User Manual and Keywords for the Interface of FRONTS3D

This user manual provides information about how to run FRONTS3D interface in the windows environment and some details of the input data file and other information.
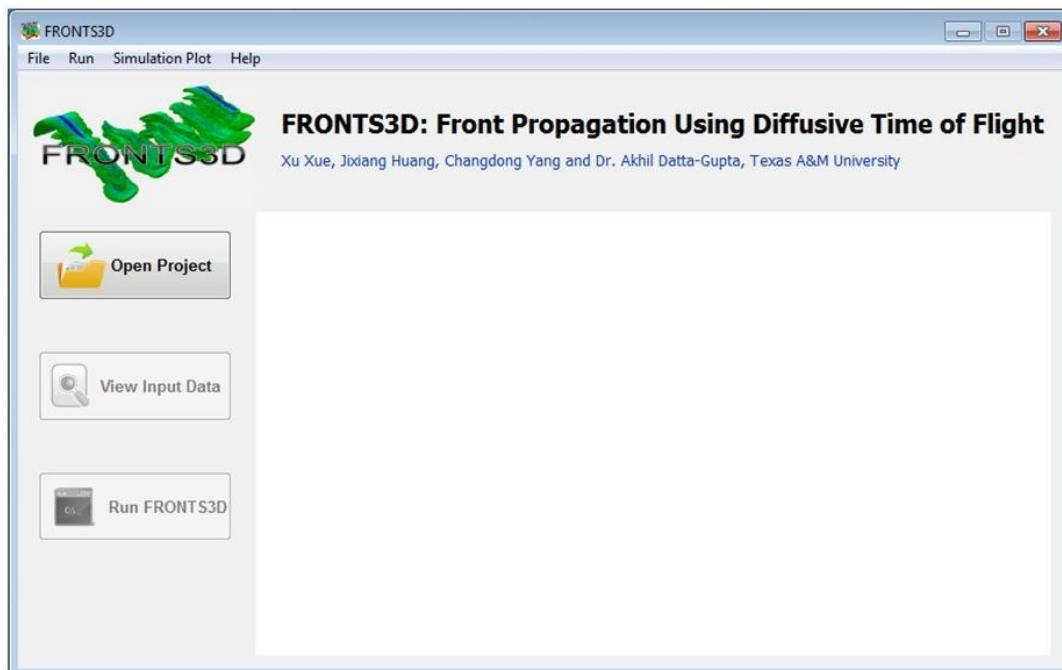
1. Installation:

Double click on FRONTS3D.exe.

It will guide you through all the steps to set up the software in your computer. It is recommended that you install the program with a simple path name, for example, C:/ FRONTS3D.
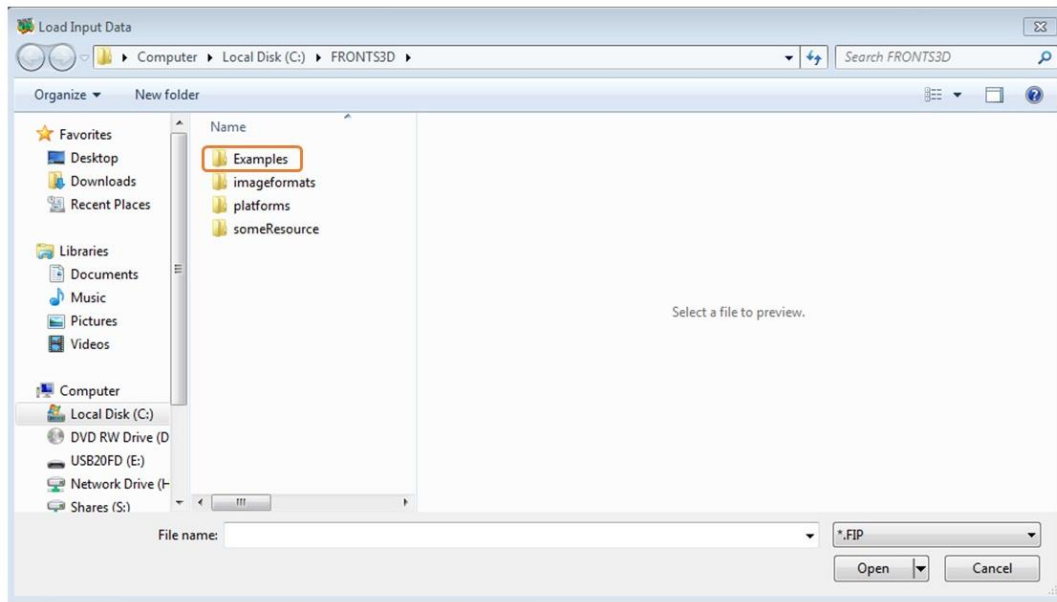
2. After installation is complete:

