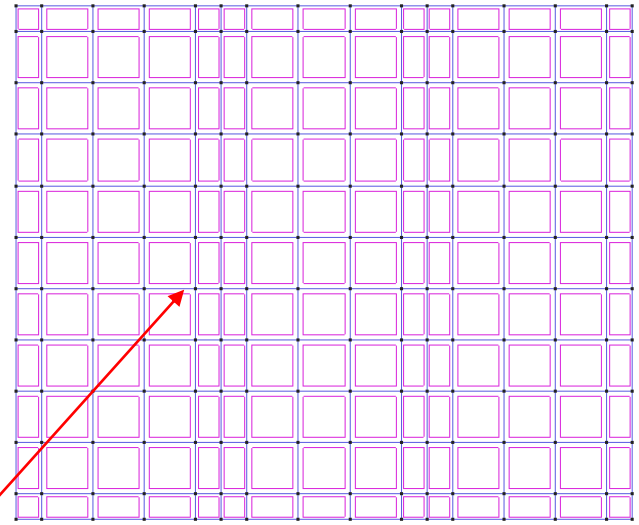
(1) **Set the GeMA problem type**



Select GeMa problem type

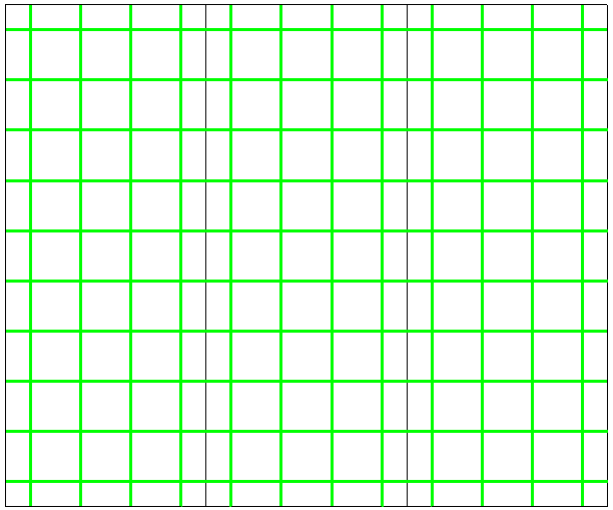
(2) **Duplicate the nodes from lines**





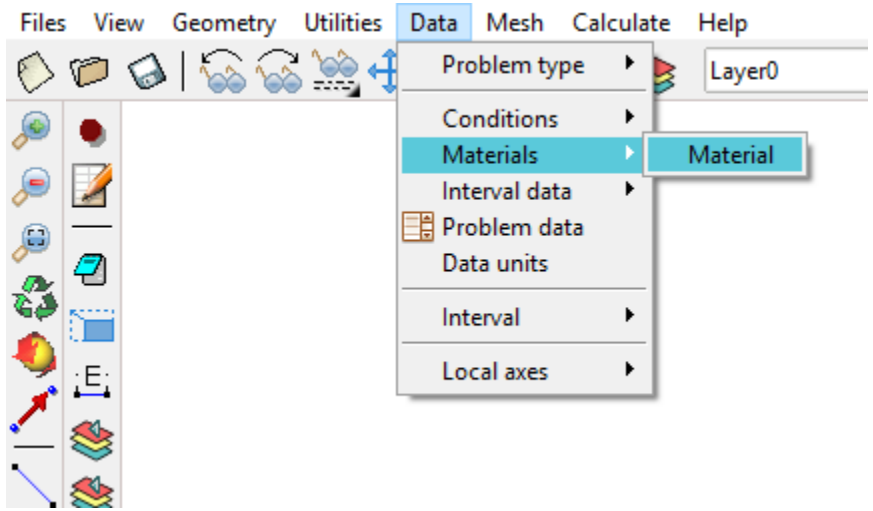
In this model, the straight lines represent fractures, and they need to be duplicated in order to generate the interface elements.

Select the lines in the model that will represent the fractures.



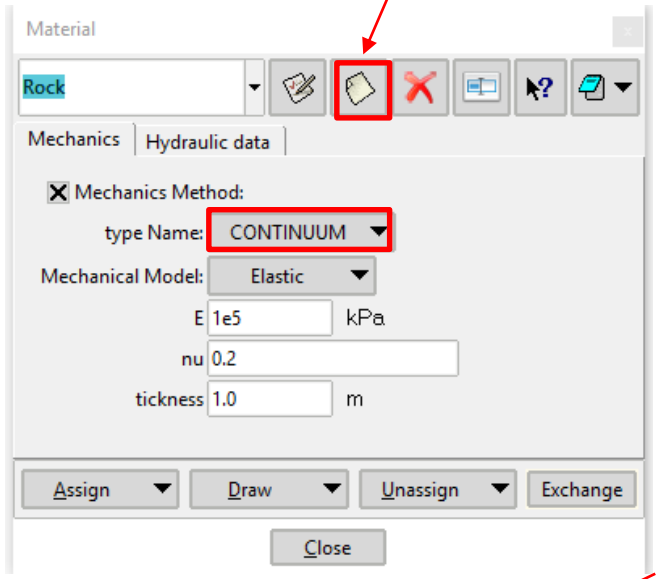
Steps – Material

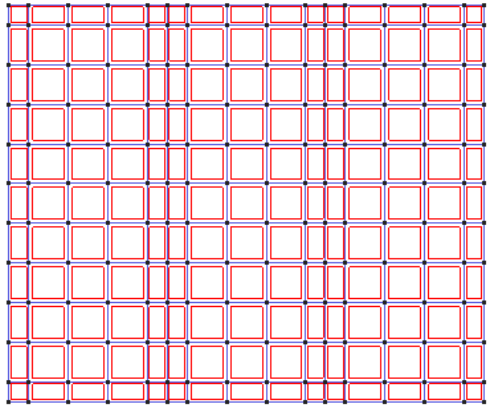
(3) **Generate material**



(a) Create a new material for the solid elements

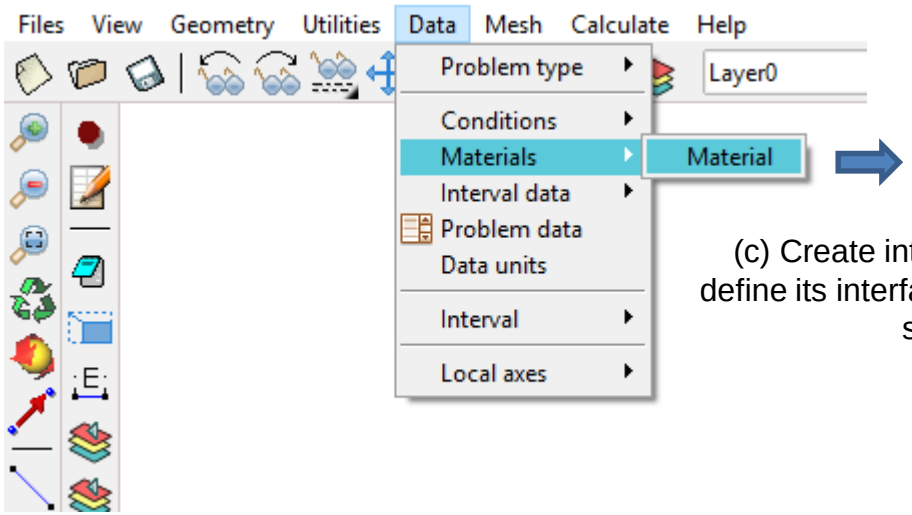
Create new material





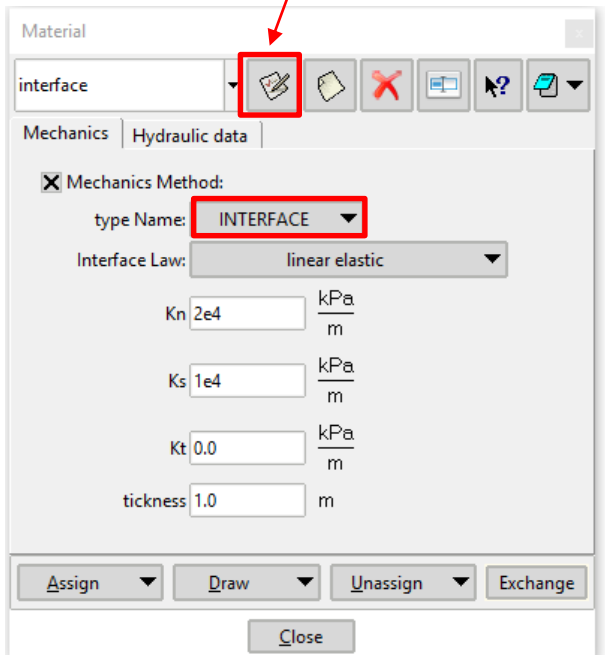
Steps – Material

(3) **Generate material**



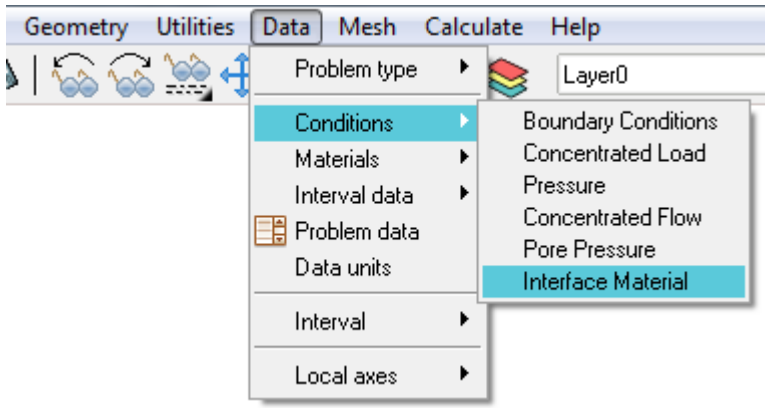
(c) Create interface material, define its interface properties and save

