FEFLOW PLUG-IN FOR FULLY COUPLED LAND SUBSIDENCE MODELS BASED ON BIOT'S CONSOLIDATION THEORY
USER MANUAL

1 INTRODUCTION

The *Subsidence* Plug-In (so-called *SUB*) for FeFlow is developed based on the poroelasticity theory (or Biot's theory) with aims to solve hydro-mechanical coupled problems such as land subsidence due to groundwater extraction, consolidation analysis. Another stand-alone program so-called *PSUB* is also developed to visualize results of *SUB.*

The main features of the *SUB* are:

- **Fully coupled:** *SUB* is developed based on the Biot's theory.
- **Easy to use**: For FeFlow users, *SUB* is simple to use, just load Plug-In and run. All the third party libraries are linked statically. Hence, *SUB* has only one .dll file.
- **Fast**: *SUB* is developed with C++, the matrix operations and sparse matrix solver are supported by Eigen and Intel MKL libraries. These guarantee that *SUB* is fast. Moreover, the global stiffness matrix also is assembled using multi-threads.
- **Cross-platform**: *SUB* is developed with C++ under Qt framework. Hence, it can be complied both on Linux and Window.
- **Reliable**: *SUB* is verified with analytical solutions of Terzaghi's and Deleeuw's problems.
- **Free**: *SUB* is open-source with LGPL license.

2 INSTALLATION

2.1 For developers

For developers who want to add more features to *SUB* and *PSUB*, the following components need to be installed:

- Qt-5 static version: https://www.qt.io/download
- Eigen library: http://eigen.tuxfamily.org/index.php?title=Main_Page
- Intel MKL: https://software.intel.com/en-us/mkl
- FeFlow SDK
- Visual Studio C++ or g++ can be used as a compiler.

In .pro file, the paths of Eigen library, Intel MKL, and FeFlow SDK must be modified accordingly.

2.2 For Window end-users

For Window end-users, *SUB.dll* and *PSUB.exe* can be stored anywhere and can be used directly without any configuration.

3 STEP-BY-STEP TUTORIAL

3.1 Load Plug-In

When *SUB.dll* is loaded, *SUB.dll* needs to be activated for the first time (Figure 1).

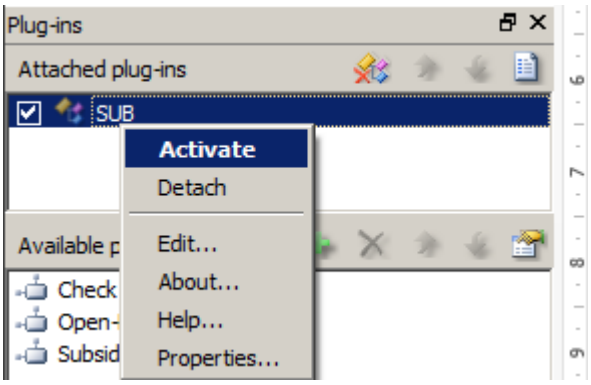

Figure 1: Attach and Active *SUB.dll*

After activating SUB.dll, in the FeFlow .fem folder (current project folder), two folders are created: *Subsidence_Outputs* and *Subsidence_Settings* (Figure 2).

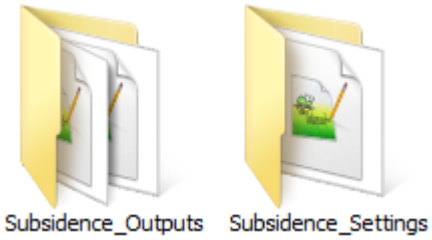

Figure 2: Two folders are created after activating SUB.dll

The *Subsidence_Outputs* folder (Figure 3) contains all results which are generated by *SUB.dll* that are:

- *coordinates.dat*: ASCII file contains nodal coordinates (X, Y, Z)
- *elements.dat*: ASCII file contains element information
- *properties*.dat: ASCII file contains material properties of each element.
- *Nodal_Solutions* folder includes *Head.txt, X-displacement.txt, Y-displacement.txt, Z-displacement.txt.* Those are nodal solutions of all nodes.
- *Element_Stress* folder stores all the element stress results: *Sxx.txt, Syy.txt, Szz.txt, Sxy.txt, Syz.txt, Sxz.txt.*