1. From Start →Programs, click on FRONTS3D or directly click on FRONTS3D from the desktop.
2. Once the program starts, the main screen appears as below (**Figure 1**).
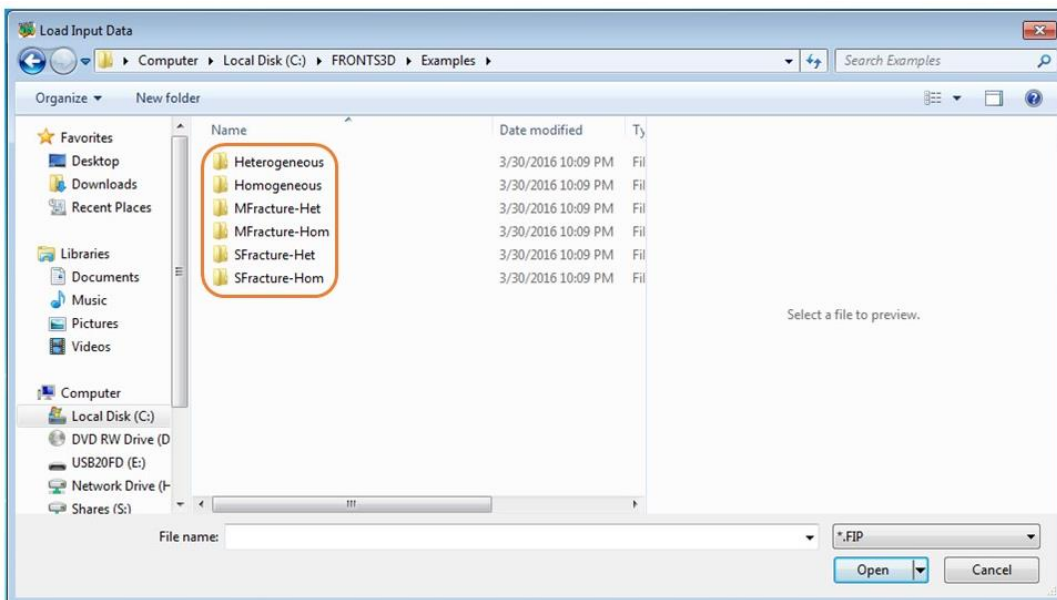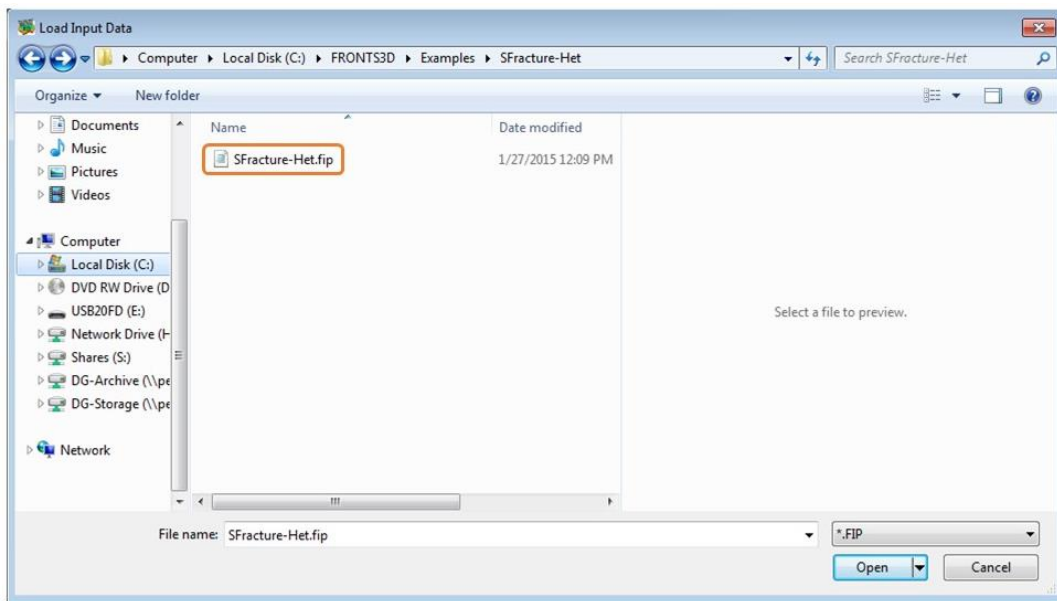