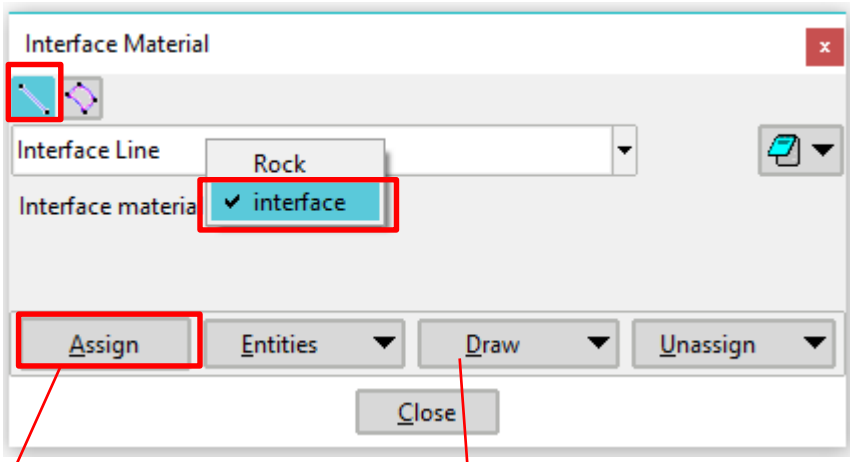
Create interface material and save it



Steps – Conditions

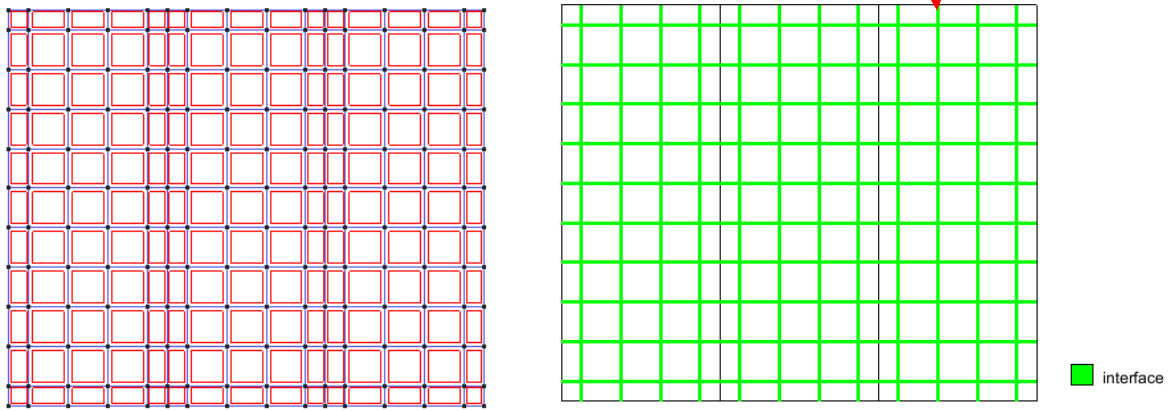
(4) **Attach conditions for the interface material**





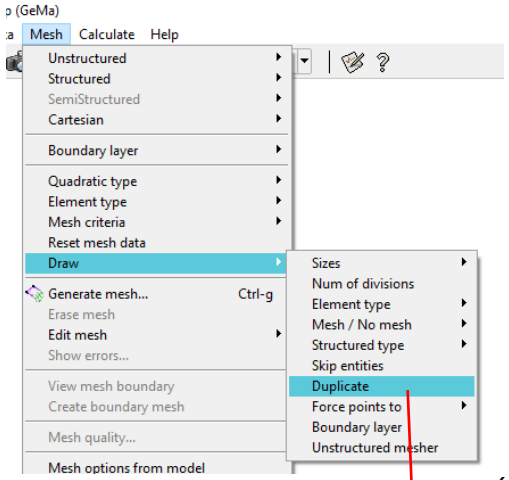
(a) Assign the interface material into the lines

(b) Verify if the attachment was done correctly

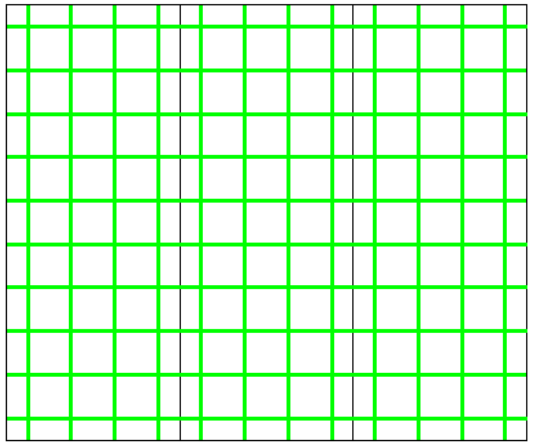


Steps – Conditions

(4) **Attach conditions for the interface material**

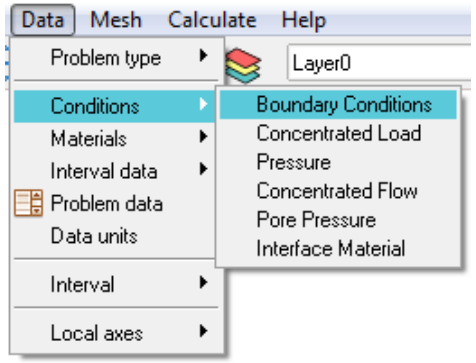


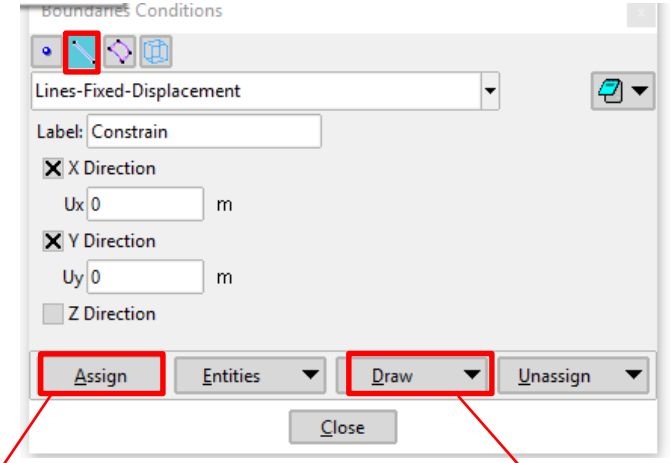
(c) Duplicate interface material



Steps – Constraints

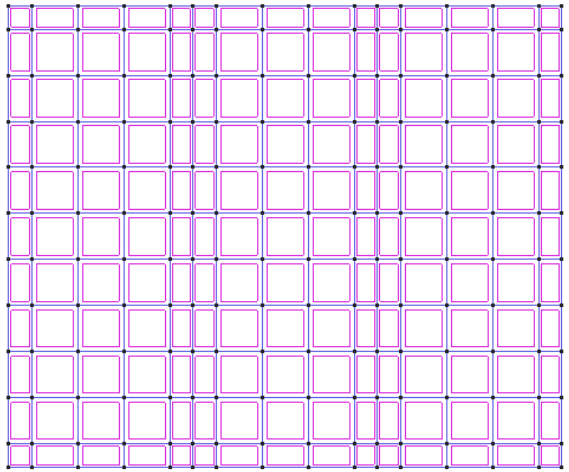
(5) **Apply the boundary conditions of the problem**

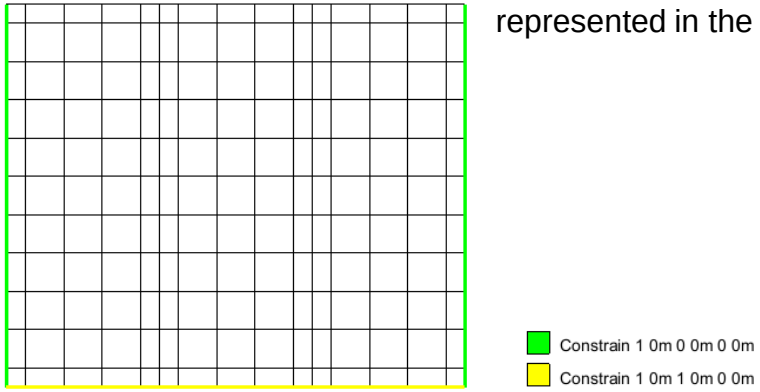




(a) Assign boundary conditions

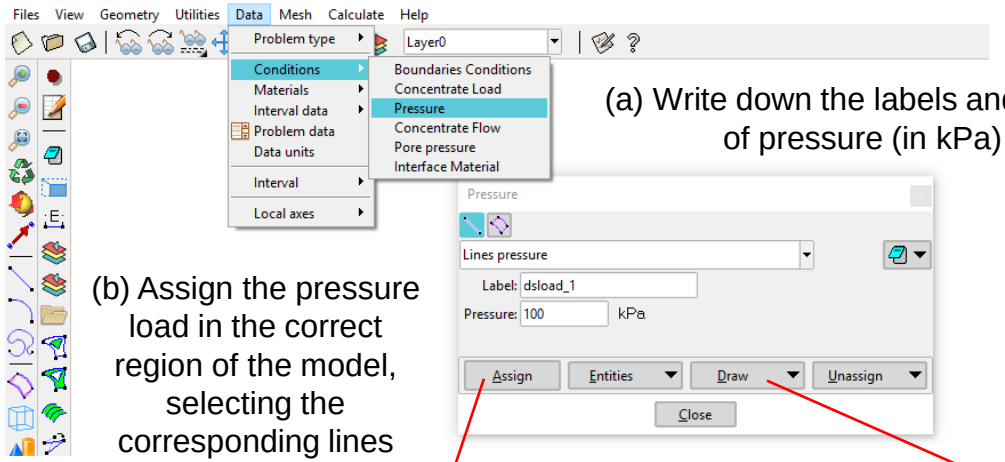
(b) Verify if the boundary conditions were correctly represented in the model





