Step-By-Step Guide To Simulate Slope Model Using CUBIT and SPEPECFEM3D_GEOTECH

STEP 1. Create a CUBIT journal file "cubit_example.jou" with following content:

```
#--------------BEGIN "cubit_example.jou"-------------------------------------------
# create model
create vertex 0 0 0
create vertex 50 0 0
create vertex 50 0 6
create vertex 43 0 6
create vertex 18 0 18
create vertex 0 0 18
create curve vertex 1 2
create curve vertex 2 3
create curve vertex 3 4
create curve vertex 4 5
create curve vertex 5 6
create curve vertex 6 1
create surface curve 1 2 3 4 5 6
sweep surface 1  vector 0 1 0 distance 20
compress all
# mesh surface
surface 8 size 2
surface 8 scheme pave
mesh surface 8
# mesh volume sweeping surface mesh
volume 1  size 2
volume 1 redistribute nodes off
volume 1 scheme Sweep  source surface 8    target surface 1   sweep_smooth Auto  sweep_transform
least_squares  autosmooth_target off
mesh volume 1
# define block
set duplicate block elements off
block 1 volume 1
# define boundary conditions
Sideset 1 surface 2
sideset 1 name 'bottom_ssbcux_ssbcuy_ssbcuz'
Sideset 2 surface 8
sideset 2 name 'front_ssbcuy'
Sideset 3 surface 1
sideset 3 name 'back_ssbcuy'
Sideset 4 surface 7
sideset 4 name 'left_ssbcux'
Sideset 5 surface 3
sideset 5 name 'right_ssbcux'
```

```
compress all
# save and export mesh
save as "cubit_example.cub" overwrite
set large exodus file off
export mesh "cubit_example.e" overwrite
#-----------------END "cubit_example.jou"-------------------------------------
```

Note: Above procedure can also be done manually using CUBIT's GUI.

STEP 2: Convert "Binary" EXODUS file to "ASCII" format

ncdump cubit_example.e > cubit_example.txt

Note: You must have installed NetCDF libraries. NetCDF is a free software which can be downloaded from
http://www.unidata.ucar.edu/downloads/netcdf/index.jsp

STEP 3: Compile exodus2sem tool which is located at utilities/ folder

gcc exodus2sem.c -o exodus2sem

Note: For more information, please check the header in exodus2sem.c file.

STEP 4: Convert exodus file to SEM files

./exodus2sem cubit_example.txt

STEP 5: Copy all generated files to input/ folder

STEP 6: Create a material properties file "cubit_example_material_list" in input/ folder with the following content:

```
# material properties (id,domain,gamma,ym,nu,phi,coh,psi)
1
1, 1, 18.8, 1e5, 0.3, 20.0, 29.0, 0.0
```

STEP 7: Create a main input file "cubit_example.sem" in input/ folder with the following content:

```
#-----------------BEGIN "cubit_example.sem" ----------------------------------------------------
#pre information
preinfo: method='sem', ngllx=3, nglly=3, ngllz=3, nenod=8, ngnod=8, inp_path='./input',
out_path='./output/'

#mesh information
mesh: xfile='cubit_example_coord_x', yfile='cubit_example_coord_y', zfile='cubit_example_coord_z',
&
```

```
confile='cubit_example_connectivity', idfile='cubit_example_material_id'

#boundary conditions
bc: uxfile='cubit_example_ssbcux', uyfile='cubit_example_ssbcuy', uzfile='cubit_example_ssbcuz'

#material list
material: matfile='cubit_example_material_list'

#control parameters
control: cg_tol=1e-8, cg_maxiter=5000, nl_tol=0.0005, nl_maxiter=3000, nsrf=9, srf=1.0 1.5 2.0 2.15 2.16 2.17 2.18 2.19 2.20

#save data options
save: disp=1
#----------------END "cubit_example.sem" ------------------------------------------------
```

STEP 8: Run prgroam

go to SPECFEM3D_GEOTECH folder in terminal and type

./bin/semgeotech ./input/cubit_example.sem

Note: You must have compiled SPECFEM3D_GEOTECH. For more detail please see manual.

STEP 9: Visualize

Start ParaView and Open "output/cubit_example.case" file

Note: You must have installed ParaView. ParaView is a free opensource visualization software which can be downloaded from http://www.paraview.org/.