NETool™ *5000.1.0 User Guide*

© 2012 Halliburton

HALLIBURTON | Landmark Software
& Services

Note

Contents

Introduction

NETool is building a numerical model based on the reservoir, completion, fluid and simulation setup information. This information is imported or defined by the user in the graphical user interface. The user is therefore shielded from the logic and algorithms described below, this is taken care of automatically by NETool.

When building the numerical model the individual components in a well completion are mapped onto a discretized network system consisting of nodes with flow connections linking the nodes.

Two examples of network system are shown in **Figure 1** and **Figure 2**.
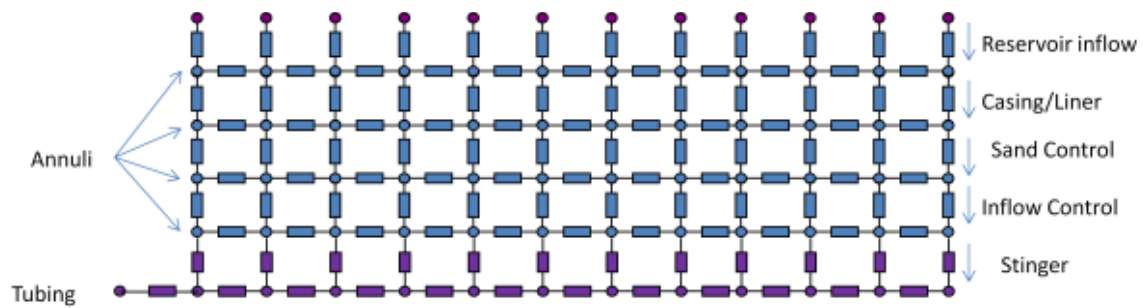
Figure 1. NETool General Node Configuration
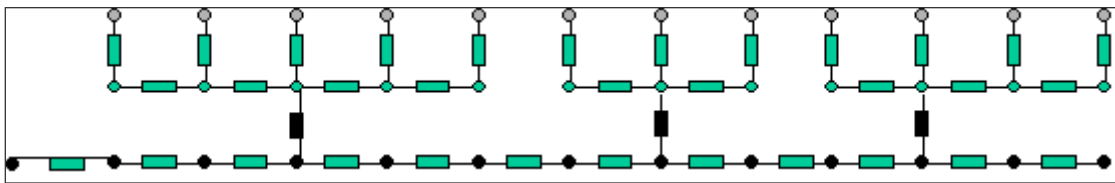
Figure 2. Node Configuration for ICV Completion

The general node configuration is presented in **Figure 1**. The uppermost row of nodes in this figure shows the layer of reservoir nodes (also called external nodes). The next four rows of nodes represent annuli within the well, and the lowermost row represents the inner production tubing. The number of annular layers dependinds on the completion type.

As mentioned above the user is shielded from the details of the numerical network shown and described above. Via the NETool user interface, the user specifies reservoir properties, completion type and configuration along the well. All parameters can be varied, mixed and combined along the well making it possible to model complex completion.

NETool defines the numerical network and the pressure drop correlations to be used for each individual flow connection. This is based on the specifications supplied by the user.

Flow between the reservoir layer and the annulus layer (top row and the next one) is specified by advanced PI correlations involving 3D permeability averaging and phase exchange effects towards the wellbore.

By configuring the completion type, the fluid flow performance between the annuli and tubing layers, within the annuli layers, and within the tubing layer can be modeled.

Figure 2 shows a node system used for Inflow Control Valve completions, which typically consists of ICVs, packers and blank pipes. This nodal network is the results of a completion where flow from a long area of the formation is controlled by a single entry point into the tubing and where the annular flow is separated from other entry points by packers.

Basic Assumptions

NETool simulations are based on the following assumptions:

- Three phase black oil based PVT formulation of the hydrocarbon fluid system with oil, gas and water. NETool can run in three-phase, two-phase or single phase modes. It allows for vaporized oil, condensates, wet gas, multiple bubblepoints and multiple dewpoints.

