GiD2GeMA

GeMA problem type for GiD pre processor

Francisco Dias
Thiago Perry
Cristian Mejia

Version 1.1.0 – July 3, 2021

Table of contents

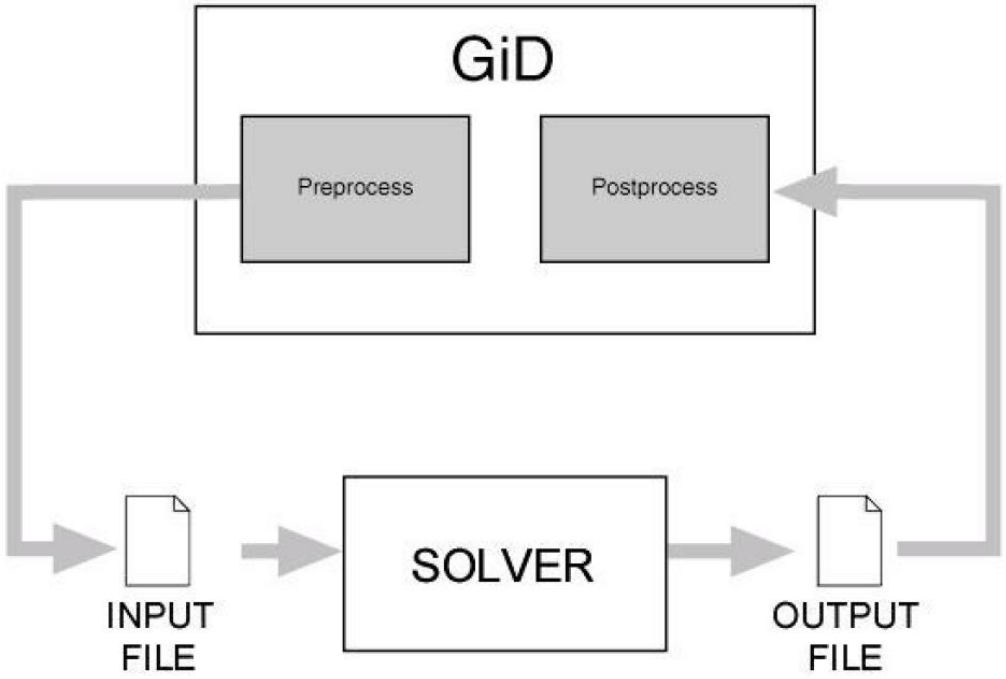
Overview

GiD environment

- GiD is a general-purpose
  - Must be configurated for each solver
  - Solver do not need be modified
- Problem type
  - Configuration is done by a set of files, called problem type

Problem type defines:

- Conditions
- Materials
- General data
- Unit systems
- Symbols
- Format of the input file for the solver



Classic problem type (deprecated)

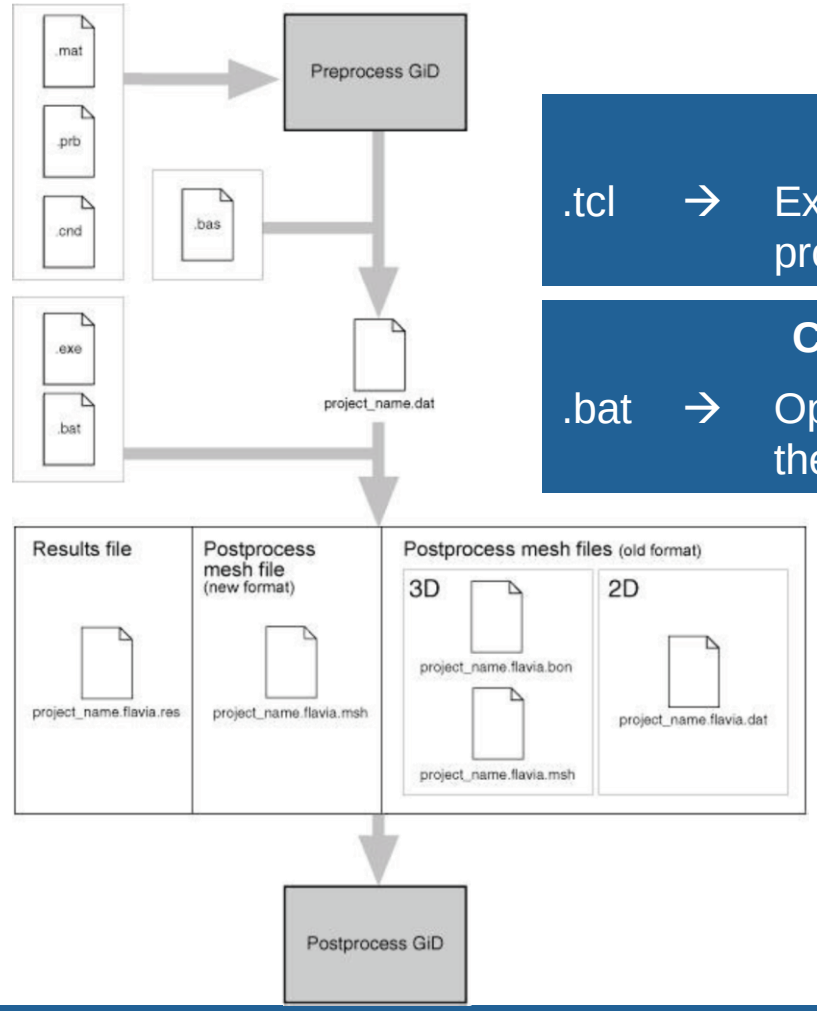
- Organization

Configuration files

.xml	→	XML-based configuration
.cnd	→	Conditions definitions
.mat	→	Materials properties
.prb	→	Problem and intervals data
.uni	→	Units Systems
.sim	→	Conditions symbols
.geo	→	Symbols geometrical definitions

Template files

.bas	→	Information for the data input file
***.bas	→	Information for additional files



Tcl extension files

.tcl	→	Extensions to GiD written in the Tcl/Tk programming language

Command execution files

.bat	→	Operating system shell that executes the analysis process

New problem type system

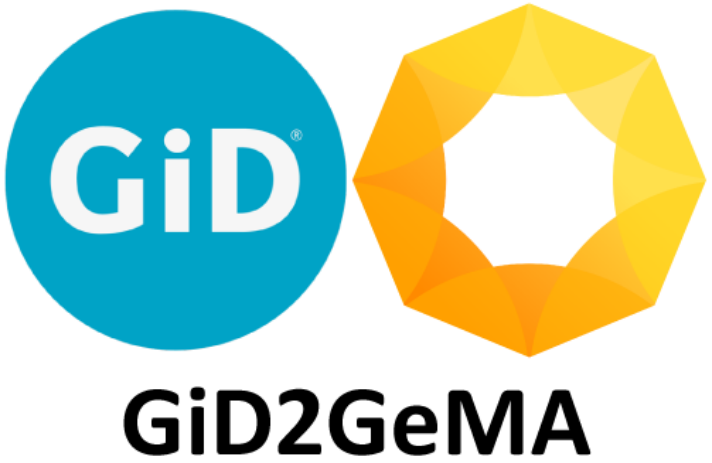
- Available from the 13th version of GiD
- Classic problem type is still supported by GiD, but it is considered deprecated
- New organization:

New problem type		
.spd	→	Main configuration file of the data tree.
.tcl	→	Main TCL file, initialization.
scripts/**.tcl	→	Output description to the file of analysis.
.cnd	→	Conditions definition.