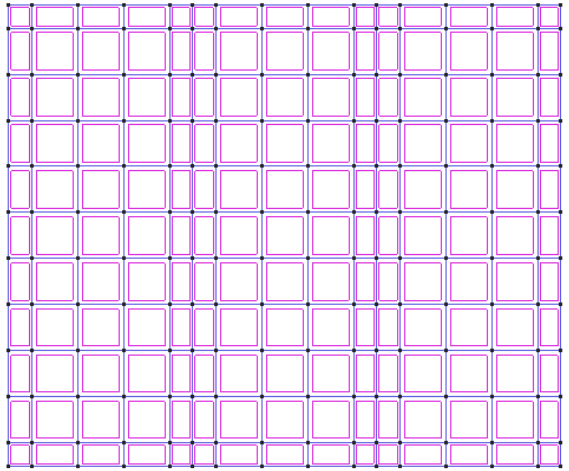
Steps – Pressure Load

(6) **Apply pressure load**

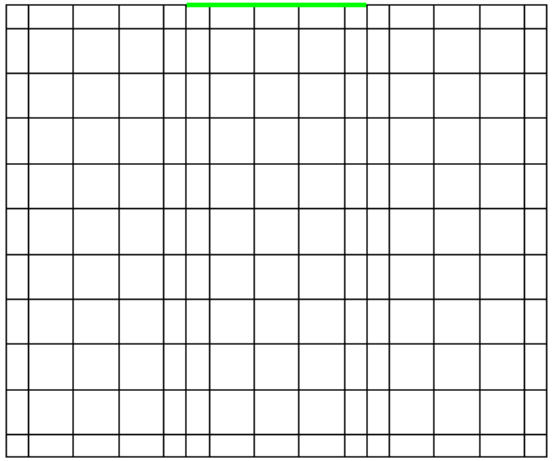


(a) Write down the labels and values of pressure (in kPa)

(b) Assign the pressure load in the correct region of the model, selecting the corresponding lines

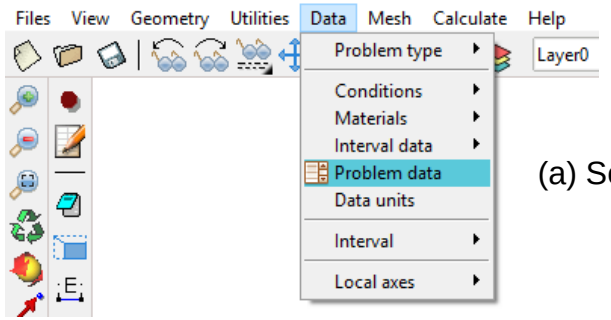


(c) Verify if the pressure load was correctly incorporated to the model

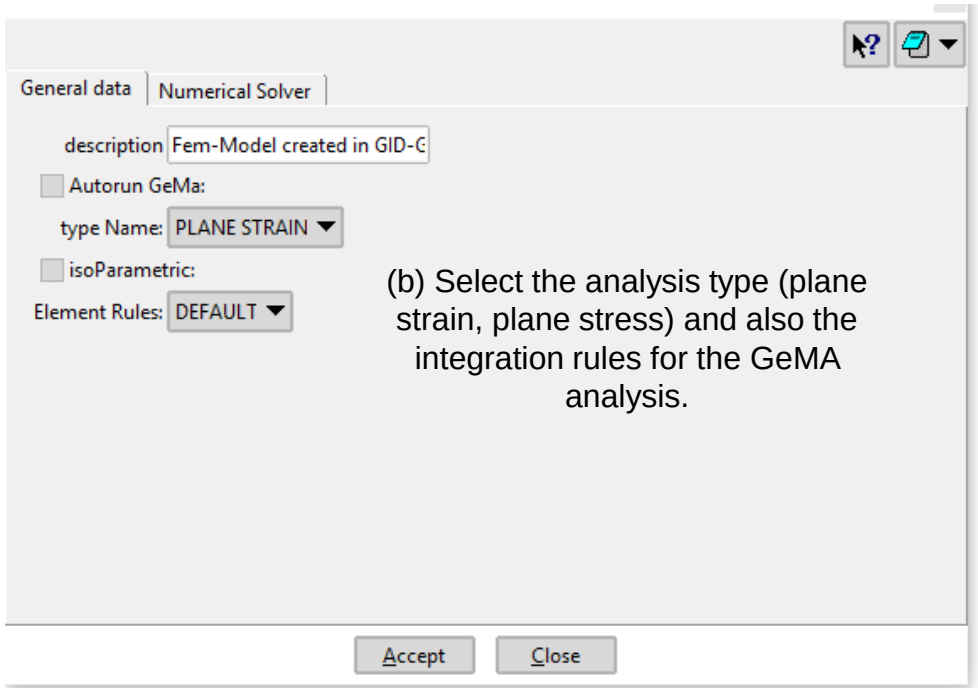


Steps – General Problem Data

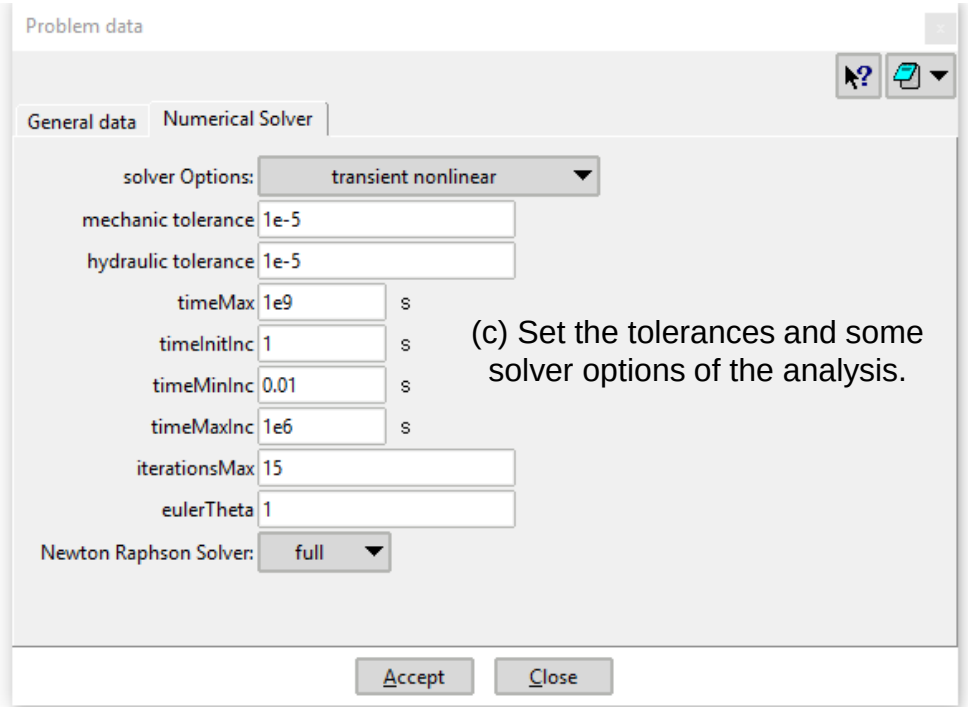
(7) **Set the general problem data**



(a) Select the problem data icon



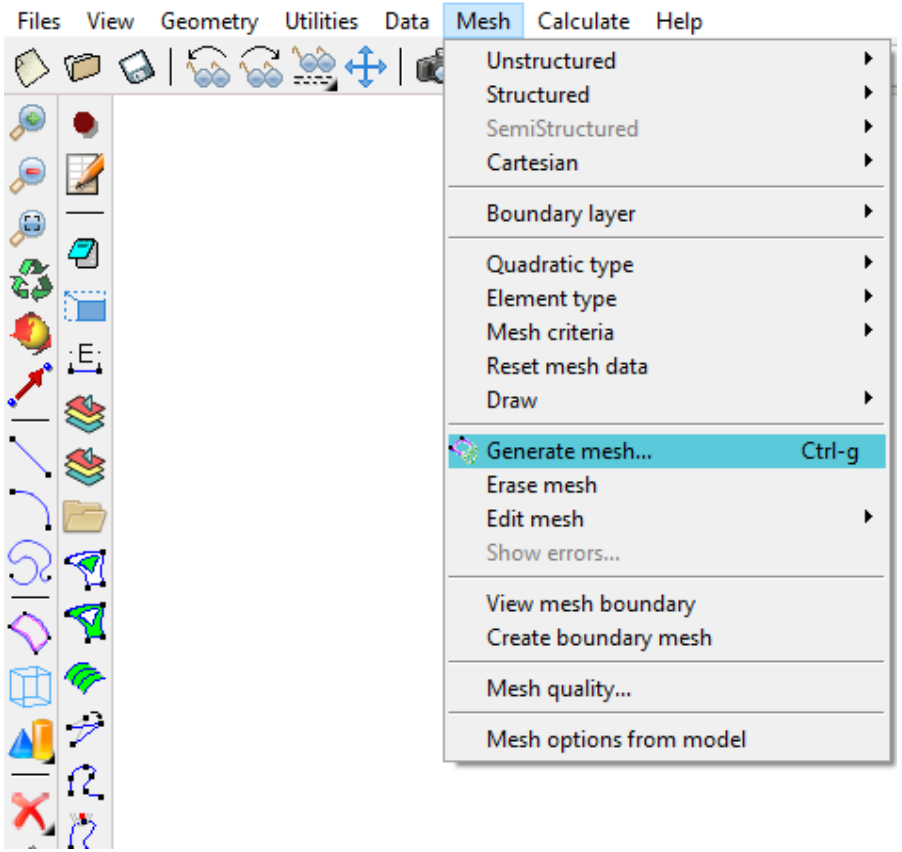
(b) Select the analysis type (plane strain, plane stress) and also the integration rules for the GeMA analysis.