- Isothermal, steady-state flow conditions for sub-sonic flow conditions.

- The flow within the well is locally one-dimensional, i.e. the annulus or tubing flow between two adjacent nodes along the well is averaged over the cross section within the annulus or tubing.

- The general pressure drop calculations within the annulus and the tubing are general momentum balance equations for pipe flow (Bernoully) including friction, compressibility and hydrostatic phenomena.

- Linear Darcy flow equation is used if annulus is filled with gravel or collapsed rock.

- The reservoir flow performance is based on Productivity Index models and phase fractions. Local PIs are defined according to the average reservoir properties. Permeabilities can be locally averaged along the well trajectory. The averaging process considers heterogeneities and anisotropies.

- The pressure drop calculations through completions components are based on a variety of different configurable correlations or equations depending on the actual component.

- Flow in and out of junctions is treated by simple relationships, thus avoiding multi dimensional computations.

- The network geometry of nodes and flow channels is sufficiently general to allow simulation of most completion types, however, simple enough to satisfy the requirement of computational efficiency and numerical stability.

- There are also capabilities to simulate using single-phase, power-law type fluids, i.e. drilling muds.

Overview of Software User Interface

NETool must be populated with well information, reservoir and completion data to perform simulations. It can be populated with data from logs, spreadsheets and reservoir simulation models, or by simple manual data entry.

There are also correlations available to generate PVT and Relative Permeability data within NETool itself.

An overview of data requirements for running NETool is illustrated in **Figure 3**.
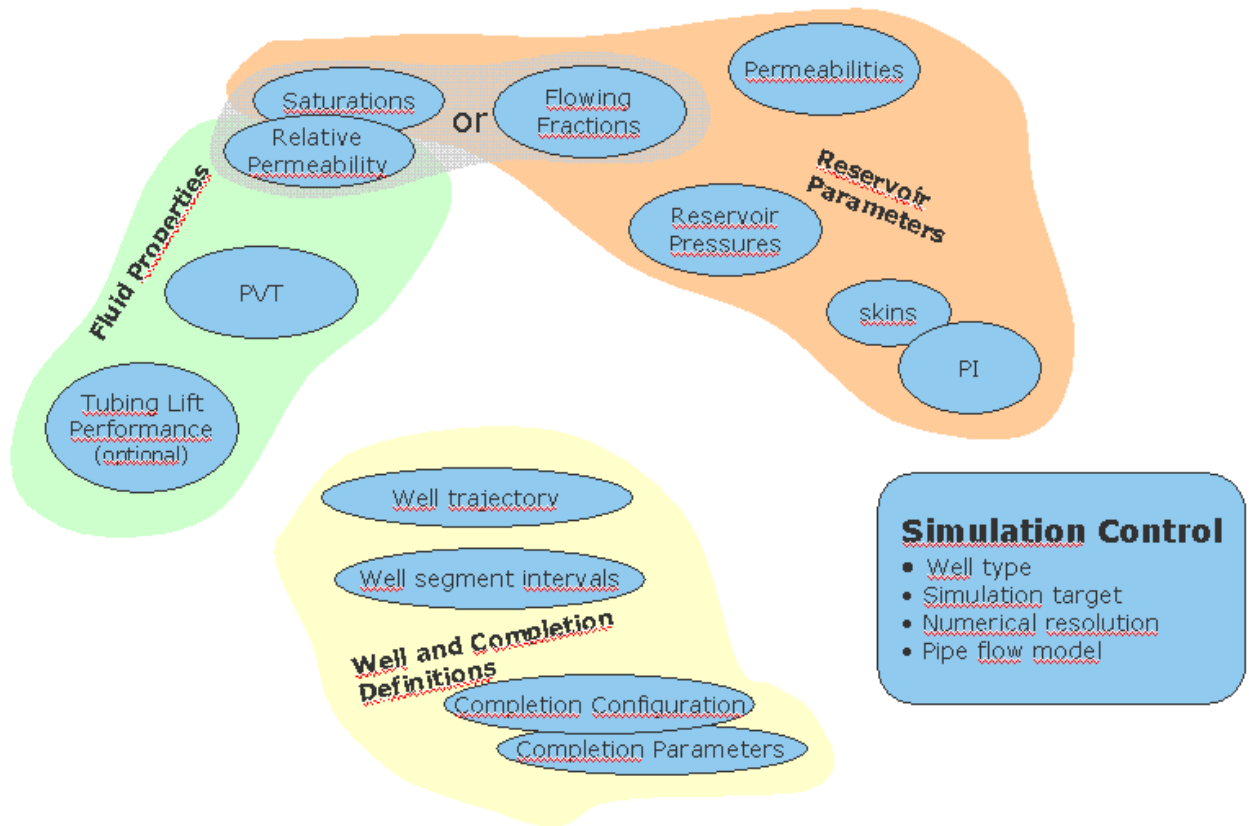
Figure 3. NETool Data Requirements