Programming languages:
- **XML**
- **XPATH**
- **TCL/TK**

Auxiliary libraries:
- **Customlib**
- **TDOM**

GiD2GeMA problem type

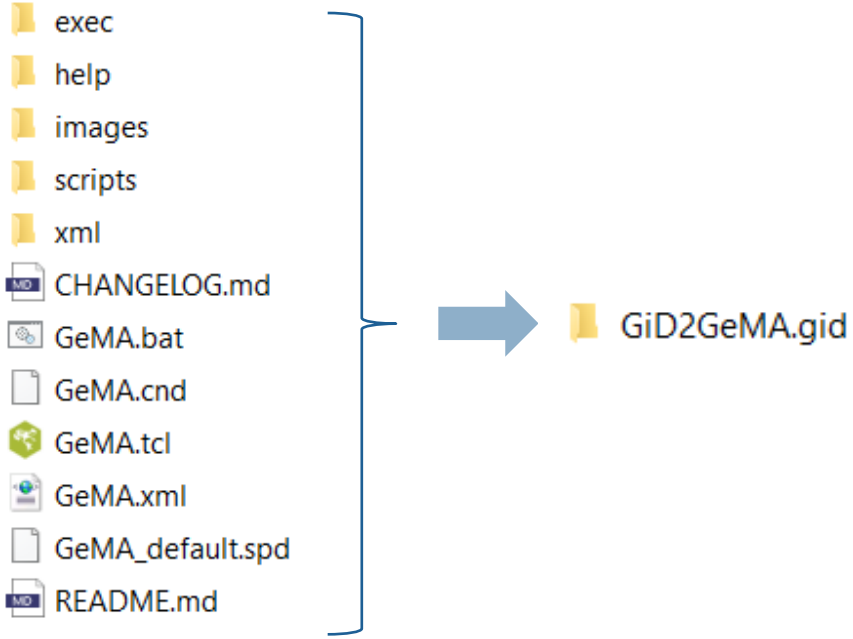


- GiD2GeMA is a problem type for GeMA (Geo Modeling Analysis) framework
- This application was developed to preprocess and run models
- Official versions: https://git.tecgraf.puc-rio.br/gema/GiD2GeMA.gid/-/releases
- Support for 2D and 3D models
- For more information about versions, see “CHANGELOG.log” file

How to install

How to install

1. Put the set of folders and files inside a folder named as “GiD2GeMA.gid”



2. Copy “GiD2GeMA.gid” folder to GiD problem types folder.

It depends on your GiD version and its installation directory. For example, for GiD version 14.0.5 with default installation directory:
“C:\Program Files\GiD\GiD 14.0.5\problemtypes”

3. Restart GiD program

What’s new in this version

What’s new in this version

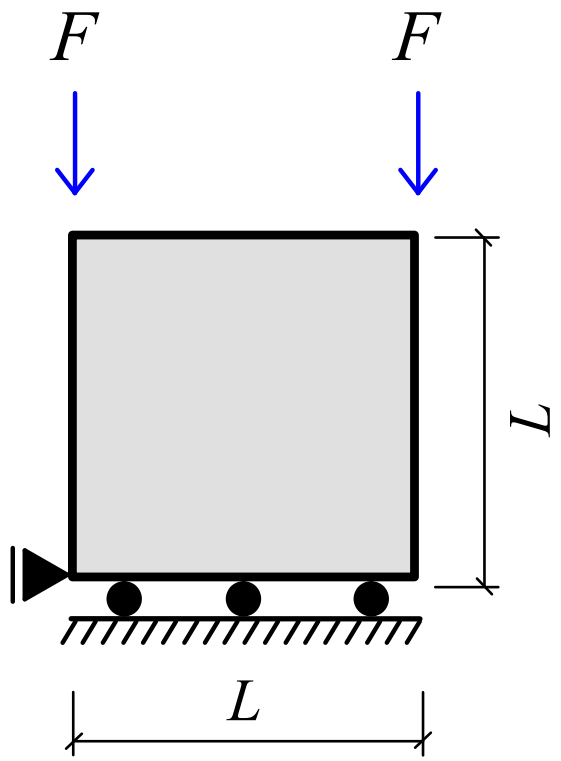
Version 1.1.0 - 2021-07-03

- Changed
  - Visual identity update.
  - Embedded GeMA upgraded to version 1.4.0 revision 3614.

See the CHANGELOG for more information.

2D mechanical example

Definitions

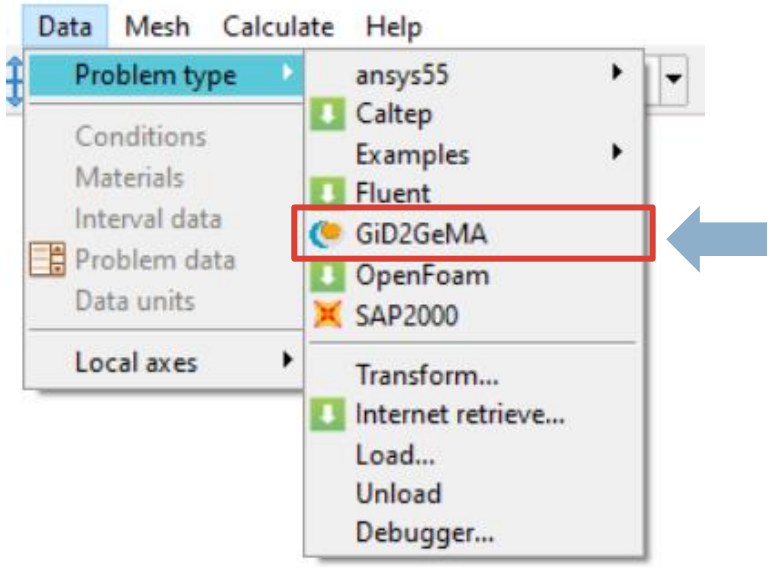


Parameters	Values	Unit
Young's modulus, E	3.00E+07	kPa
Poisson's ratio, υ	0.20	-
Applied load, F	100	kN
Length, L	1.0	m
Thickness, t	0.1	m