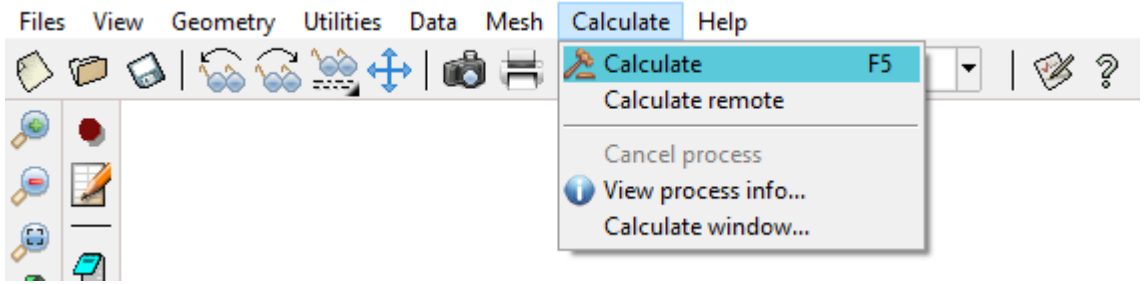


(c) Set the tolerances and some solver options of the analysis.

Steps – Mesh

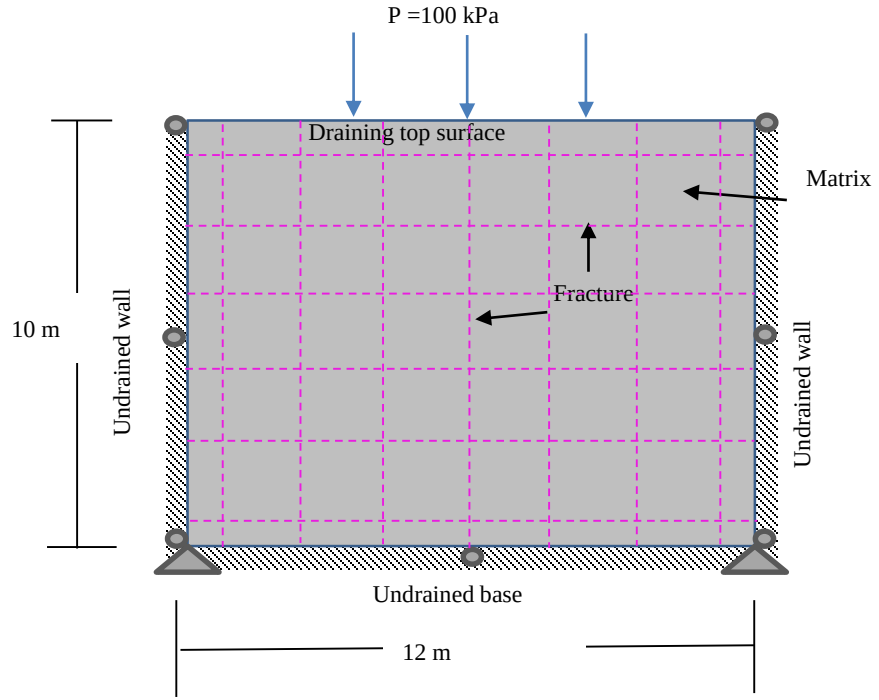
(8) **Generate mesh and GeMA input files**





Click on ‘Calculate’ to generate GeMA input files

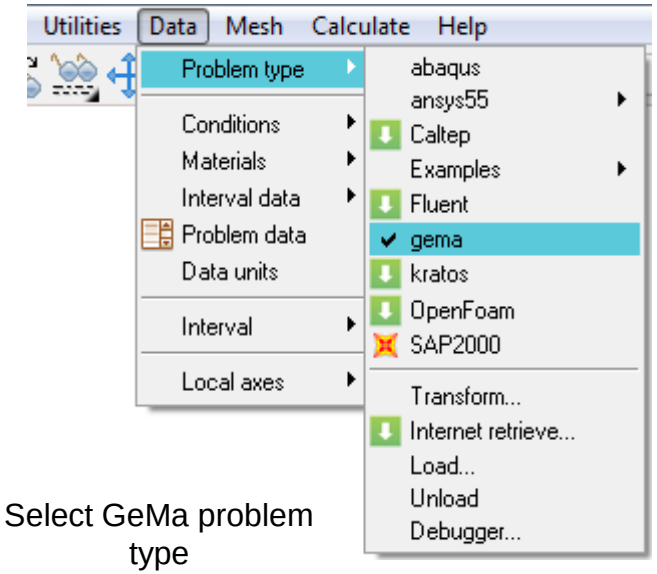
Example 5 – Hydro-Mechanical Test – 2D



Parameters	Example 5
Young's modulus, E(kPa)	1.00E+05
Possion's ratio, υ	0.2
Normal stiffness of fracture, k_n (kPa/m)	2.00E+04
Tangencial stiffness of fracture, k_s(kPa/m)	1.00E+04
Hydraulic permeability of the matrix, k_m(m/s)	1.00E-11
Fracture aperture, bx=by(m)	4.9e-4
Relative compressibility, $\beta_{fr} = \beta_m$(1/kPa)	0
Fluid viscocity, μ (cp)	1
Fracture spacing, s(m)	1
Specific weight of water, γ_w(kN/m^3)	10
Analysis time, t(s)	1.00E+09