Figure 1. Starting Screen

3. To load a new project, click File →Open Project or click Open Project button on the main screen. Then click on the Examples folder to select the case you want and finally click on the case file. All the suffix of data file should be .fip in order to run the FRONTS3D interface. See **Figs. 2, 3 & 4** for sequence to load a new project.

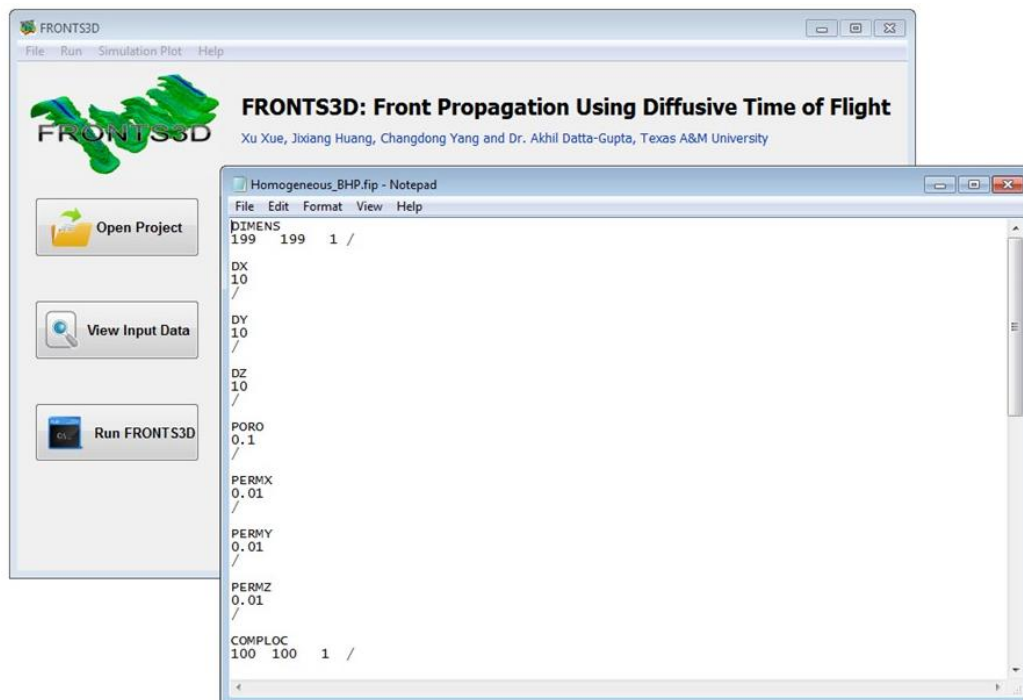**Figure 2. Select Case Folder**

Figure 3. Select Specific Case

Figure 4. Select .FIP Data File

4. Once the data file is loaded into the interface, you can click View Input Data button on the main screen to view and modify the input data file. **(Figure 5).** You can check **Keywords Reference Manual** at the back for the keywords appear in the input data file.
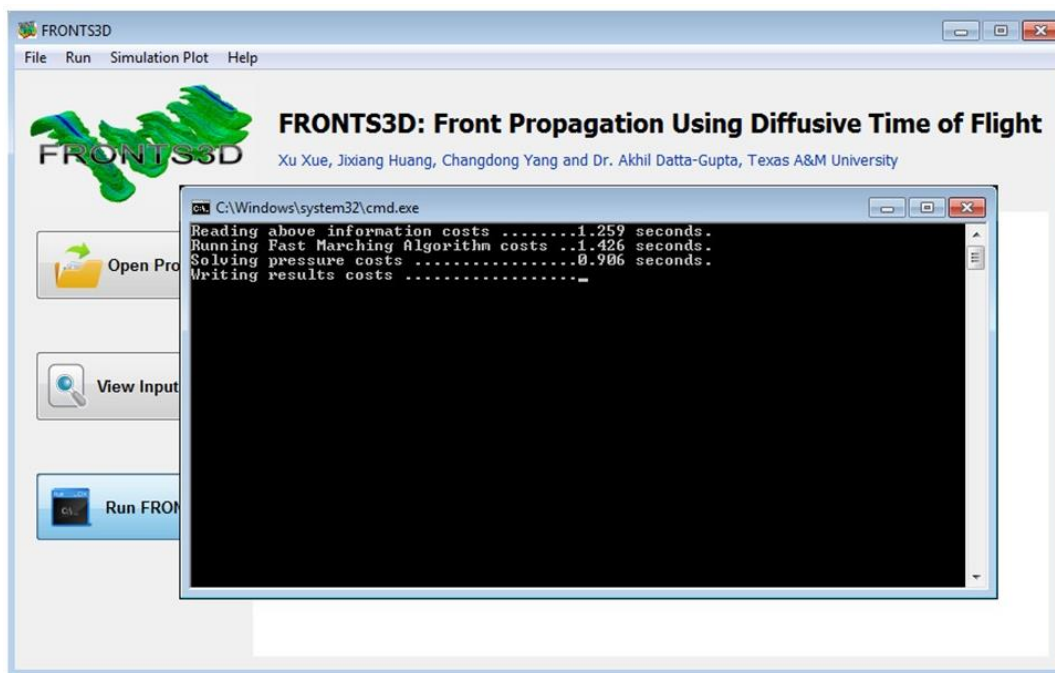
5. Once modifications are complete (modified files need to be saved), the project is ready to be executed. For execution, click Run FRONTS3D button on the main screen or click Run → Run F3D (or Run F3D Parallel with two running simulations). A DOS window will appear indicating progress of the simulation. **(Figure 6).** After successful completion of the execution, the DOS window will close with a message 'Simulation Complete'.

Note:

If there is some mistake in the input data file, the DOS window will close with a message 'Simulation could not progress'. Then you need to check the modified input data files.

Figure 6. Simulation is in Progress

6. Now the user is ready to view the results. Followings are the major points to consider while viewing the results: Res

a. Click Simulation Plot → Reservoir Data, you can view initial data plots in the reservoir such as permeability and porosity. **(Figure 7).**

b. Click Simulation Plot → Drainage Volume, you can view DV vs. Tau or Time. **(Figure 8).**

c. Click Simulation Plot → Pressure Depletion, you can view bottomhole pressure and pressure field. **(Figure 9).**