Problem Type and model info

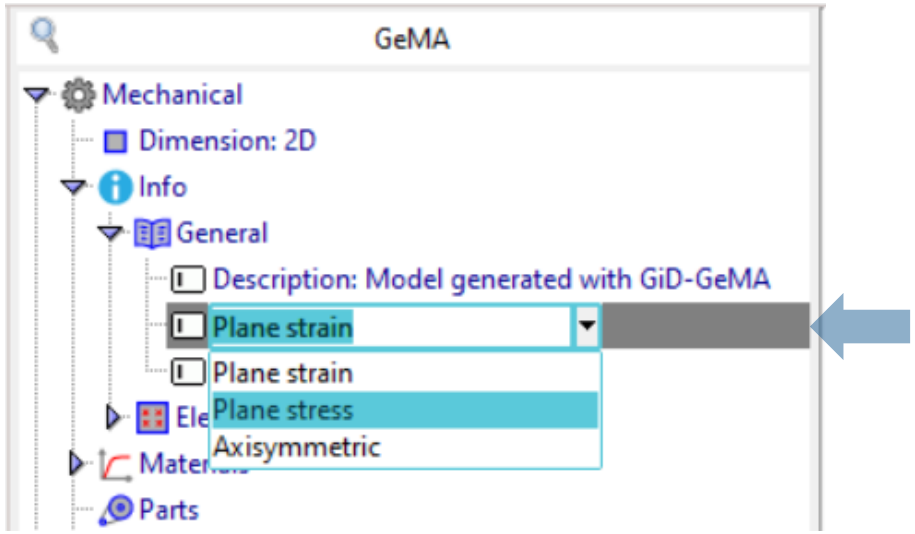
1. Set the problem type

a) **In the GiD menu, select “Data > Problem type > GiD2GeMA” option.**



2. Set model info

a) **In the tree data, doble-click on “Mechanical > Info > General > 2D analysis type” field. Select “Plane stress” in the drop-down list**

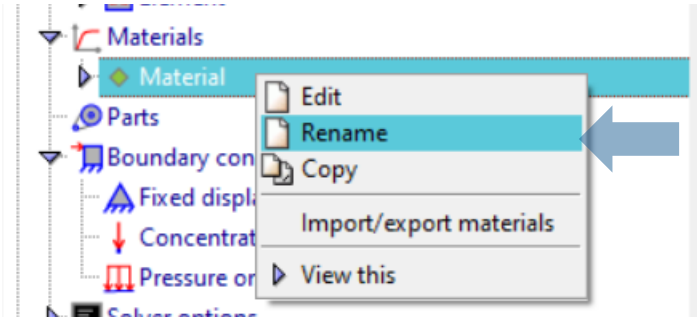


Remark: To learn how create the geometry and the mesh, see “Help > Tutorials…” in GiD menu.

Material definition

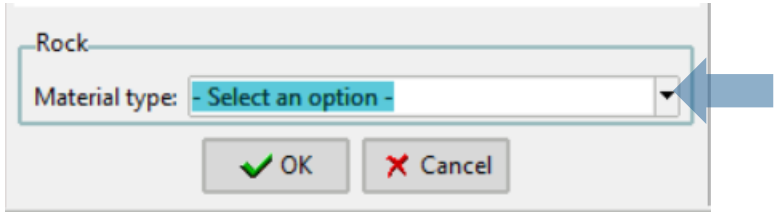
3. Set the material

a) Open “Mechanical > Materials” folder in the data tree. Right click on “Material” and select “Rename” to change the material name to “Concrete”.

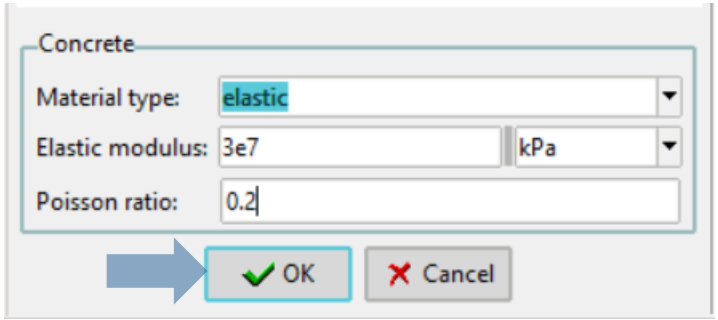


b) Double-clik on material name “Concrete”.

c) On the resulting window, select the material type.



d) Complete the fields and click on “OK” button

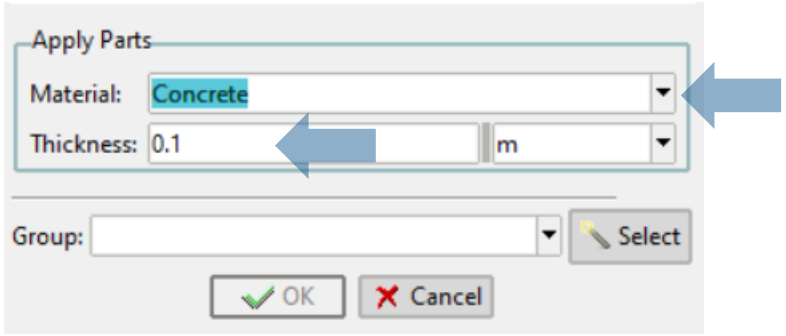


Material assignment

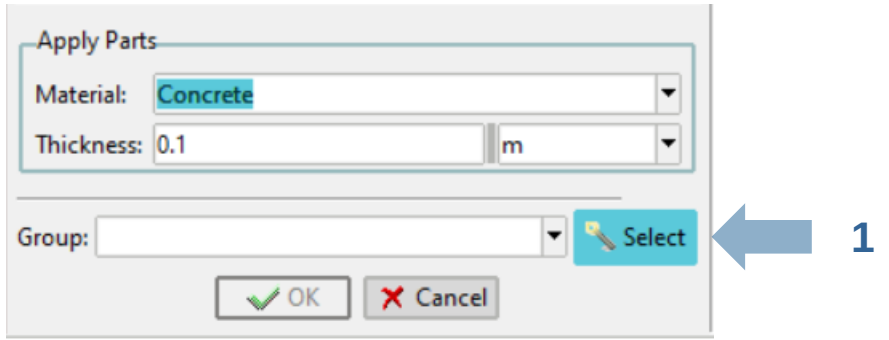
4. Assign material

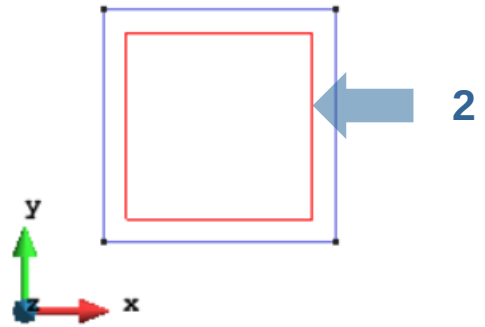
a) In the tree data, doble-click on Double-click on “Mechanical > Parts”.

b) On the resulting window, select the material name on the drop-down list and type the thickness.



c) Click on the “Select” button and select the surfaces to apply these properties.