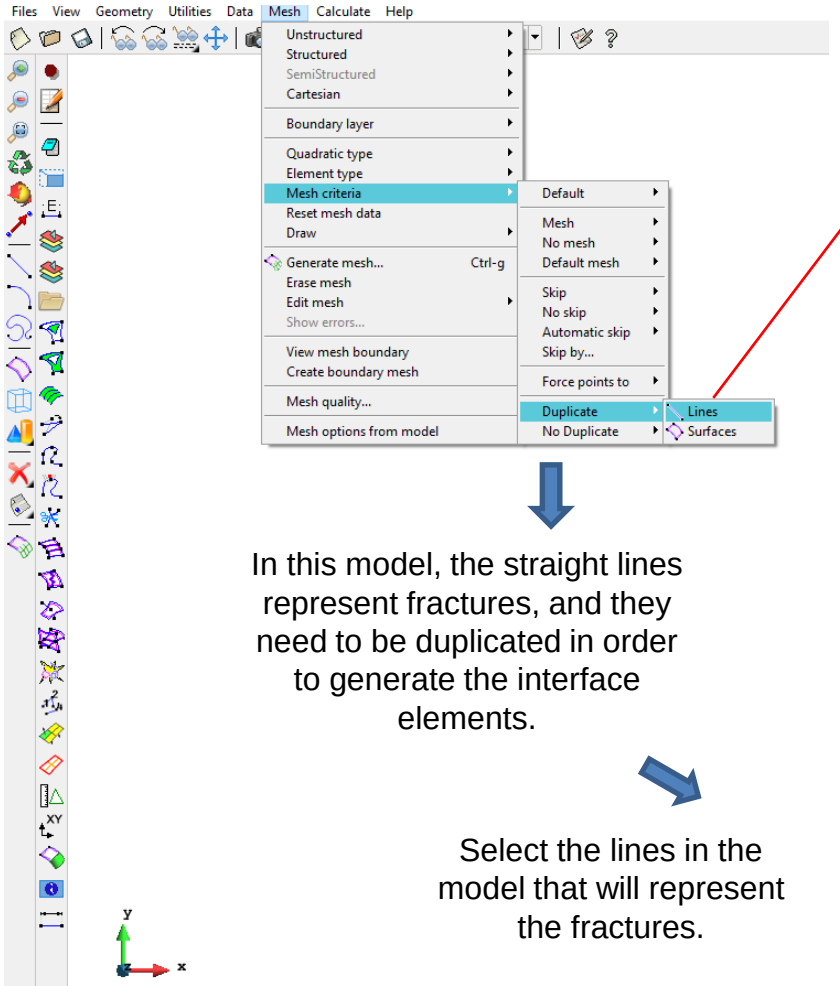
Steps – Problem Type

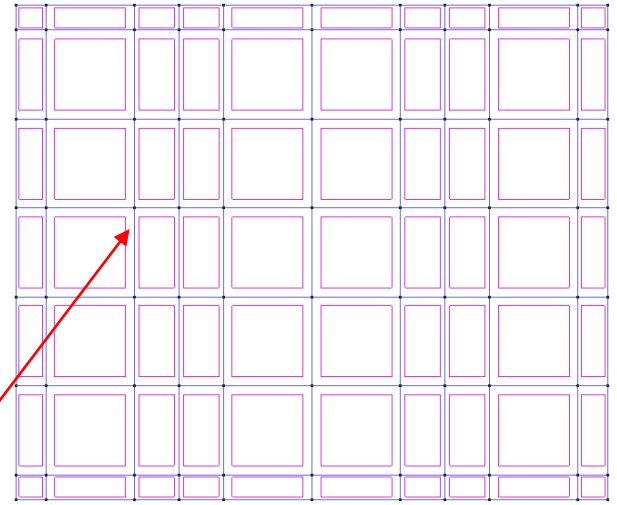
(1) **Set the GeMA problem type**



Select GeMa problem type

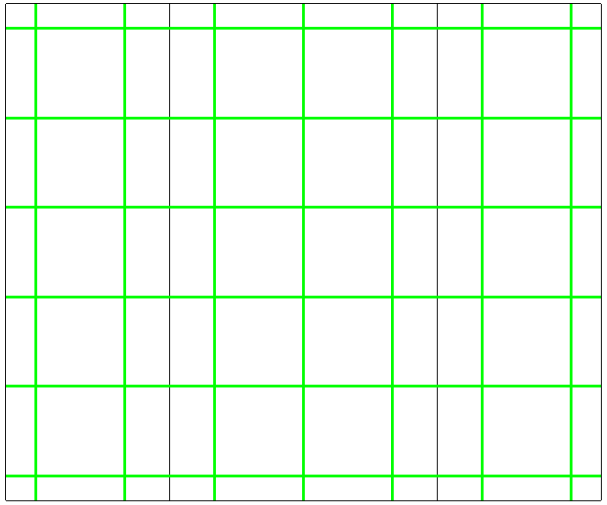
(2) **Duplicate the nodes from lines**





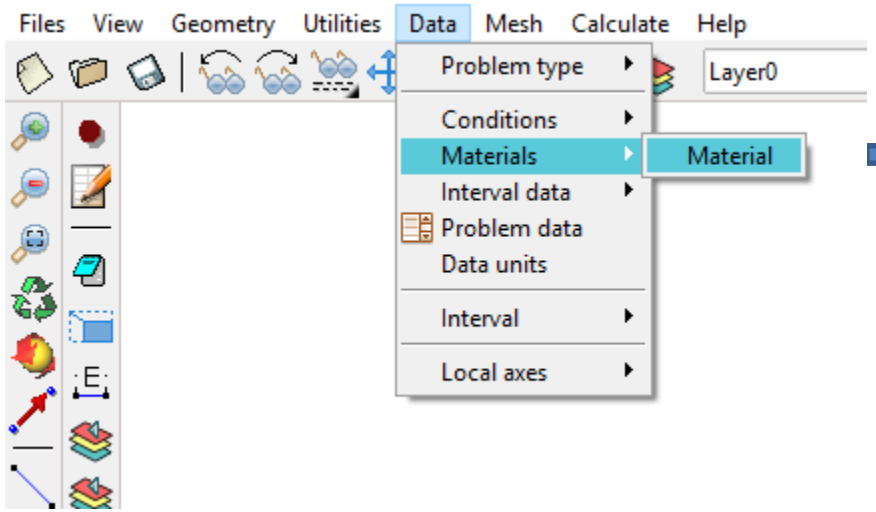
In this model, the straight lines represent fractures, and they need to be duplicated in order to generate the interface elements.

Select the lines in the model that will represent the fractures.



Steps – Material

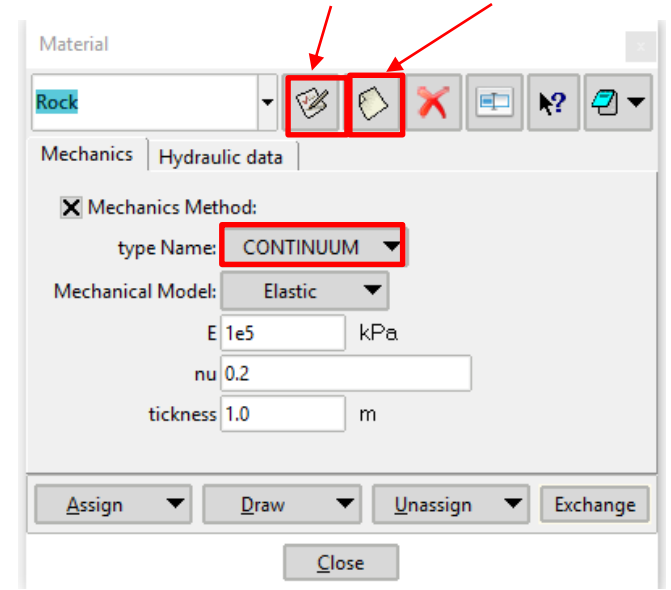
(3) **Generate material**

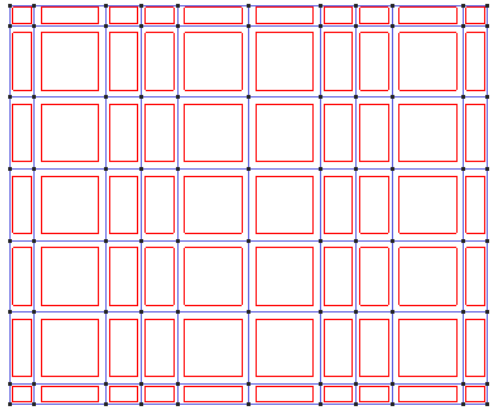


(a) Create a new material for the solid elements

(a.2) Save material

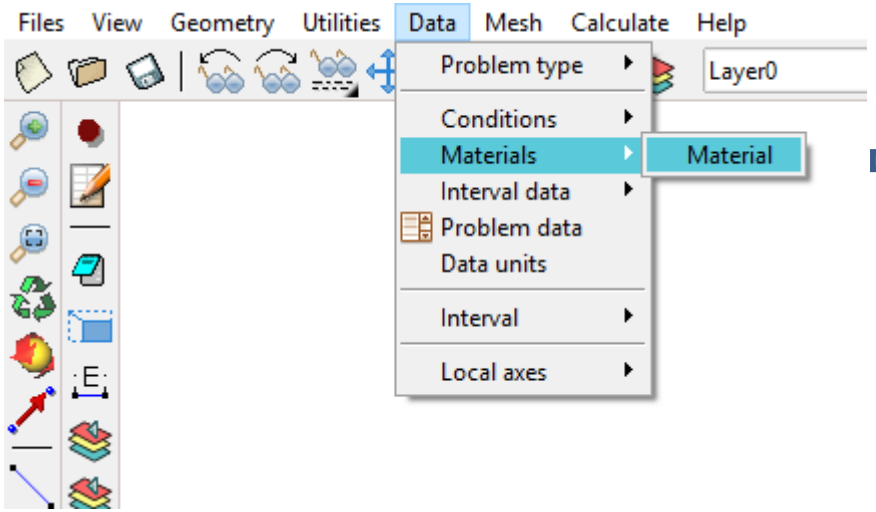
(a.1) Create new material



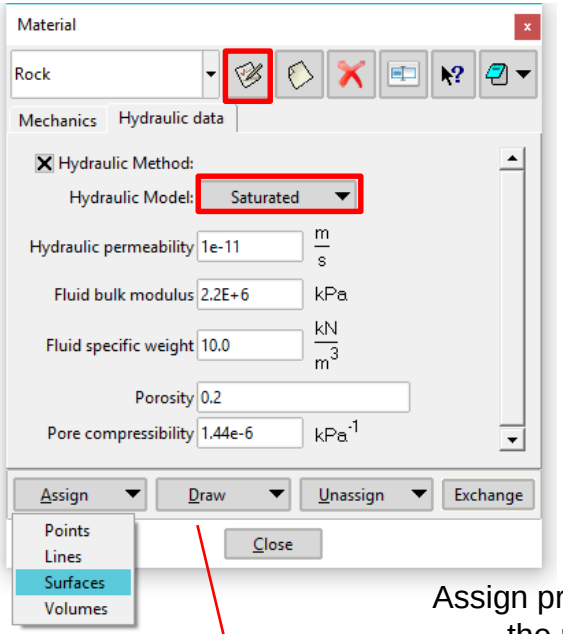


Steps – Material

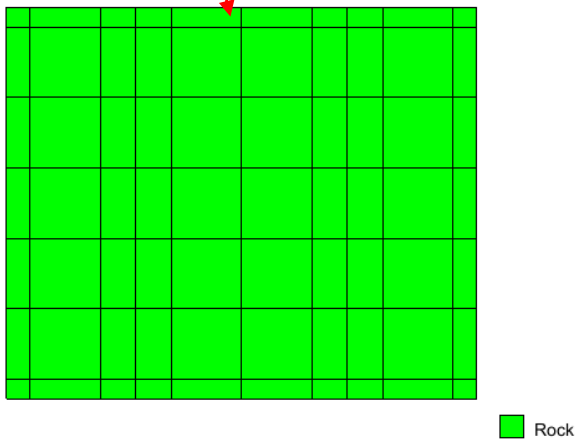
(3) **Generate material**



(b) Set hydraulic data for the Rock material, following a hydraulic saturated model

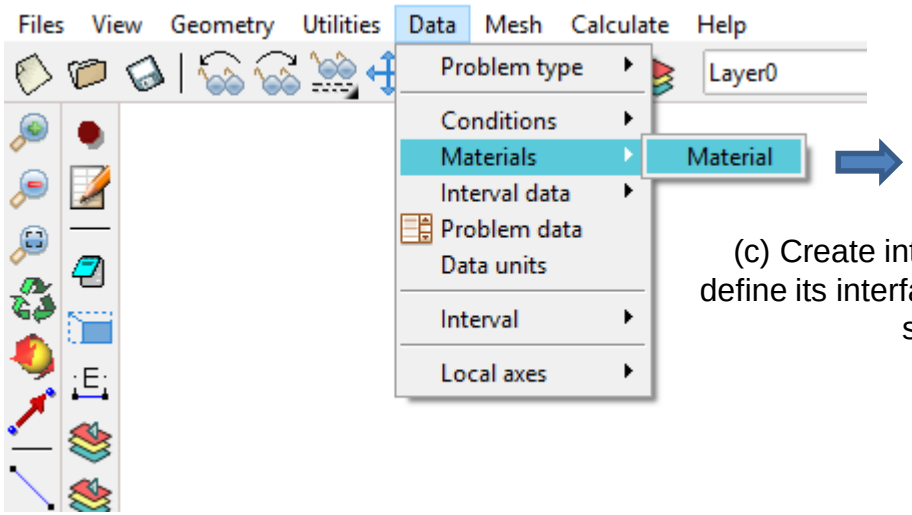


Assign properly to the surface the rock properties.