d. Click Simulation Plot → Rate Approximation, you can view production rate and cumulative production. **(Figure 10).** Note: You need to select the data you want to see and click the Plot button, then you can see the plot.
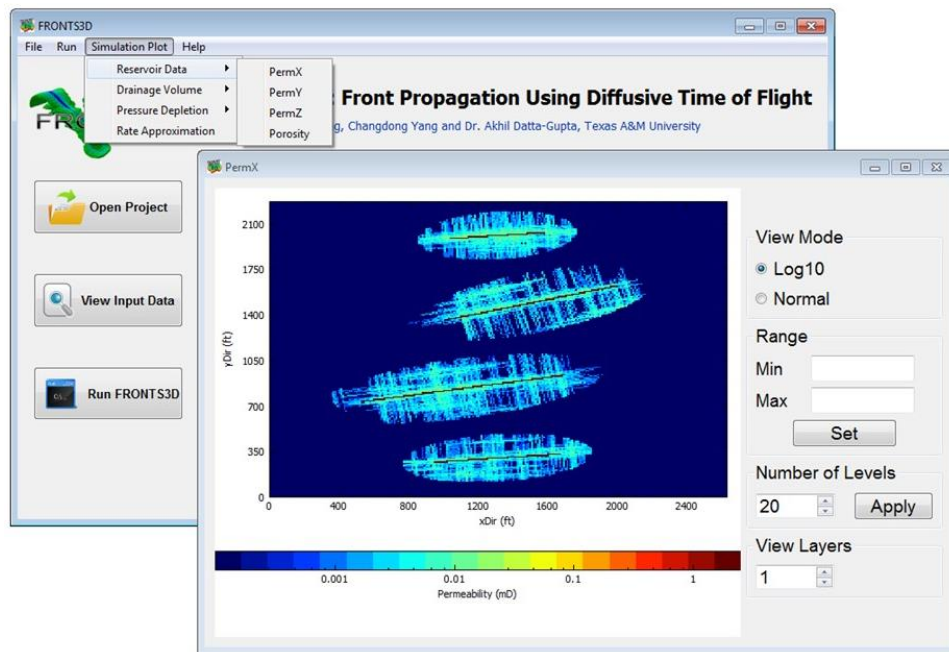