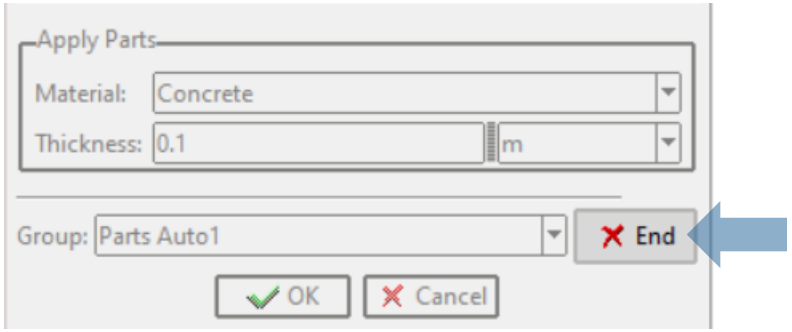




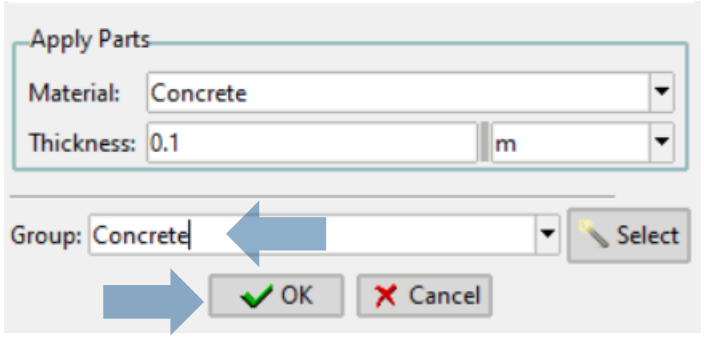
Material assignment

4. Assign material

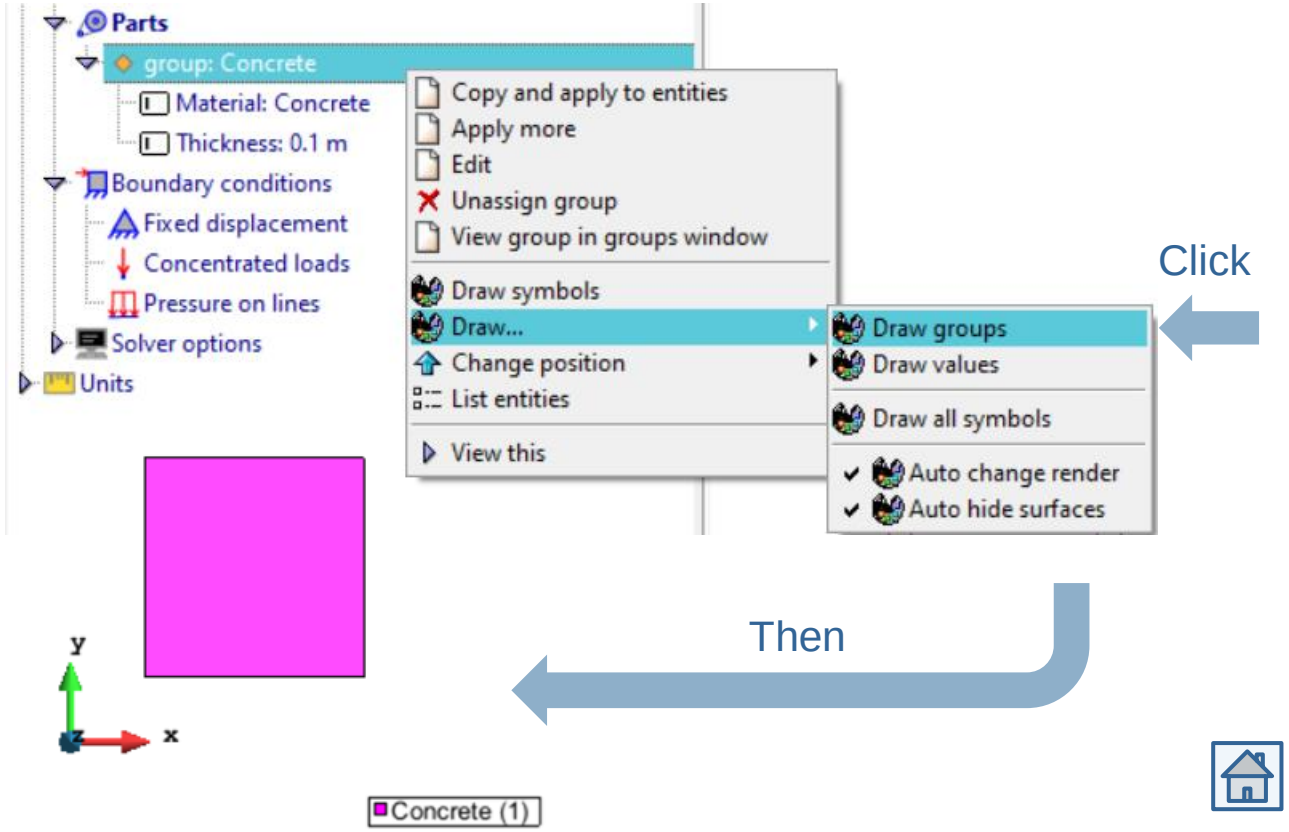
d) Click on the “End” button to finish the selection.



e) Rename the group and click on “OK” button.



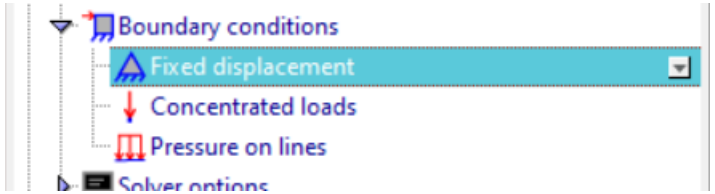
f) To check the assignment, right-click on “Mechanical > Parts > group: Concrete” in the data tree. Select “Draw… > Draw groups”



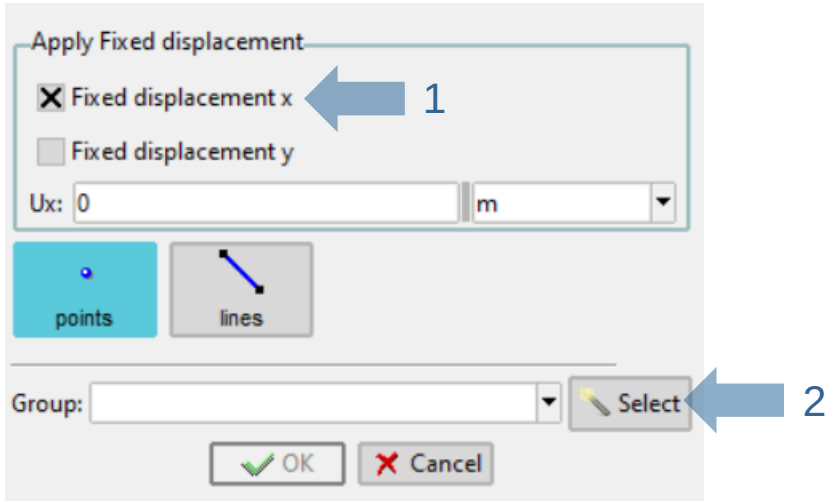
Fixed displacement

5. Assign fixed displacement

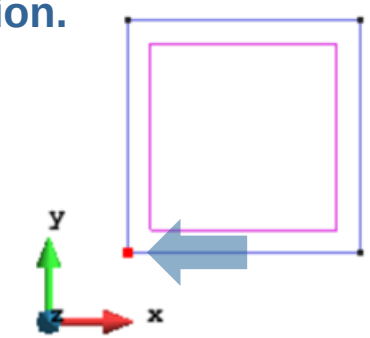
a) Doble-click on the “Mechanical > Boundary conditions > Fixed displacement” condition.