The capability to import a reservoir simulation model loads grid block dimensions, permeability, saturations, pressures and other reservoir and fluid properties. This is a very efficient way to supply NETool with the basic data. Currently NETool can import reservoir simulation model files in formats supported by the standard black-oil Eclipse E100 reservoir simulator as well as some Eclipse E300 grids.

The main menus and views in graphical user interface are shown in **Figure 4**.
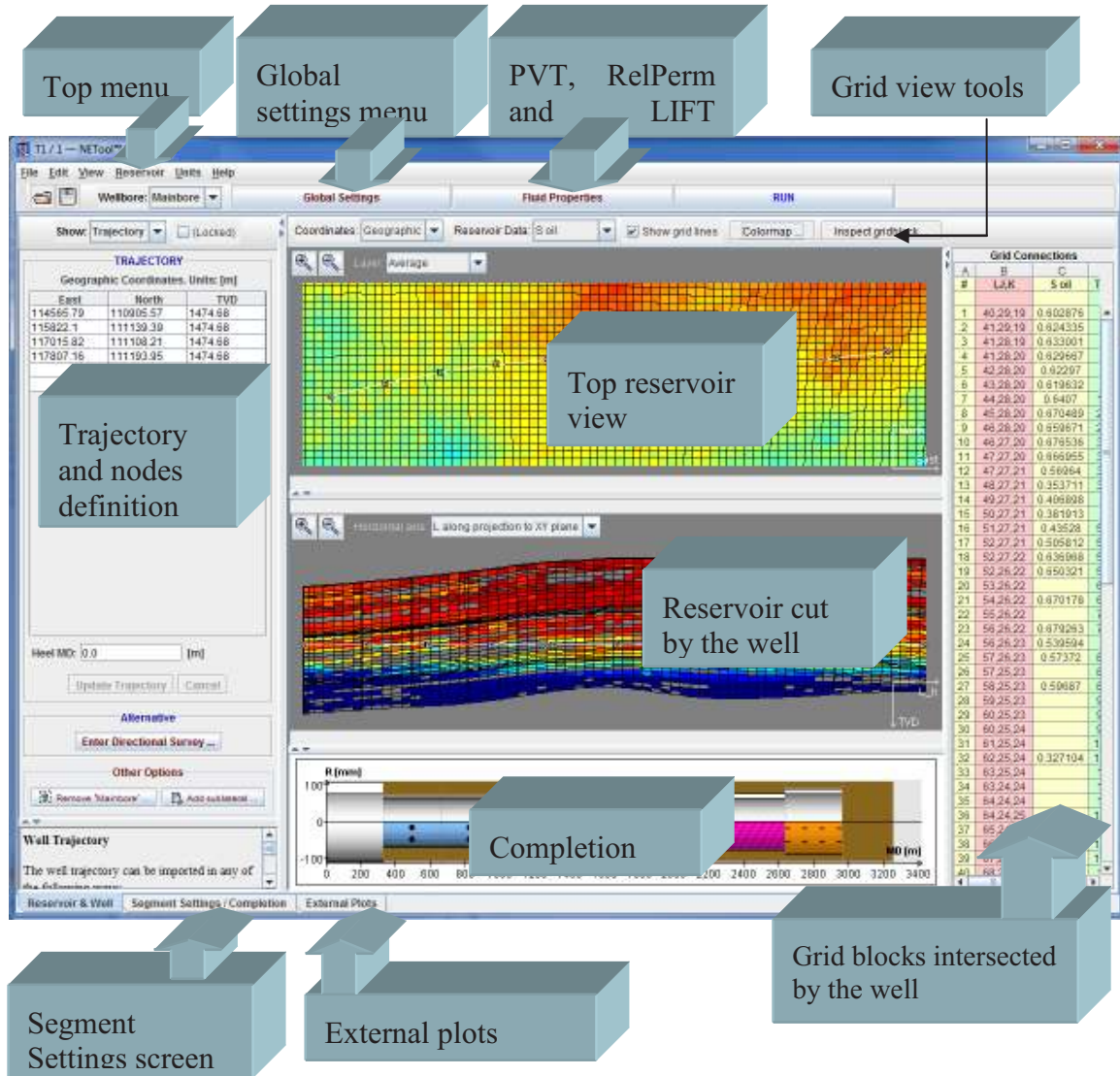
Figure 4. Main Menu and Views

