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, <i>E</i> (kPa)	1.00E+06
Poisson's ratio, v	0.25
Applied pressure load, P (kPa)	1000





Steps – Problem Type and Materials







Exchange

Rename material

\?

7

Datete material

(b) Write down the elastic properties

 \mathbf{T}

New naterial

Update

1

.

kPa

m

 \mathbf{T}

<u>C</u>lose

Unassign

CONTINUUM

Elastic

Draw

Steps – Load

(3) Pressure Load		(4)	Boundary Condit	ions
Geometry Utilities Data Mesh Calculate H Problem type Conditions Bou Materials Interval data Data units Interval Local axes P	Help Layer0 indary Conditions iccentrated Load ssure iccentrated Flow e Pressure iface Material Pressure Surfaces pressure Label: dsload_S Pressure: 1000 kPa.	Geometry Utilities Data	a Mesh Calculate Help Problem type Image: Conditions Image: Concentrate of the concente of the concentrate of the concentrate of the concen	onditions ed Load ed Flow aterial Sourdary Conditions Surfaces-Fixed-Displacement Labet Constrain X Direction Ux 0 m (a) Apply the Y Direction Uy 0 m Z Direction Uz 0
(a) Assign the pressure load in the top surface of the model, selecting the corresponding entities.	<u>Assign</u> Entities ▼ <u>D</u> raw ▼ <u>U</u> nassign ▼ <u>C</u> lose		(b) Assign boundary conditions	Assign Entities ▼ Draw ▼ Unassign ▼ Close



Steps – General Problem Data

	GID x64 Project: cargatop (GeMa)	
(5) Set the general problem data	Files View Geometry Utilities Data Mesh Calculate Image: Conditions Image: Conditions Image: Conditions Image: Conditions Image: Conditions Image: Conditions	e Help Layer0
	Materials > Interval data > Problem data Data units Interval > Local axes >	(a) Select the problem data icon Problem data
General data Numerical Solver	№ ? 2 -	General data Numerical Solver
description Fem-Model created in GID Autorun GeMa: type Name: PLANE STRAIN isoParametric: Element Rules: DEFAULT	(b) Select the analysis type (plane strain, plane stress) and also the integration rules for the GeMA analysis.	solver Options: transient nonlinear mechanic tolerance 1e-5 hydraulic tolerance 1e-5 timeMax 1e9 \$ timeInitInc 1 \$ timeMinInc 0.01 \$ timeMaxInc 1e6 \$ iterationsMax 15 eulerTheta 1 Newton Raphson Solver: full \checkmark
Ac	cept <u>C</u> lose	<u>A</u> ccept <u>C</u> lose





Steps – Mesh and GeMA input files







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/m ³)	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







▼ Exchange

Steps – Pressure Load and Boundary Conditions

(3) Pressure Load

Data Mesh Calcu	ılate Help	
Problem type 🔹 🕨	📚 Layer0	▼ ③ ?
Conditions Materials Interval data Data units Interval Local axes	Boundaries Conditions Concentrate Load Pressure Concentrate Flow Pore pressure Interface Material	Pressure Surfaces pressure Label: dsload_S Pressure:-10 kPa.

(a) Assign pressure load in the top surface

(4) Boundary Conditions

Data Mesh Cal	culate Help	
Problem type	🕨 🎽 base	▲ 1 38 3
Conditions Materials Interval data Problem data Interval Local axes	Boundaries Con Concentrate Lo	ditions ad Boundaries Conditions Surfaces-Fixed-Displacement Label: Constraint X Direction X Y Direction X Z Direction
		Assign Entities
		Close

(a) Assign boundary conditions



Steps – Pore Pressure

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

(5) Apply pore pressure in	Data Mesh Calculate Help		Pore pressure	×
the top surface	Problem type 🔸 📚 Layer0			
	Conditions Boundary Conditions Materials Concentrated Load Interval data Pressure Concentrated Flow Pore Pressure Data units Interface Material Interval Interface Material		Surfaces Pore Label: pore-1 ▼ Pore matrix Pm 15 kPa. Pore fracture	k? 2 ▼
	(b) Assign the pore press in the correct region of th selecting the correspondi	ure values le model, ng surface	Assign Entities Draw VUn Close	assign