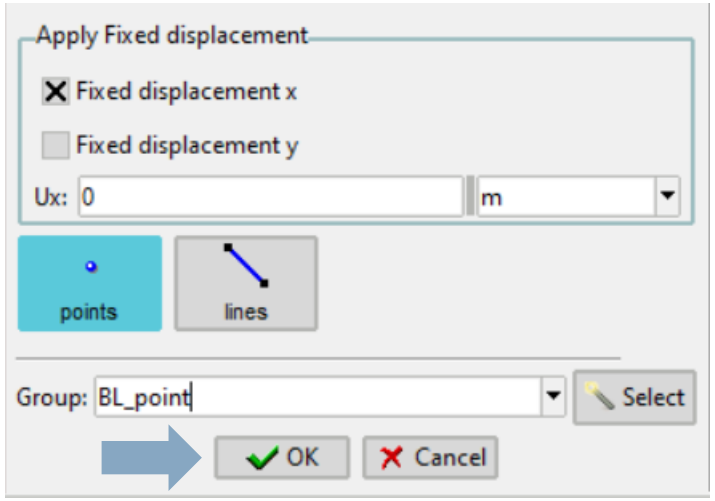
b) On the resulting window, select fixed displacement in x direction. Click on “Select” button.



c) Select the bottom left corner of the square and finish selection.



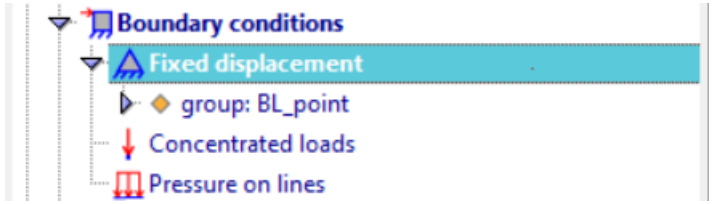
d) Rename the group and click on “OK” button.



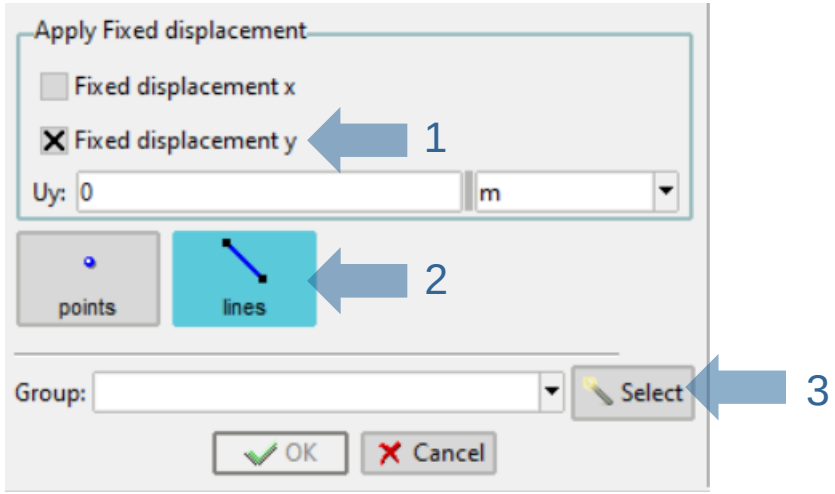
Fixed displacement

5. Assign fixed displacement

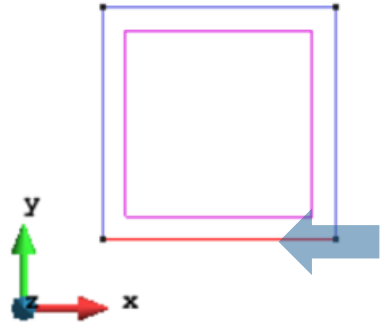
e) Doble-click on the “Mechanical > Boundary conditions > Fixed displacement” condition.



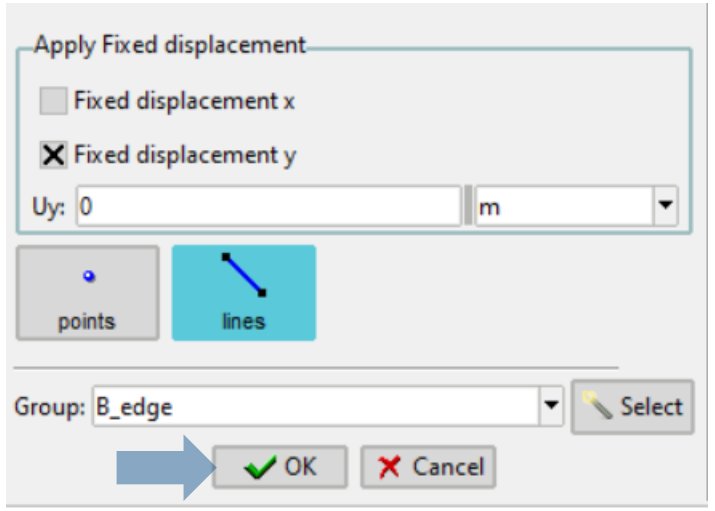
f) On the resulting window, select fixed displacement in y direction. Click on “lines” and “Select” button.



g) Select the bottom edge of the square and finish selection.



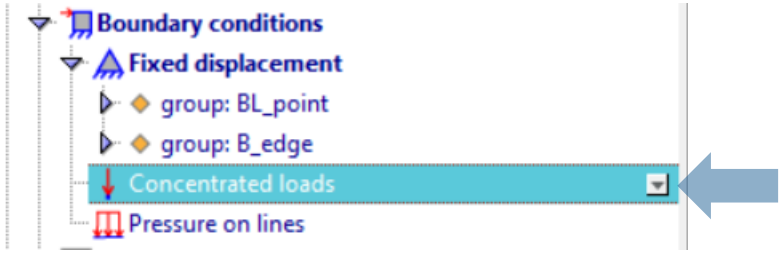
h) Rename the group and click on “OK” button.