Top Menu

The top menu bar contains:

File – Various capabilities to organize projects and wellcases (load, save, etc)

Edit – Preferences, detailed editing of well trajectory and laterals

View – Wellbore (mainbore or lateral)

Units – Selecting the basic Unit set: SI, Metric or Field.

Help – User Manual and Technical Manual. Enter license key. Display version information.

Reservoir and Well View

- This view is displayed when the Reservoir & Well tab is selected in the bottom left corner. An example is given in **Figure 5**. The view visualizes well and reservoir data, and facilitate entry of well paths and well segmentation:

Well paths can be defined by three alternative methods: i) import of deviation survey data, ii) import of UTM coordinates or iii) interactively by clicking and dragging with the mouse in the graphical views or by entering values in the table on the left hand side. The well path is represented by a collection of *Trajectory Points*.

Well segmentation subdivides wells into segments. The segment boundaries are defined by *Nodes* within NETool. A segment contains equal reservoir and completion properties within the entire segment. The properties can be different from segment to segment.

The uppermost graphical panel on the screen shows a lateral, birds-eye view of the reservoir and the well path. If a reservoir grid is present the view will show reservoir simulation grid layers and their respective reservoir properties.

The middle graphical panel illustrates a cross section of the reservoir along the defined well trajectory. If a reservoir grid is present the view shows detailed property distributions for the cross section along the well.