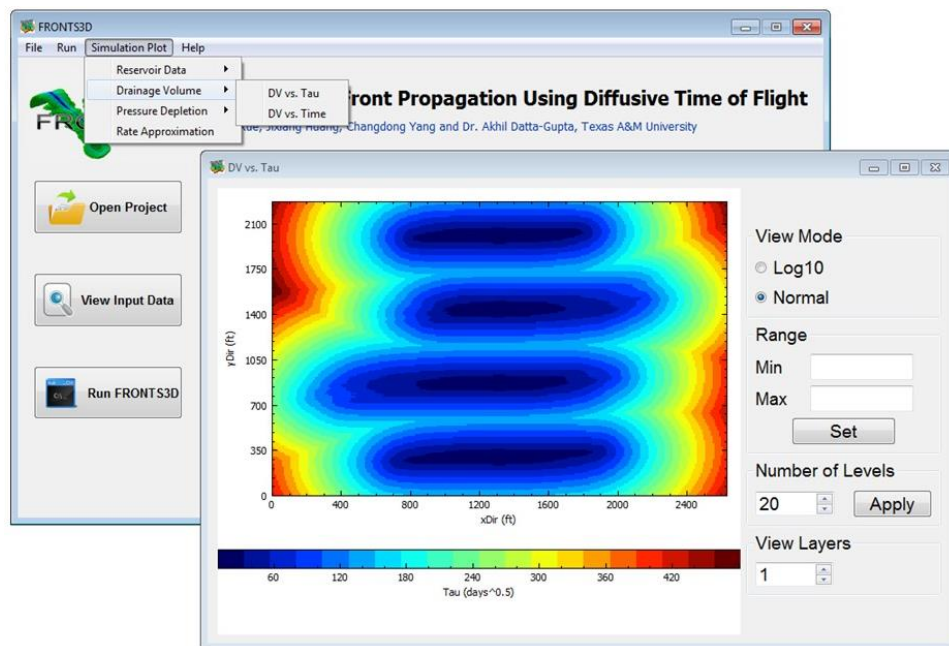**Figure 7. Select and View Initial Reservoir Data**
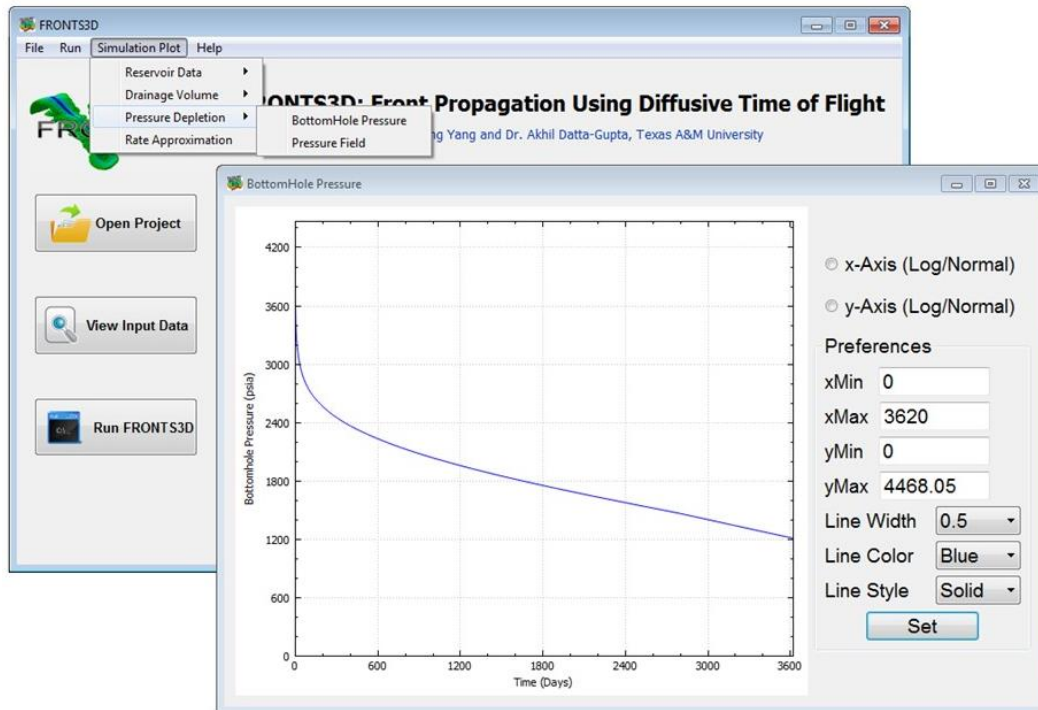