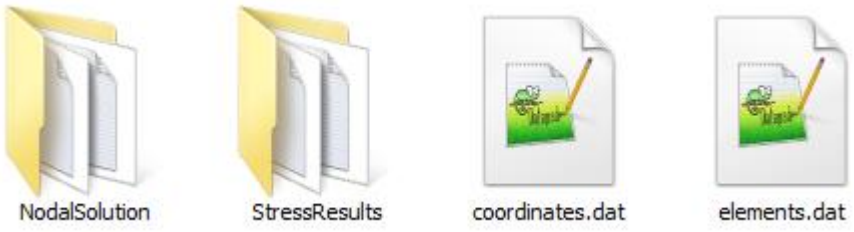

Figure 3: Results of SUB.dll are stored in *Subsidence_Outputs* folder

In the *Subsidence_Settings* folder, the *SUB_setting.dat* is created. This file is used to control extra settings of *SUB* (Figure 4). When this file is deleted, the default values are:

- Results are written for all calculation step
- Element stress is saved (*Element_Stress* folder).
- Material properties are not saved (*properties.dat*)
- Mesh information is saved (*coordinate.dat* and *elements.dat*).

```
#WRITE_RESULTS_AFTER_Nth_STEP 1
#WRITE_STRESS_RESULTS_YES/NO YES
#WRITE_MATERIAL_INFORMATION_YES/NO NO
#WRITE_MESH_INFORMATION_YES/NO YES
```

Figure 4: Control extra settings of *SUB* via *SUB_setting.dat* file

3.2 Assign Material Properties

The material properties are assigned for each element in models. For fully coupled analysis, necessary input parameters are:

- The hydraulic conductivity K_xx, K_yy, K_zz in x,y, z-direction: These parameters are also used by FeFlow.
- After activating *SUB.dll*, the user elemental data *Bulk_Modulus*, *Porosity*, *Grain_Compressibility*, *Poisson_Ratio* are created. These are used to assign bulk modulus *K*, porosity *n*, the compressibility of soil skeleton C_s, and Poisson's ratio μ.

Figure 5: User elemental data that are generated after activating *SUB.dll*

3.3 Assign Boundary Conditions

The boundary conditions are assigned for nodes in models. For fully coupled model, the hydraulic boundary conditions are the initial head, the fixed hydraulic head (Dirichlet type), and well boundary conditions (extract rate). These are already available in FeFlow.

SUB.dll creates three different user nodal data to store and assign boundary conditions for the mechanical field that are *X_Displacement, Y_Displacement,* and *Z_Displacement.*

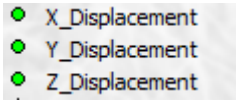

Figure 6: User nodal data created by *SUB.dll*

3.4 Run Simulation

After all input parameters are prepared; the simulation process can be started by pressing *Run* Button. The log-panel is used to show information during the simulation process (Figure 7). When the simulation is finished, all results are saved in *Subsidence_Outputs* folder.

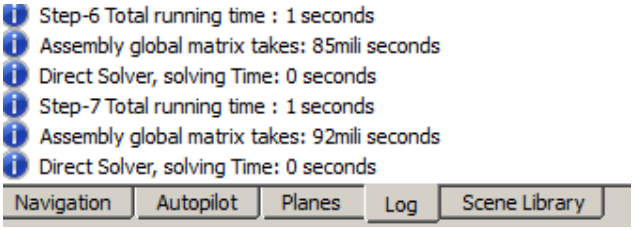

Figure 7: Showing information using the Log-panel

3.5 View results

Results can be visualized by using the stand-alone program *PSUB.exe* (Figure 8).