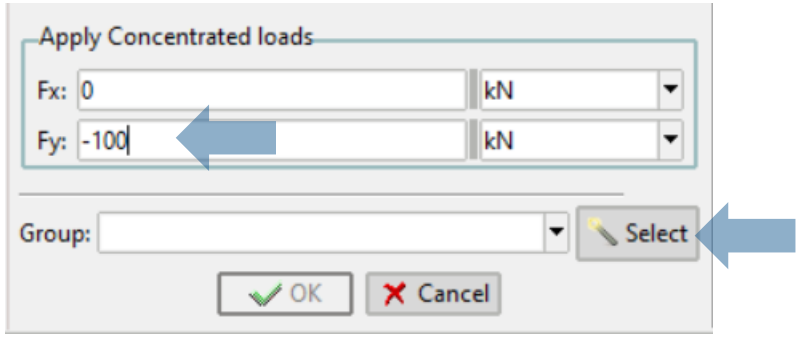
Concentrated loads

6. Assign concentrated loads

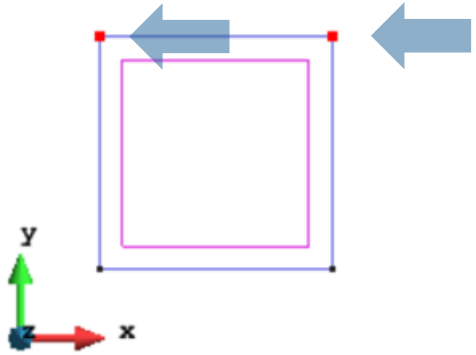
a) Doble-click on the “Mechanical > Boundary conditions > Concentrated loads” condition.



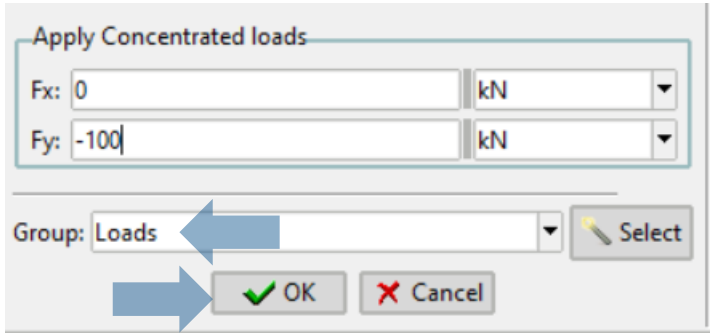
b) On the resulting window, select fixed displacement in x direction. Click on “Select” button.



c) Select the top corners of the square and finish selection.



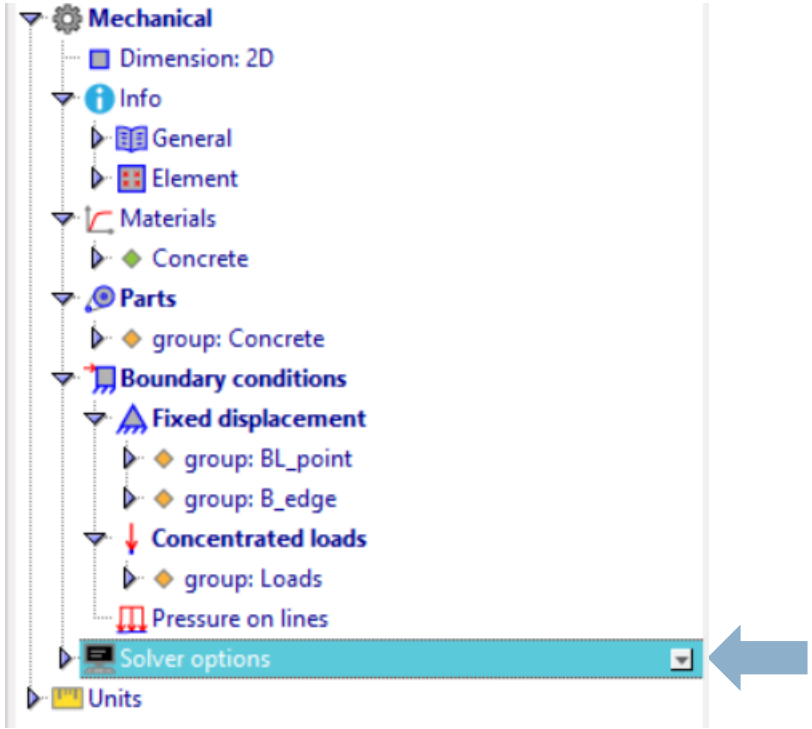
c) Rename the group and click on “OK” button.



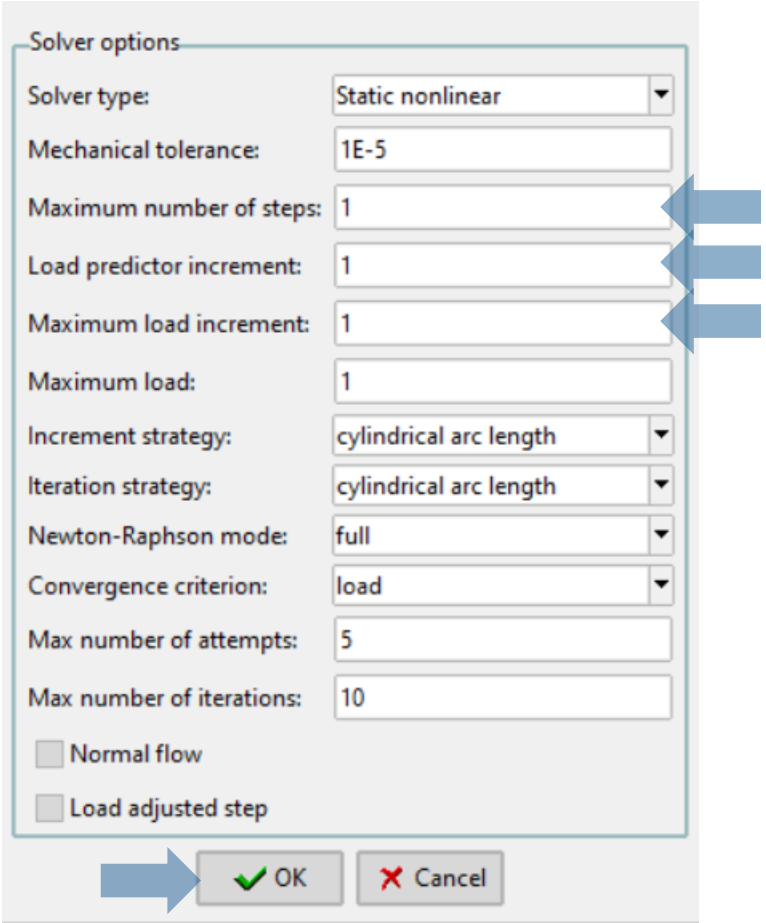
Solver options

7. Set solver options

a) Doble-click on the “Mechanical > Solver options” field.