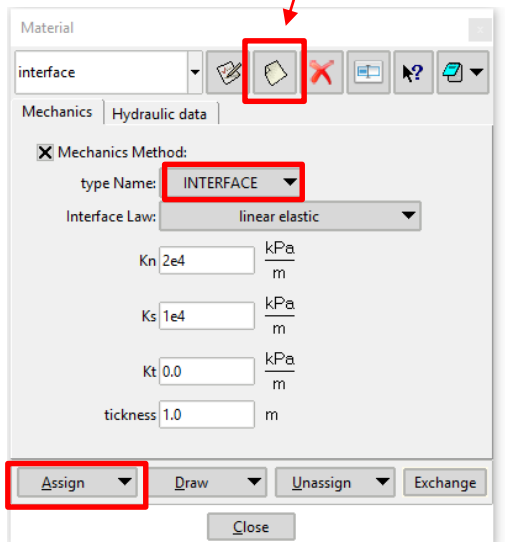
Steps – Material

(3) **Generate material**

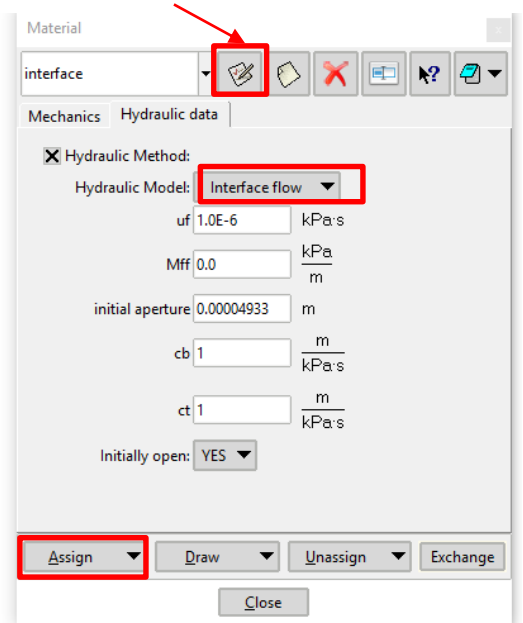


(c) Create interface material, define its interface properties and save

Create interface material

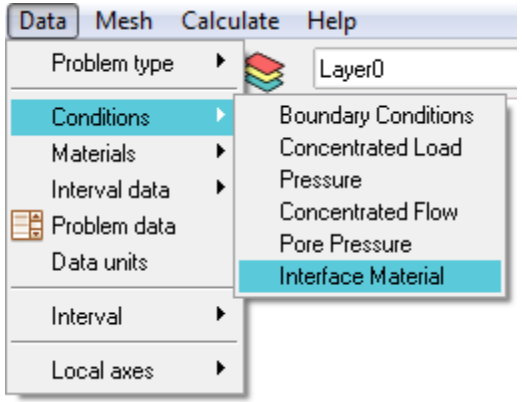


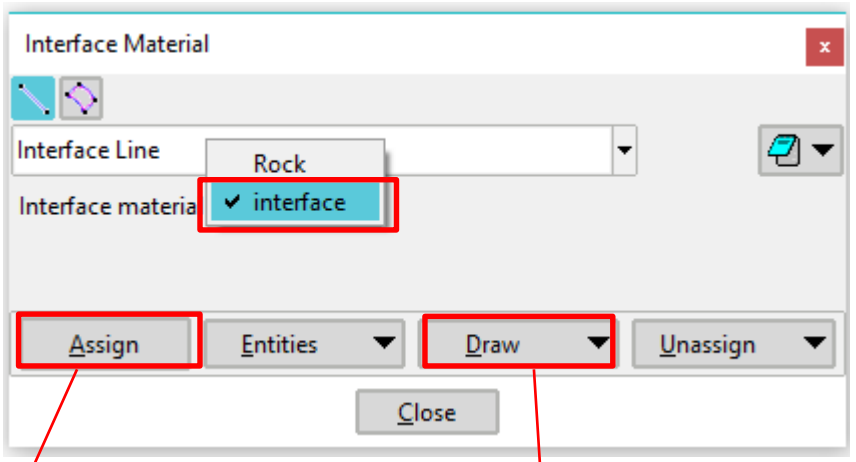
Save material



Steps – Conditions

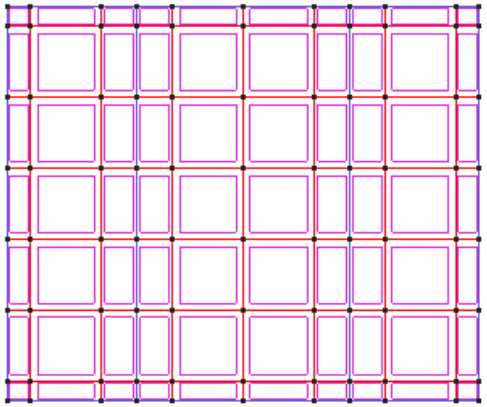
(4) **Attach conditions for the interface material**

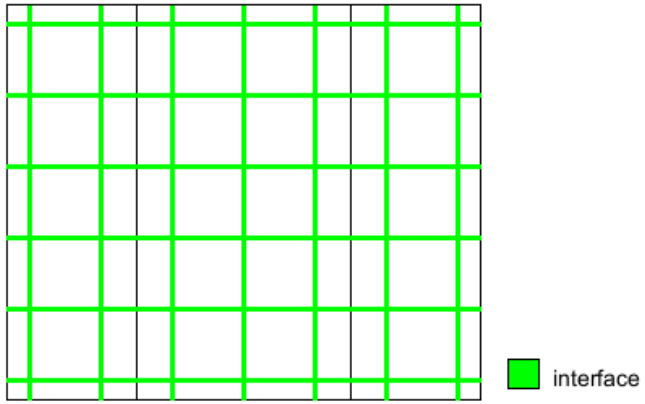




(a) Assign the interface material into the lines

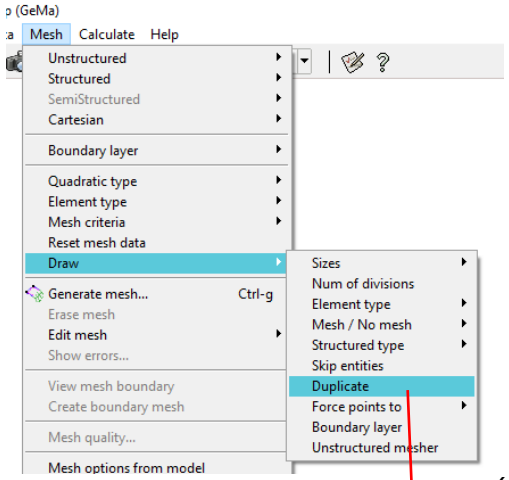
(b) Verify if the attachment was done correctly



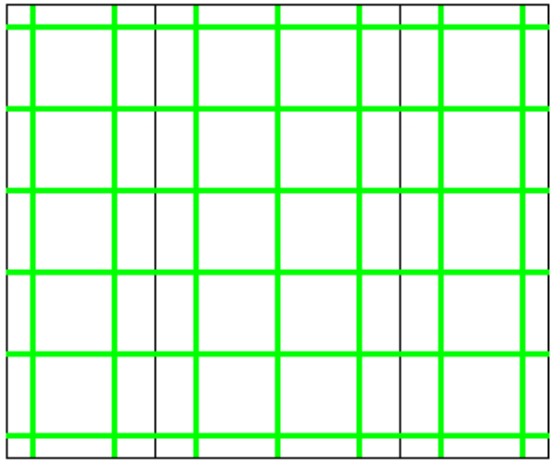


Steps – Conditions

(4) **Attach conditions for the interface material**

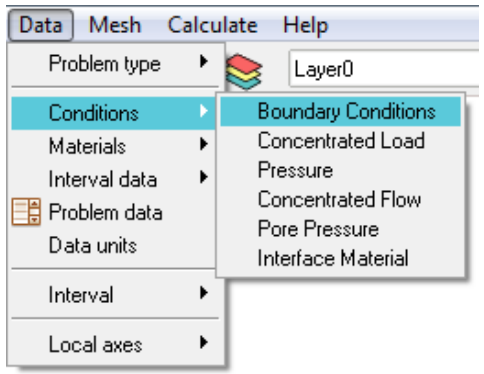


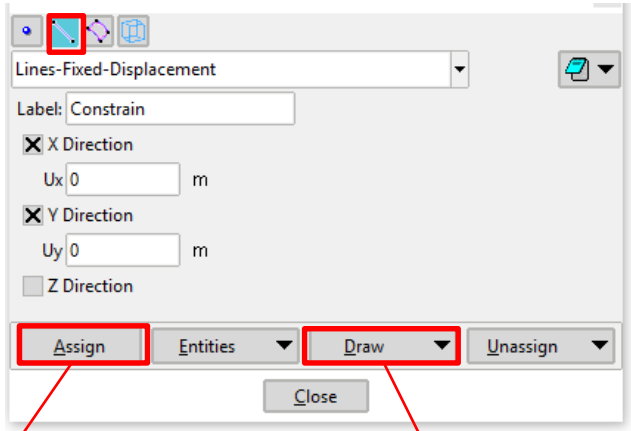
(c) Duplicate interface material



Steps – Constraints

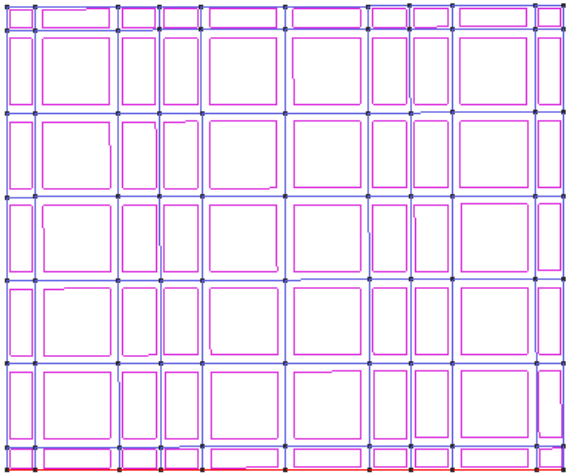
(5) **Apply the boundary conditions of the problem**