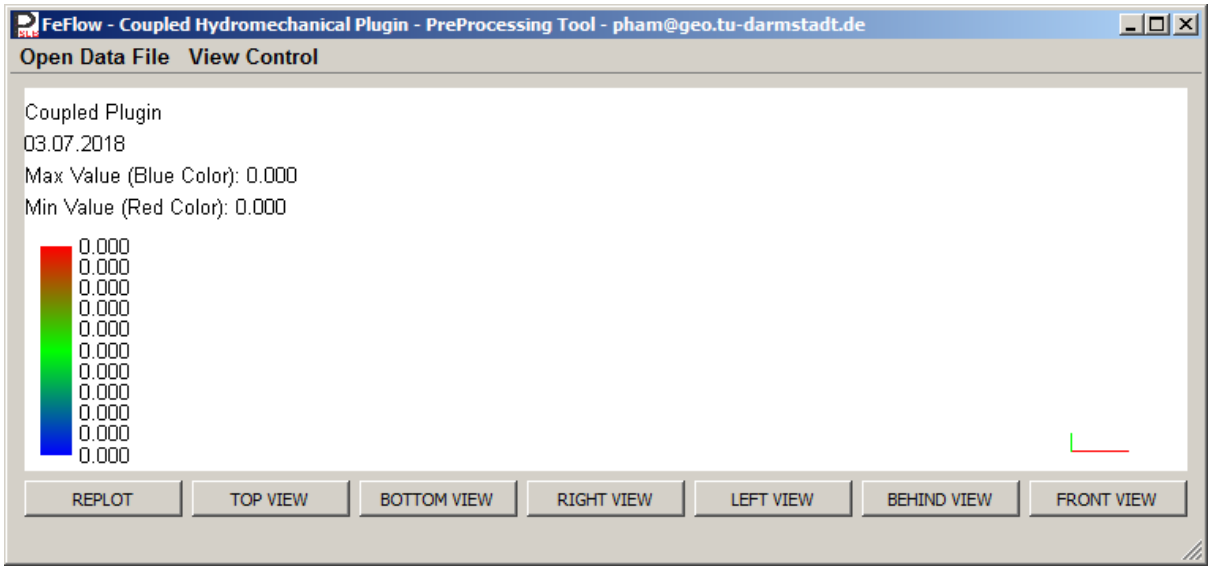

Figure 8: The user interface of *PSUB.exe*

In *PSUB.exe*, the mesh data needs to be loaded first. The mesh data is *coordinates.dat* and *elements.dat* which are stored in *Subsidence_Outputs* folder (Figure 9).

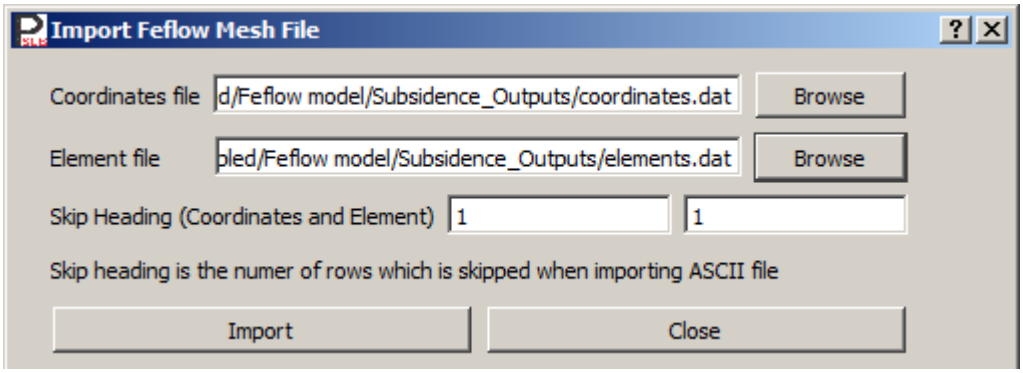

Figure 9: Load mesh data

Then, the nodal results that are saved in *Subsidence_Outputs/Nodal_Solutions* are loaded (Figure 10).

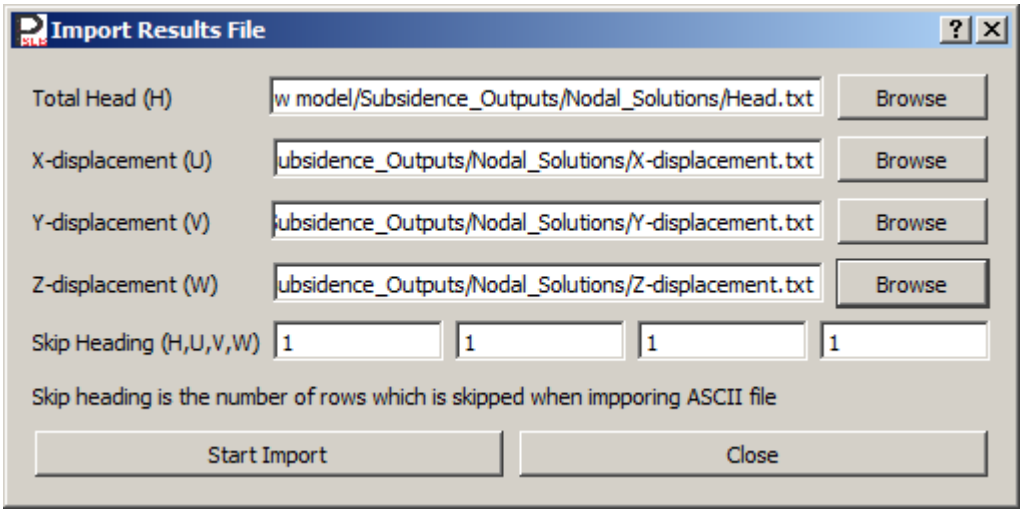

Figure 10: Load nodal solutions

When all results are imported, they can be viewed with many options as Figure 11:

- Scale coordinate, deformation
- Changing contour type
- Specific calculation step
- Animation

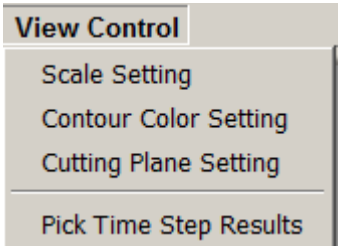

Figure 11: Visualize results with *PSUB* program