Figure 8. Select and View Drainage Volume
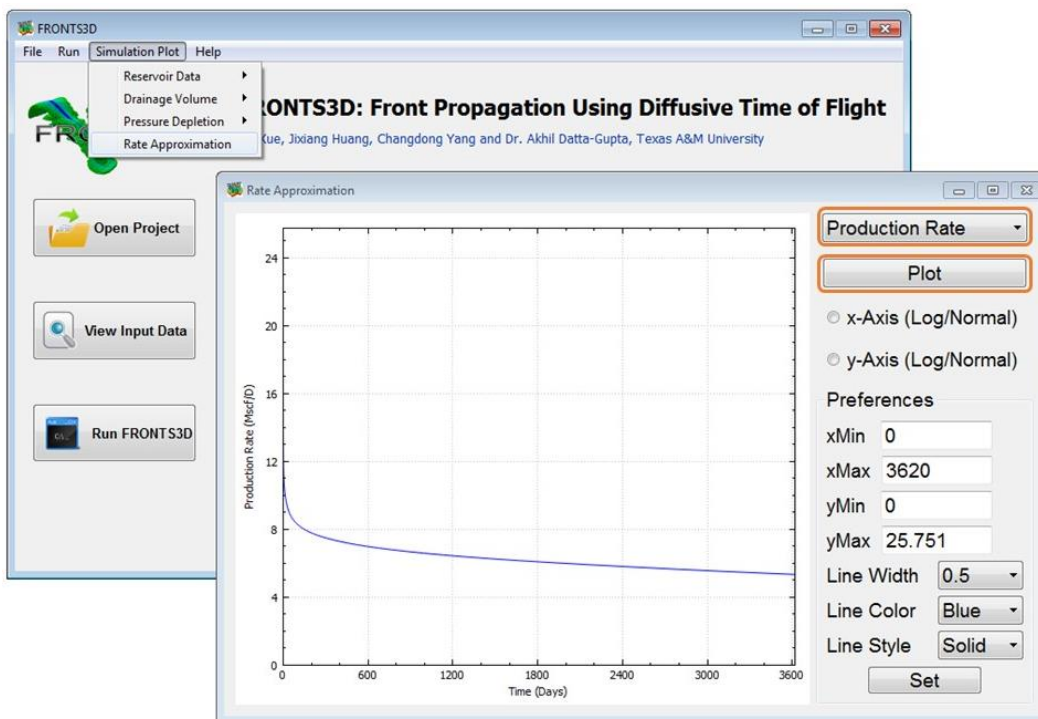
Figure 9. Select and View Pressure Depletion