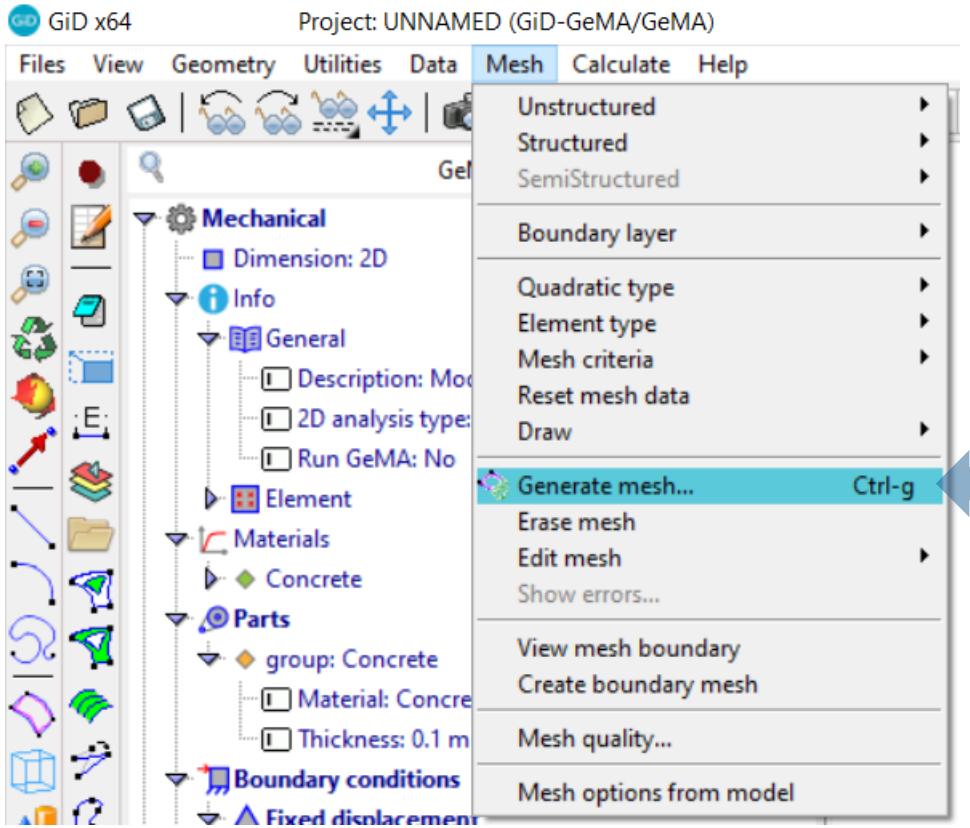
b) On the resulting window, set the correct options for the solver. Click on “OK” button.



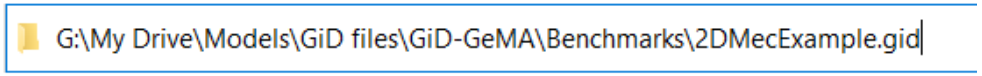
GeMA input files

8. Generate mesh and input files

a) Click on the “Mesh > Generate mesh…” option.

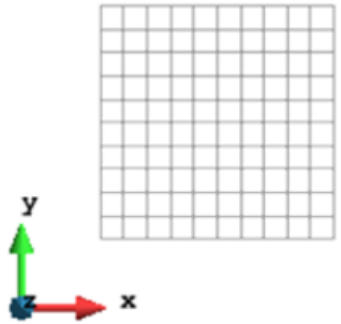


b) Save the project in a directory of your choice. For example:



This version is compatible with cloud storage services.

Remark 1
The mesh must be generated for each change in the groups of boundary conditions or parts.

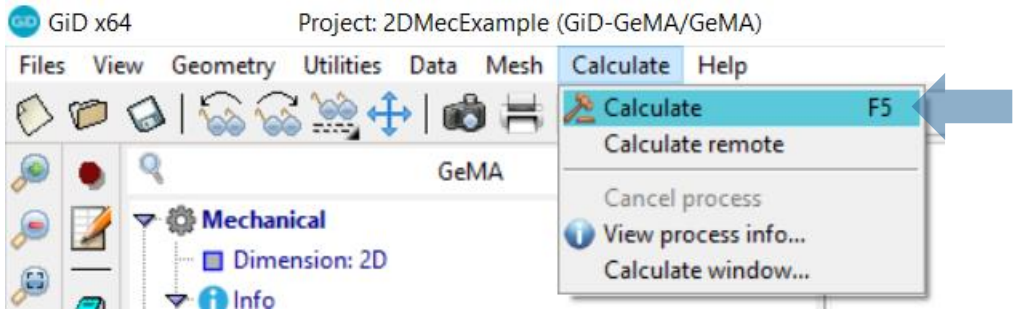


Remark 2
To learn how create the geometry and the mesh, see “Help > Tutorials…” in GiD menu.

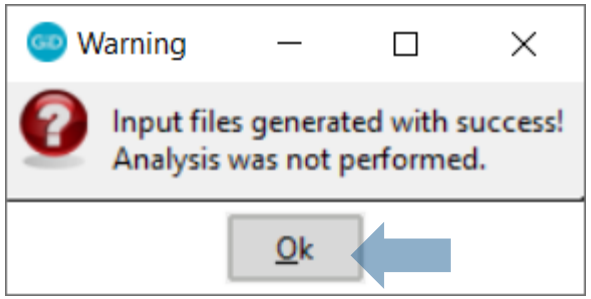
GeMA input files

8. Generate mesh and input files

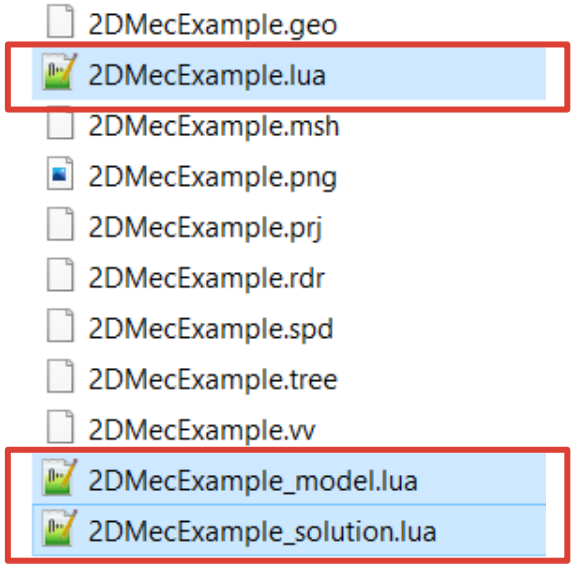
c) Click on the “Calculate > Calculate” menu option.



d) In the notification window, click on “Ok” button.



e) GeMA input files was generated. To get them, go to the project directory chosen in the step 8b.