Steps – General Problem Data

	GID x64 Project: cargatop (GeMa)				
	Files View Geometry Utilities Data Mesh Calculate Help				
(6) Set the general	🌔 🎾 🍛 🎧 🎧 🥁 🕂 🛛 Problem type 🔸 🍃 Layer0				
problem data	🔎 🌒 Conditions 🕨				
P	→ Materials →				
	Problem data				
	Data units (a) Se	elect the problem data icon			
	Interval 🕨				
	💁 ː Eː Local axes 🕨		Problem data		
	X				
					▶? 🕗 🔻
	Problem data		General data Numerical S	iolver	
	▶? 🕗 🕶				
	General data Numerical Solver		solver Uptions:	transient nonlinear	–
	description Fore Model association CID CoM		mechanic tolerance	1e-5	
			hydraulic tolerance	1e-5	
	Autorun Gema:		timeMax	605 s	
	type Name: 3D 🔻		timeInitInc	1 s	
	isoParametric:		the shift of the	0.01	
	Element Rules: DEFAULT 💌		timemininc	0.01 S	
			timeMaxInc	10 s	
			iterationsMax	15	
			eulerTheta	1	
	(b) Select the analysis type (plane		Newton Raphson Solver:	full 🔻	
	strain, plane stress, 3D,				
	axisymmetric) and also the		(c) Set th	e tolerances and sor	ne
	integration rules for the GeMA		optio	ns for the analysis.	
	analysis.				
		-	A	ccept <u>C</u> lose	
	<u>Accept</u>		L		



Steps – Mesh

(7) Generate mesh and GeMA input files

۶

٠

٠





Click on 'Calculate' to generate GeMA input files.





Example 3 – Hydro-Mechanical Test – 3D



Parameters	Example 3
Young's modulus, <i>E</i> (kPa)	1.00E+03
Poisson's ratio, v	0.3
Hydraulic Permeability in x, k (m/s)	1.16E-05
Specific weight of water, γ_w (kN/m ³)	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







Steps – Pressure Load and Boundary Conditions

(3) Pressure Load	
(3) Pressure Load	Pressure Surfaces pressure Label: dsload_S Pressure_10 KPa
(a) Assic	gn pressure load

in the top surface

(4) Boundary Conditions





Steps – General Problem Data

	GiD x64 Project: cargatop (GeMa)			
	Files View Geometry Utilities Data Mesh Calculate Help			
(5) Set the general	🌔 📁 🎯 🐼 🐼 🥁 🕂 🛛 Problem type 🔸 🍃 Layer0			
problem data	🔎 🎍 Conditions 🕨			
	Materials			
	Problem data			
	Data units (a) Se	elect the problem data icon		
	Interval			
	Local axes		Problem data	×
	Problem data			
			General data Numerical Solv	ver
			solver Options:	transient nonlinear 🛛 🔻
	General data Numerical Solver		manhania talaranga 1a	5
	description Fem-Model created in GID-GeM			
	Autorun GeMa:		hydraulic tolerance Te-	
	type Name: 3D 🔻		timeMax 60	5s
	isoParametric:		timeInitInc 1	S
			timeMinInc 0.0	D1 s
			timeMayInc 10	
			iterationaliter 15	
	(b) Select the analysis type (plane		euler i heta 1	
	strain, plane stress, 3D,		Newton Raphson Solver:	full 🔻
	axissymetric) and also the		(a) Set the	tolerences and some
	integration rules for the GeMA		(c) Set the	for the enclusion
	analysis.		options	
			Acc	ept Close





Steps – Mesh

(6) Generate mesh and GeMA input files





Click on 'Calculate' to run the analysis.





Gid-GeMa including interface elements



SGENERAL_D.	ATA										
NPOTN											
THE OTH	NELEM	NMATS	NDIME	NNOMAX	NELIN	Т					
1086	362	3	2	3	51	5					
%FACE DATA											
iface	Lmat	Lcodeh	Lcodem I	Loodet n	noface	nnofacec	ngauss	no(1)	no(2)		
1	3	1	4	1	2	2	2	594	604		
%NODES											
ID			x			Y		Z			
1		-5.2	43260		-2.394	710	0	.000000			
%ELEMENTS											
6400	8										
Ielem	Coned	ctivitie	s	Lmat N	nom Nno	f					
1	16724	16770	16772	16741	1674	9 1677	4 16779	5 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):



File with the created interface elements





Example 4 – Mechanical Test – 2D

P = 1 kPa



Parameters	Example 4
Young's modulus, <i>E</i> (kPa)	1.00E+06
Poisson's ratio, v	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, <i>s</i> (m)	10.0