Figure 10. Select and View Rate Approximation

Appendix A: Keywords Reference Manual

Field unit is used in all the input data.

DIMENS

This keyword is used to specify the number of grids in x, y and z direction.

DX, DY, DZ

These keywords are used to specify the distance of the grids in x, y and z direction.

PORO, PERMX, PERMY, PERMZ

These keywords are used to specify the initial values of the reservoir. INCLUDE is used to specify there are other data files to specify the initial values of each grid.

COMPLOC

This keyword is used to specify the completion well location. Column 1, 2 and 3 are used to specify the location in x, y and z direction.

INIT_TOTAL_COMPRESSIBILITY

This keyword is used to specify the initial total compressibility.

INIT_VISCOSITY

This keyword is used to specify the initial viscosity of fluid.

SIMULATION

This keyword is used to request the pressure calculation. Without this keyword, the simulator only output the diffusive time of flight and physical arrival time at each grid.
The keyword is followed by including either the file that contains simulation data or directly the required simulation data.

RPTPRE

This keyword is used to request the output of field pressure.
The keyword is followed by the keyword "FREQ=N", where N is the output frequency for time steps. The default value is 100.

NTAU

This keyword is used to specify the number of grids in 1D discretization.

NTIME

This keyword is used to specify the physical time of the pressure front.

OIL

This indicates that a run contains an oil phase.

GAS

This indicates that the run contains gas.

PRESSURE

This keyword is used to specify the initial pressure.

RCOMP

It is followed by the reference pressure and corresponding compressibility, terminated by a slash (/).

DENSITY

This keyword is used to specify the three fluid densities at standard condition, terminated by a slash (/).

The table consists of the following 3 items of data:

Item 1 The density of oil

Item 2 The density of water

Item 3 The density of gas

PVDO

This keyword is used to specify the dead oil (no dissolved gas) PVT table, terminated by a slash (/).

The table consists of 3 columns of data:

Column 1 The oil phase pressure

Column 2 The corresponding oil formation volume factor

Column 3 The corresponding oil viscosity

PVDG

This keyword is used to specify the dry gas PVT table, terminated by a slash (/).

The table consists of 3 columns of data:

Column 1 The gas phase pressure

Column 2 The corresponding gas formation volume factor

Column 3 The corresponding gas viscosity

WCONPROD

This keyword is used to specify well schedule table, terminated by a slash (/).

The table consists of 5 columns of data:

Column 1 The interval simulation time in days

Column 2 The well control mode:

GRAT: Controlled by oil rate target

ORAT: Controlled by gas rate target

BHP: Controlled by BHP target

Column 3 The BHP target or lower limit

Column 4 The Gas or Oil rate target (Mscf/day for gas, stb/day for oil)

Column 5 The number of time discretization within the interval