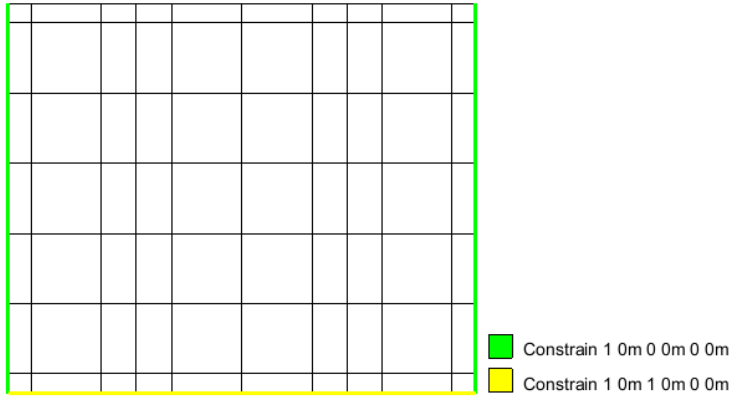




(a) Assign boundary conditions

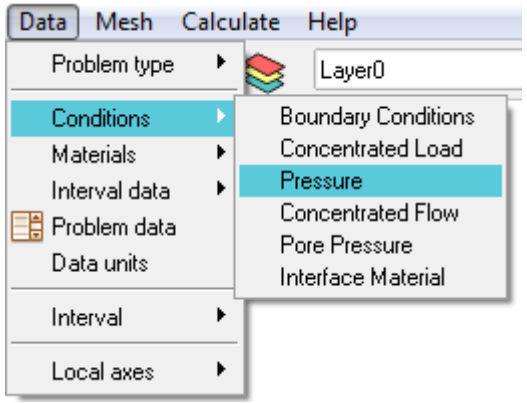
(b) Verify if the boundary conditions were correctly represented in the model



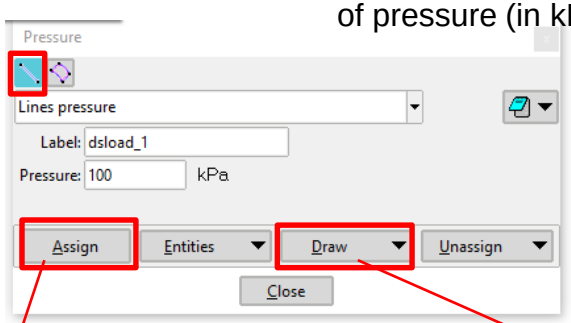


Steps – Pressure Load

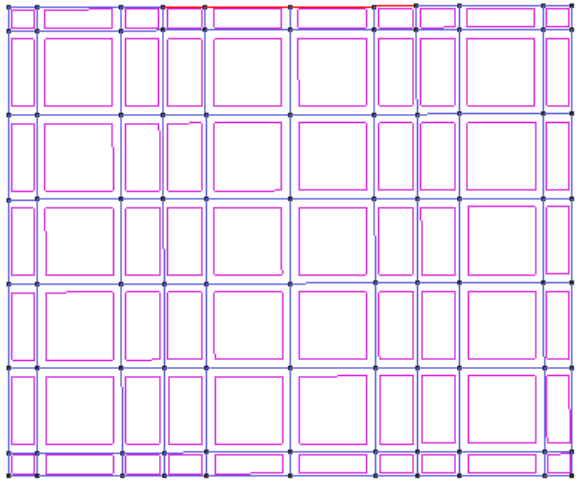
(6) **Apply pressure load**



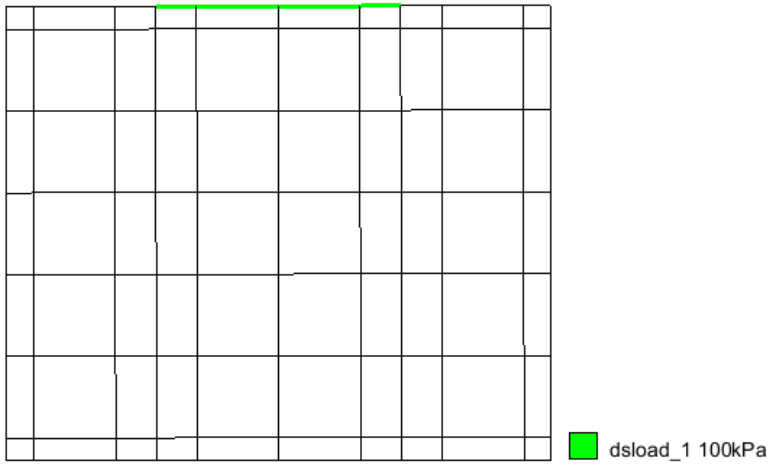
(a) Write down the labels and values of pressure (in kPa)



(b) Assign the pressure load in the correct region of the model, selecting the corresponding lines

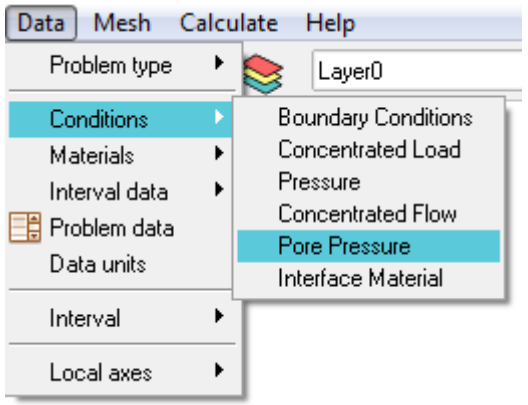


(c) Verify if the pressure load was correctly incorporated to the model

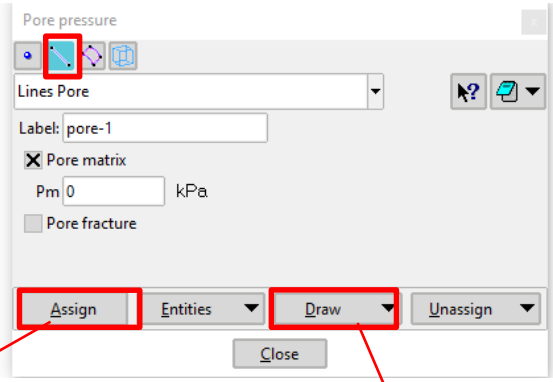


Steps – Pore Pressure

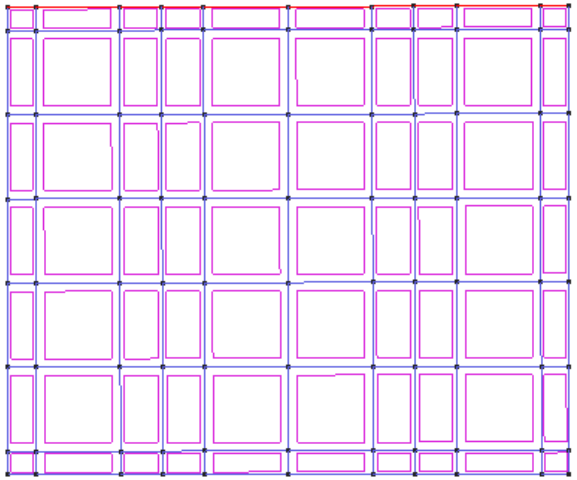
(7) **Apply pore pressure**



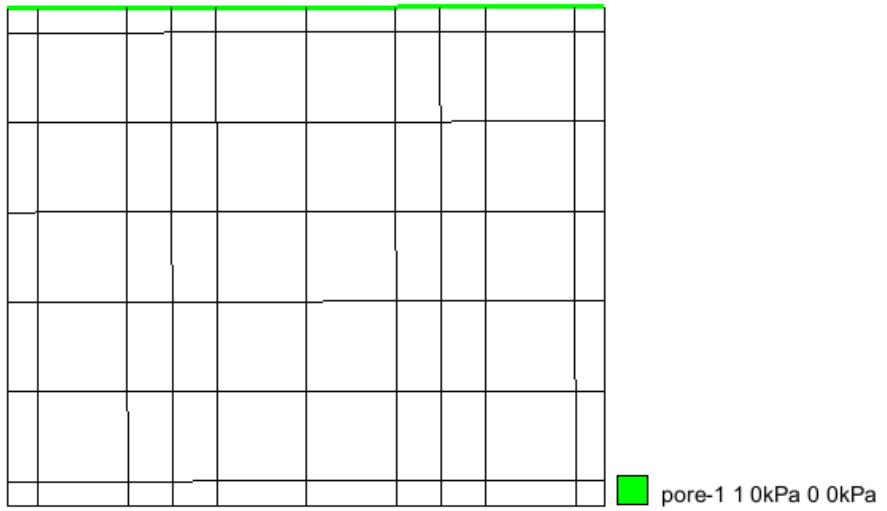
(a) Write down the labels and values of pore-pressure (in kPa)