The lowermost graphical panel shows completion schematic for the selected lateral.
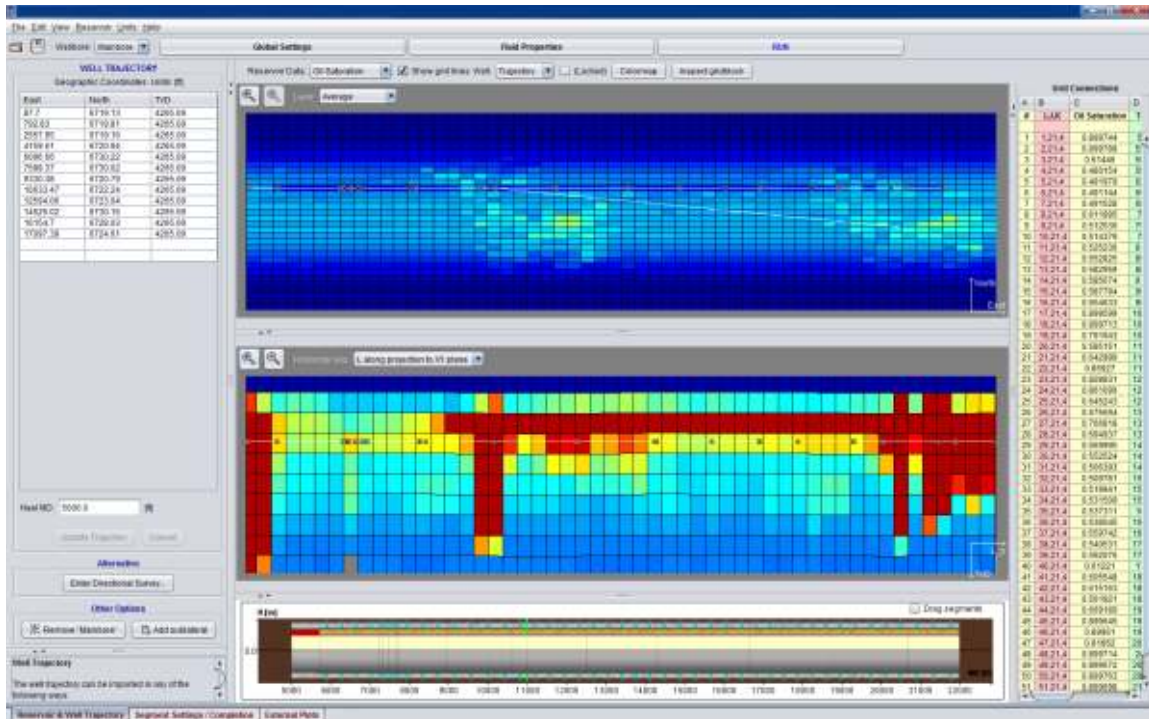
Figure 5. Example of Well View w/Reservoir Grid

Fluid Properties View

The Fluid Properties panel is used to define:

PVT data – Oil, Water, Gas. Different formats and PVT correlations available.

Relative Permeability data – Oil, Water, Gas. Different formats and RelPerm correlations available.

Lift Performance data – In Eclipse VFP table format. Gas lift is also allowed.
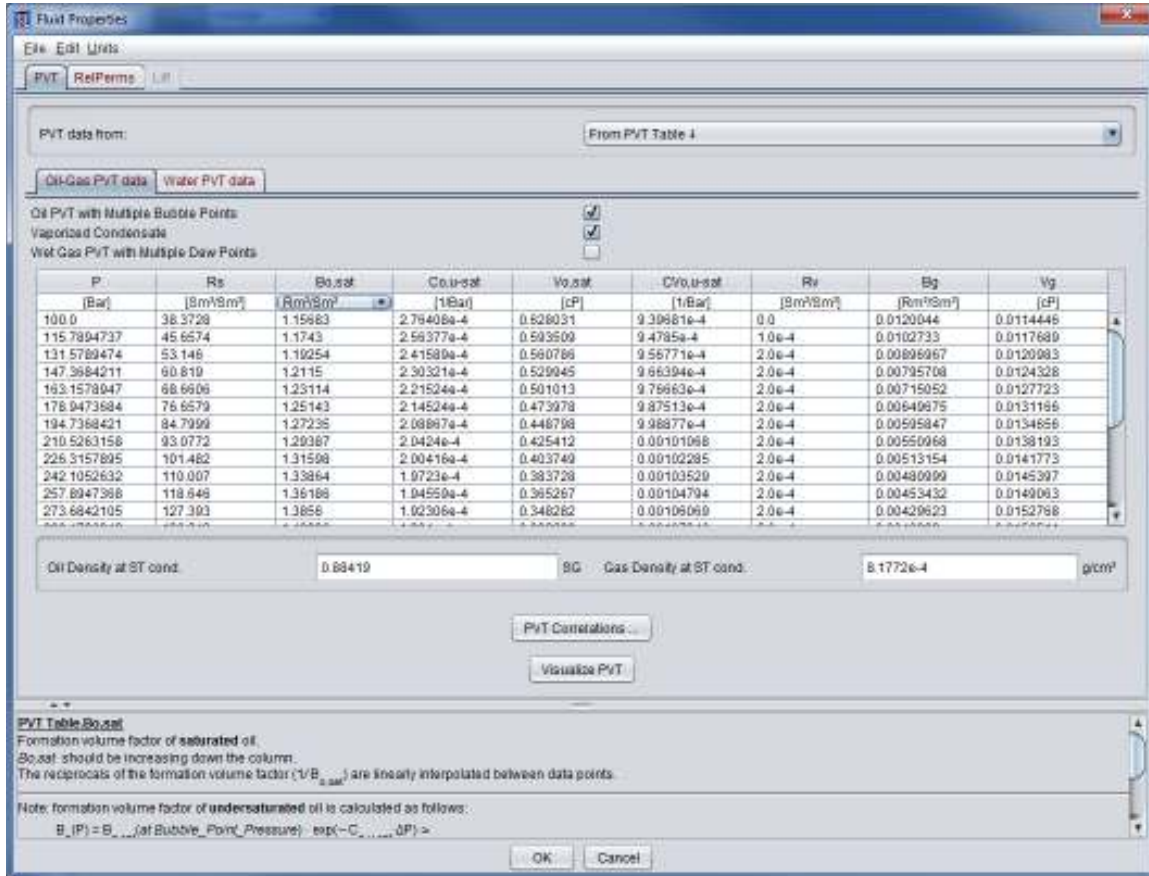
Figure 6. Example of Fluid Properties View