Steps – Problem Type

(1) Set the GeMA problem type



(2) Duplicate the nodes from lines







Steps – Material

(3) Generate material







Create new material

🗡 💷 🐶 🖉 🔻

Material

Mechanics Hydraulic data

Rock

- 🧭 🜔

Steps – Material

(3) Generate material









Steps – Conditions







Steps – Conditions

(4) Attach conditions for the interface material







Steps – Constraints

(5) Apply the boundary conditions of the problem





Steps – Pressure Load





Steps – General Problem Data

General data Numerical Solver description Fem-Model created in GD-C autorun GeMa: type Name: type Name: PLANE STRAIN ▼ isoParametric: (b) Select the analysis type (plane strain, plane stress) and also the integration rules for the GeMA analysis. IterationsMax 16 Solver options of the analysis. With the data integration rules for the GeMA analysis. Newton Raphson Solver full ▼	(7) Set the general problem data	Files View Geometry Utilities Data Mesh Calculate Help View Geometry Utilities Data Mesh Calculate Help Problem type Layer0 Conditions Materials Interval data Data units Interval • Local axes •	the problem data icon
solver Options: description Fem-Model created in GID-C Autorun GeMa: type Name: PLANE STRAIN ▼ isoParametric: Element Rules: DEFAULT ▼ (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. timeMaxle 9 \$ timeMaxle 0 \$ timeMaxle 10 \$ timeMa	Grandate	N? [Problem data Problem data General data Numerical Solver
	General data description Autorun (type Name isoParam Element Rules	Numerical Solver In Fem-Model created in GID-C GeMa: Inetric:	solver Options: transient nonlinear mechanic tolerance 1e-5 hydraulic tolerance 1e-5 timeMax 1e9 \$ timeInitInc 1 \$ (C) Set the tolerances and some solver options of the analysis. timeMaxInc 1e6 \$ iterationsMax 15 eulerTheta 1 Newton Raphson Solver: full \checkmark





Steps – Mesh

(8) Generate mesh and GeMA input files

۲

٠

٠





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, v	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, <i>s</i> (m)	1
Specific weight of water, γ_w (kN/m ³)	10
Analysis time, <i>t</i> (s)	1.00E+09





Steps – Problem Type

(1) Set the GeMA problem type



(2) Duplicate the nodes from lines Geometry Utilities Data Mesh Calculate Help 🐼 l 🐼 🐼 👾 🕂 l 🗃 1 🐼 🤶 Unstructured -> || -- | C Structured SemiStructured Cartesian ۶ B 2 Boundary layer 83 40 Quadratic type Element type Mesh criteri Default ≞♥♥■¶¶♥♥ヲ2克※自由公務業長♥♥№≧♀■Ⅱ Reset mesh data Mesh Draw No mesh Generate mesh.. Ctrl-q Default mesh Erase mesh Skip Edit mesh No skip Show errors Automatic skip View mesh boundary Skip by... Create boundary mesh Force points to Mesh quality... Duplicate Lines Mesh options from model No Duplicate Surfaces 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







Steps – Material

(3) Generate material







Steps – Conditions







Steps – Conditions

(4) Attach conditions for the interface material







Steps – Constraints

(5) Apply the boundary conditions of the problem







Steps – Pressure Load

(6) Apply pressure load





Steps – Pore Pressure







Steps – General Problem Data

(8) Set the general problem data	Files View Geometry Utilities Data Mesh Calculate Help View Geometry Utilities Data Mesh Calculate Help Problem type View Geometry Utilities Conditions Conditions Materials Conditions Conditions Conditions Materials Conditions Conditions Conditions Materials Conditions Conditions Conditions Materials Conditions Conditions Materials Conditions Conditions Materials Conditions Conditions Materials Conditions Conditions Materials Conditions Conditions Materials Conditions Conditions Conditions Conditions Conditions Materials Conditions	problem data icon
General data descriptio Autorun type Nam isoParam Element Rule	Numerical Solver Numerical Solver Merician Metrician	General data Numerical Solver General data Numerical Solver solver Options: transient nonlinear mechanic tolerance 1e-5 hydraulic tolerance 1e-5 timeMax 1e9 \$ timeMax1e9 \$ timeMinlc 0.01 \$ timeMaxlnc 1e6 \$ iterationsMax 15 eulerTheta 1
	<u>A</u> ccept <u>C</u> lose	Accept <u>C</u> lose





Steps – Mesh

(9) Generate mesh and GeMA input files

۲

٠





Click on 'Calculate' to generate GeMA input files.