(b) Assign the pore pressure values in the correct region of the model, selecting the corresponding lines

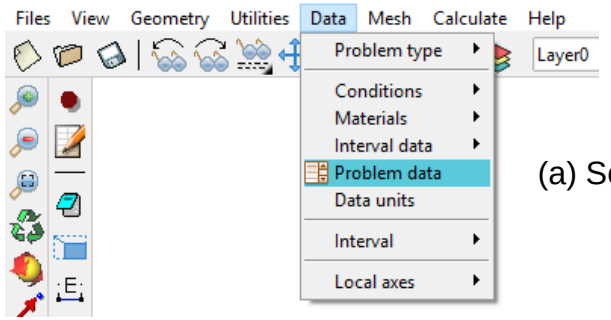


(c) Verify if the pore pressure load was correctly incorporated to the model

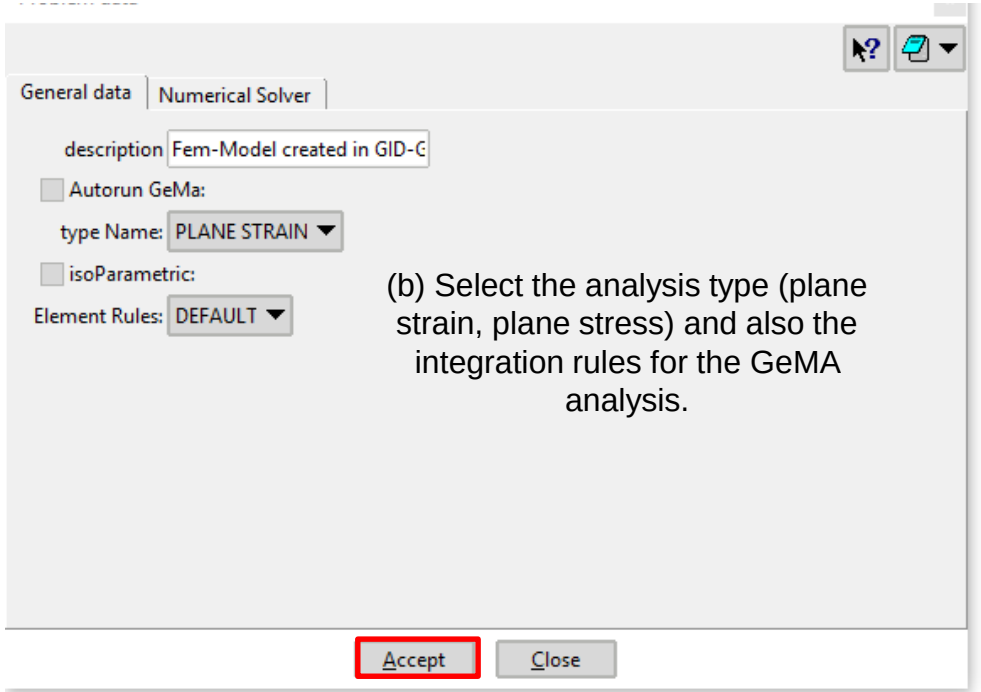


Steps – General Problem Data

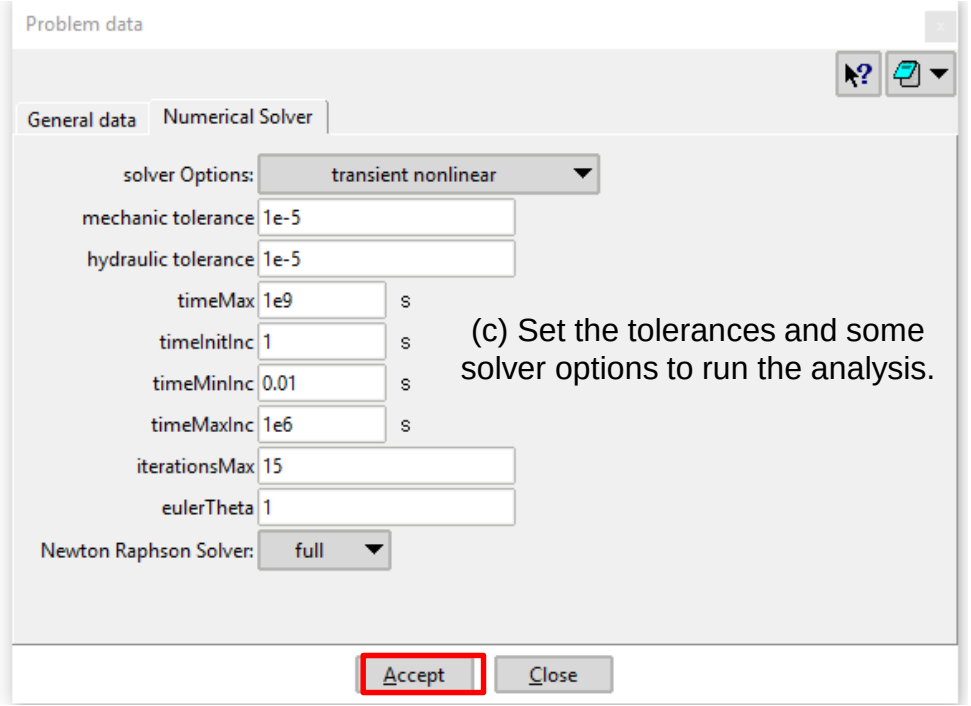
(8) **Set the general problem data**



(a) Select the problem data icon



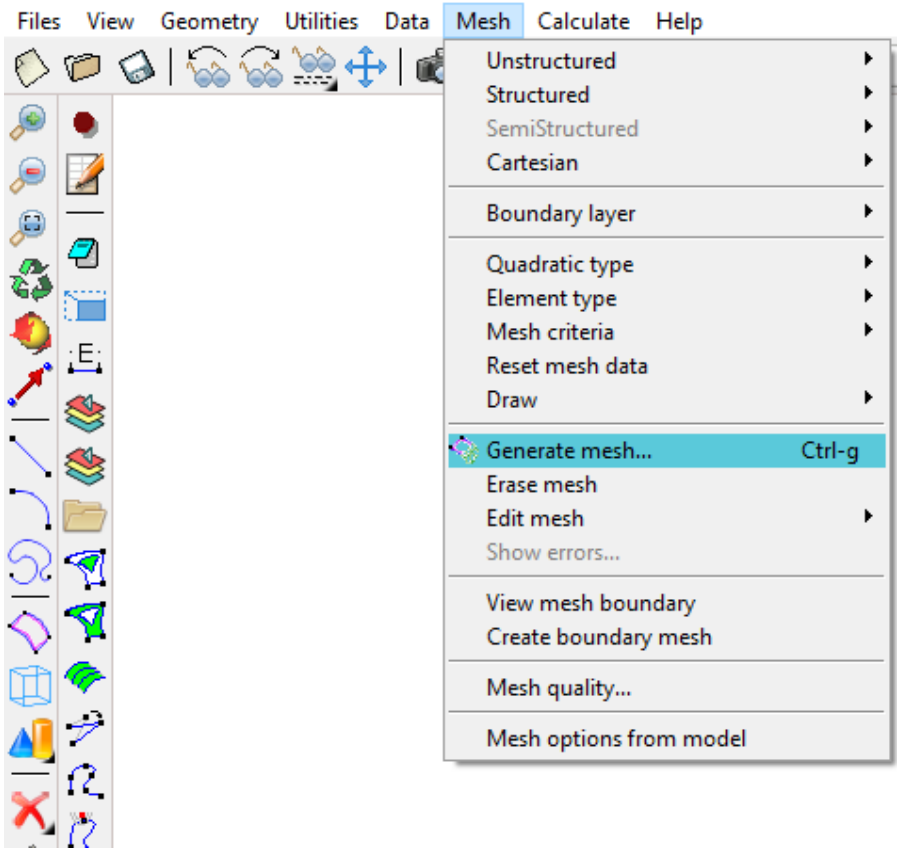
(b) Select the analysis type (plane strain, plane stress) and also the integration rules for the GeMA analysis.

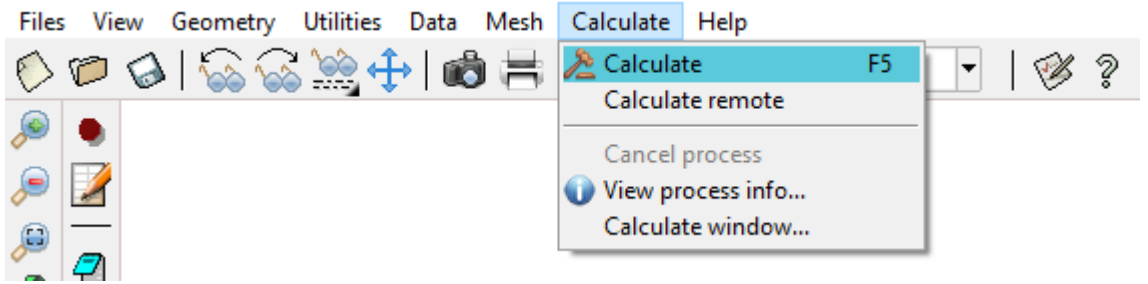


(c) Set the tolerances and some solver options to run the analysis.

Steps – Mesh

(9) **Generate mesh and GeMA input files**





Click on ‘Calculate’ to generate GeMA input files.