Global Settings View

The Global Settings panel is used to define:

General configurations – Well type, simulations targets, pipe flow correlations

Inflow – PI model specifications

Advanced - convergence configurations

Output – Specify results parameters to be reported and an option for enter PLT data

Annotation – Provide detailed description of well case

Figure 7. Example of Global Settings Window

Segment Settings/Completion View

This panel is used to define i) reservoir, ii) completion and iii) well parameters along the wellbore.

These parameters can be fully varied to honor reservoir heterogeneities along the well and to build complex completions.

Examples of some parameters:

Borehole size

Completion parts and their configurations

Reservoir properties (manually entered, imported from logs or taken from grids)

Skin information

Permeability in collapsed annulus
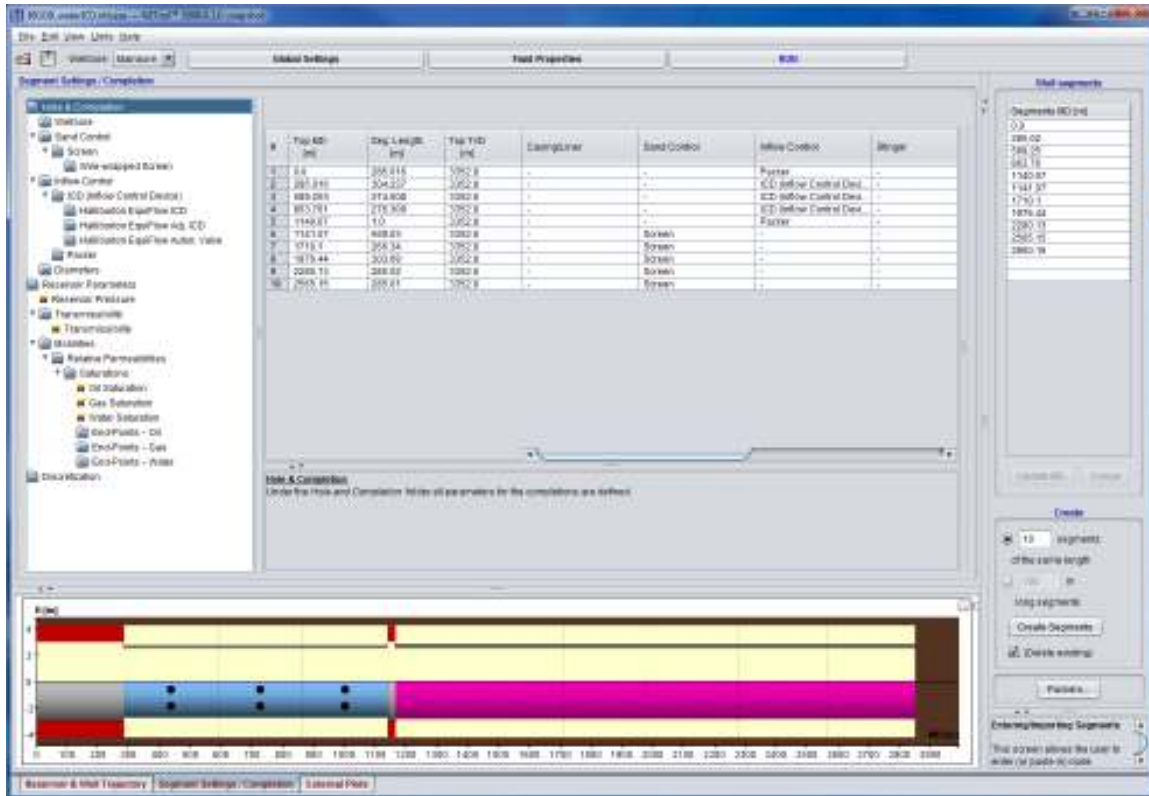
Figure 8. Example Segment Settings/Completion View