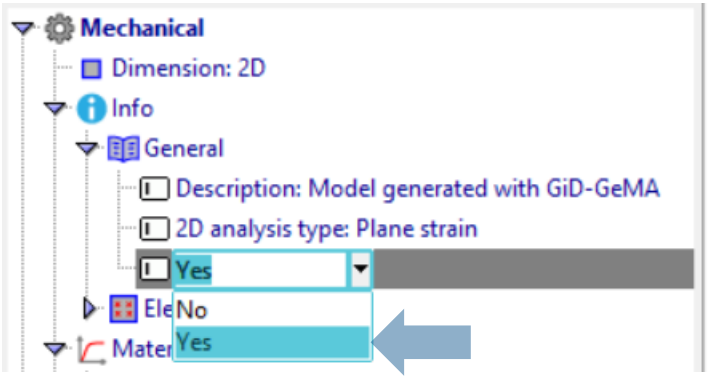


Run GeMA

9. Run GeMA

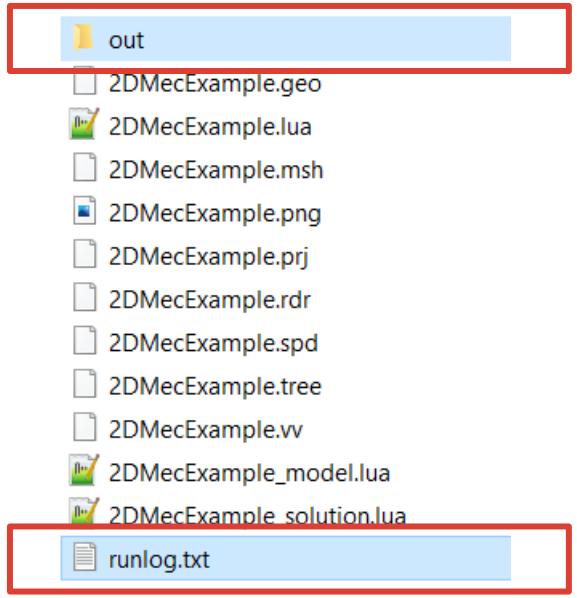
a) To enable this option, double-click on “Mechanical > Info > General > Run GeMA” field.

b) Select “Yes” in the drop-down list.



c) Click on the “Calculate > Calculate” menu option, as in step 8c.

d) GeMA output files was generated. To get them, go to the project directory chosen in the step 8b.



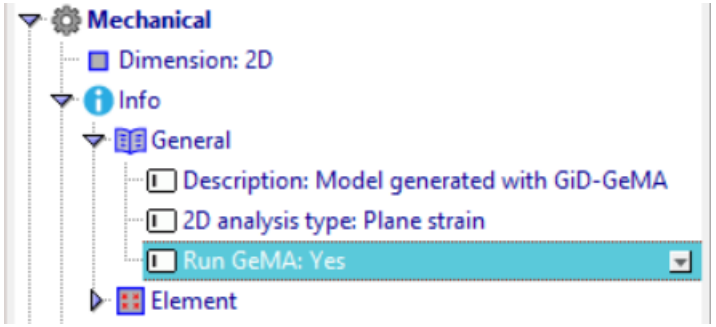
Folder with output files

Log with run information

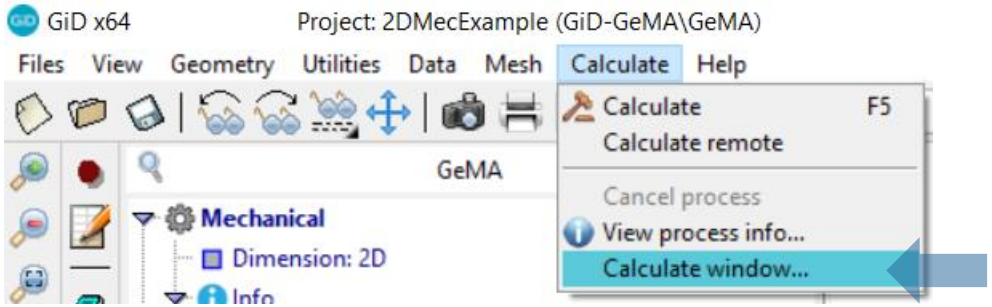
Calculate window

10. Run GeMA (calculate window)

a) Be sure the field "Run GeMA" is equal "Yes".

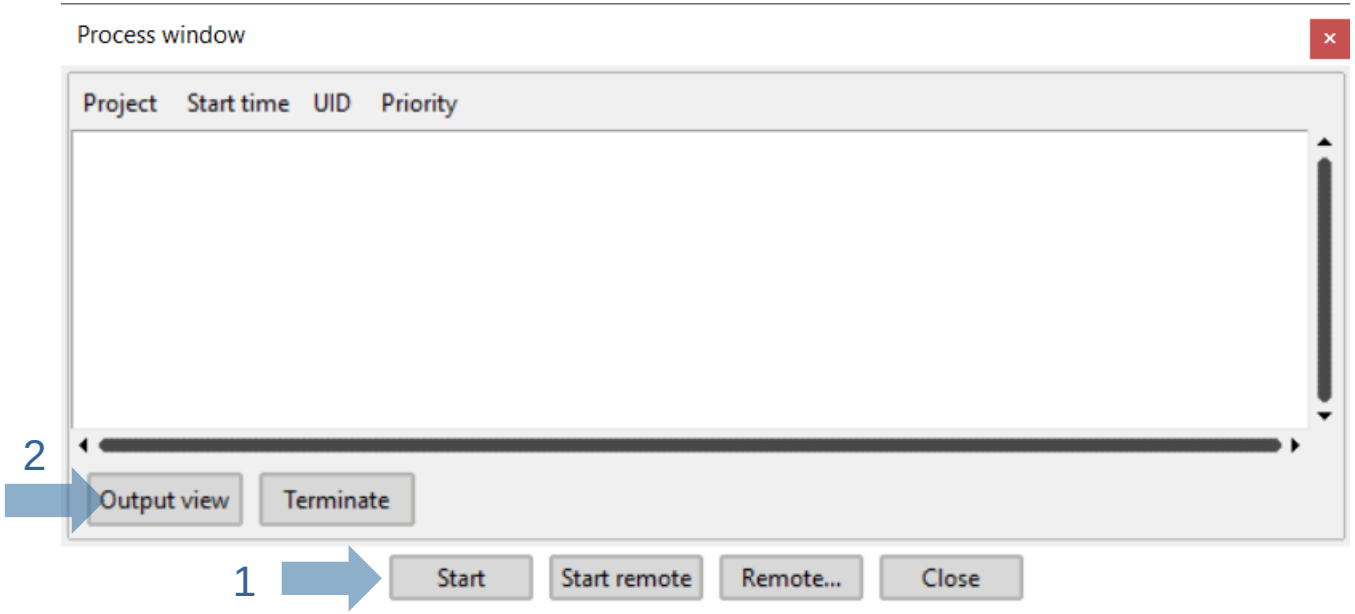


b) Click on the “Calculate > Calculate window…” menu option.



c) In process window, click on the “Start” button.

d) To follow the process info in real time, click on “Output view” button.



e) Find the output files as in step 9d.

Some tips

Some tips

- Whenever you assign any new boundary condition (fixed displacement, concentrated loads, etc.) make sure to generate the mesh again. This is an action required by GiD.
- If you find any bug, please follow these steps:
  1. Save your work and close GiD.
  2. Reopen the model and try again.
  3. Report to us all steps to reproduce error.
- To report any bug:
  - Create an issue; or
  - Send an e-mail.

TECGRAF
PUC-RIO

Contact

Francisco Dias - fcdiass@tecgraf.puc-rio.br
Thiago Perry - tvperry@tecgraf.puc-rio.br
Cristian Mejia - crisms@tecgraf.puc-rio.br