External Plots View

This panel allows for entering data that can be plotted together with NETool results. One example is to enter rate information from PLT logs that can be used for history-matching the NETool model. PLT data can also be entered in **Global settings→Output**.
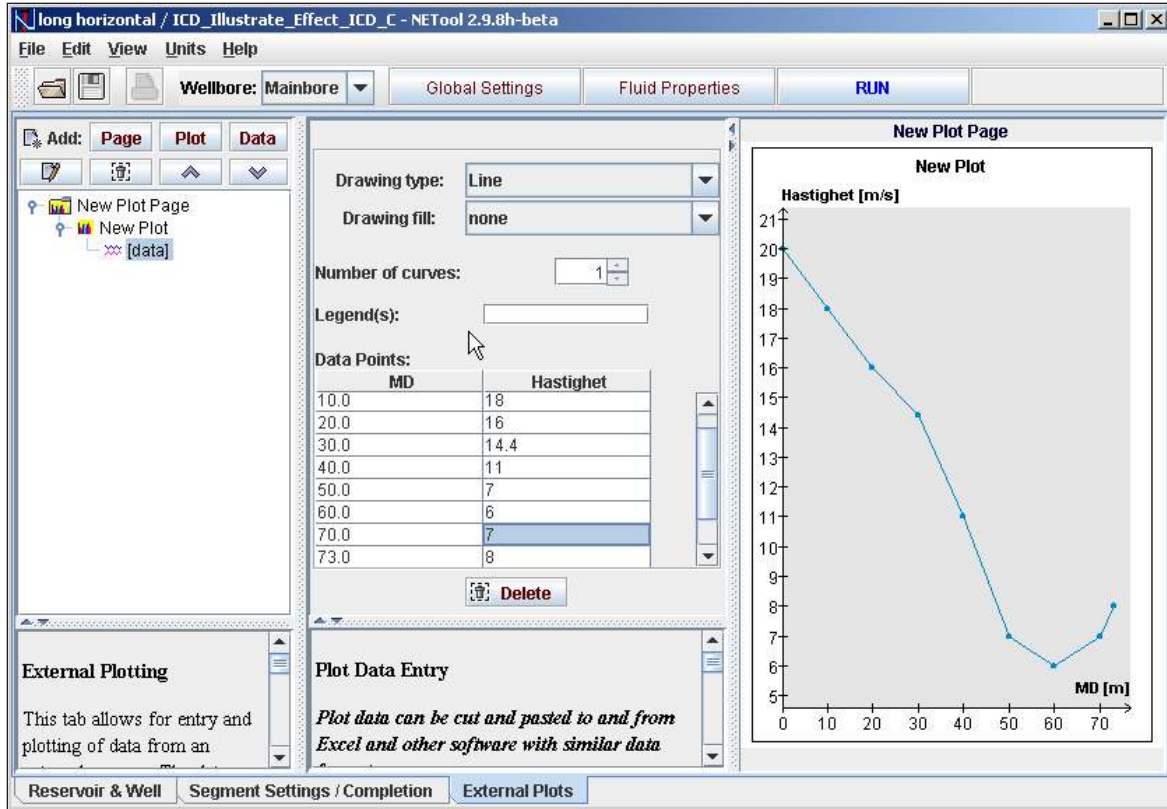
Figure 9. Example of External Plots View

Results Viewer

Once all required data for a simulation is populated, the user submits the job and the results are displayed directly on the screen as plots and tables.

The Results Viewer shows calculated flow rates, pressure drops, phase fractions, fluid velocities, densities, viscosities, fluid fluxes, permeabilities and other parameters.

The Results Viewer has capabilities for making customized plots; Legend text can be edited, compare plots combining results from different simulations can be made and it is possible to plot external data together with simulated data.

Detailed graphs of PVT and Relative Permeability data are also displayed.

The Results Viewer facilitates copy-paste functionality of the plots and tables into applications like Word and Powerpoint.

It is also possible to save results in the Plots Viewer in a separate file for later use or for exchanging results with other NETool users.
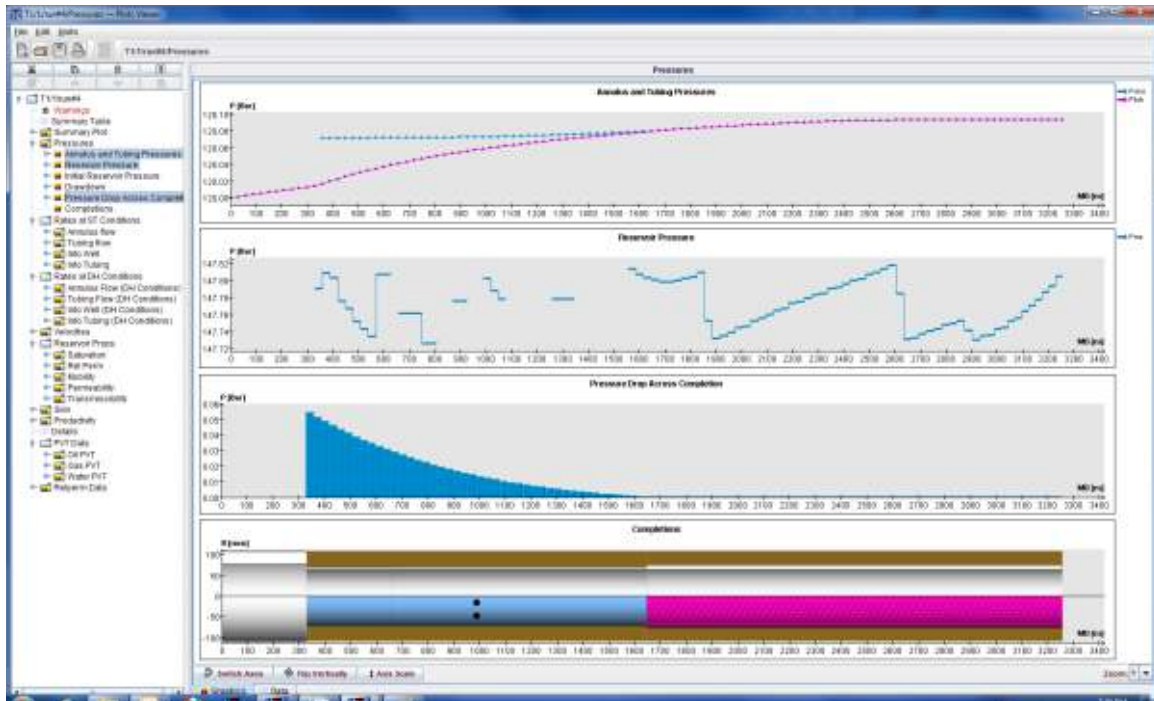
Figure 10. Example of Results Viewer

Work Processes

Work processes are as usual important for efficient usage of a software application. NETool has the Wizard capability that guides new users through the initial steps when populating NETool with data.

Figure 11 is an illustration of structured NETool work processes that may be helpful.
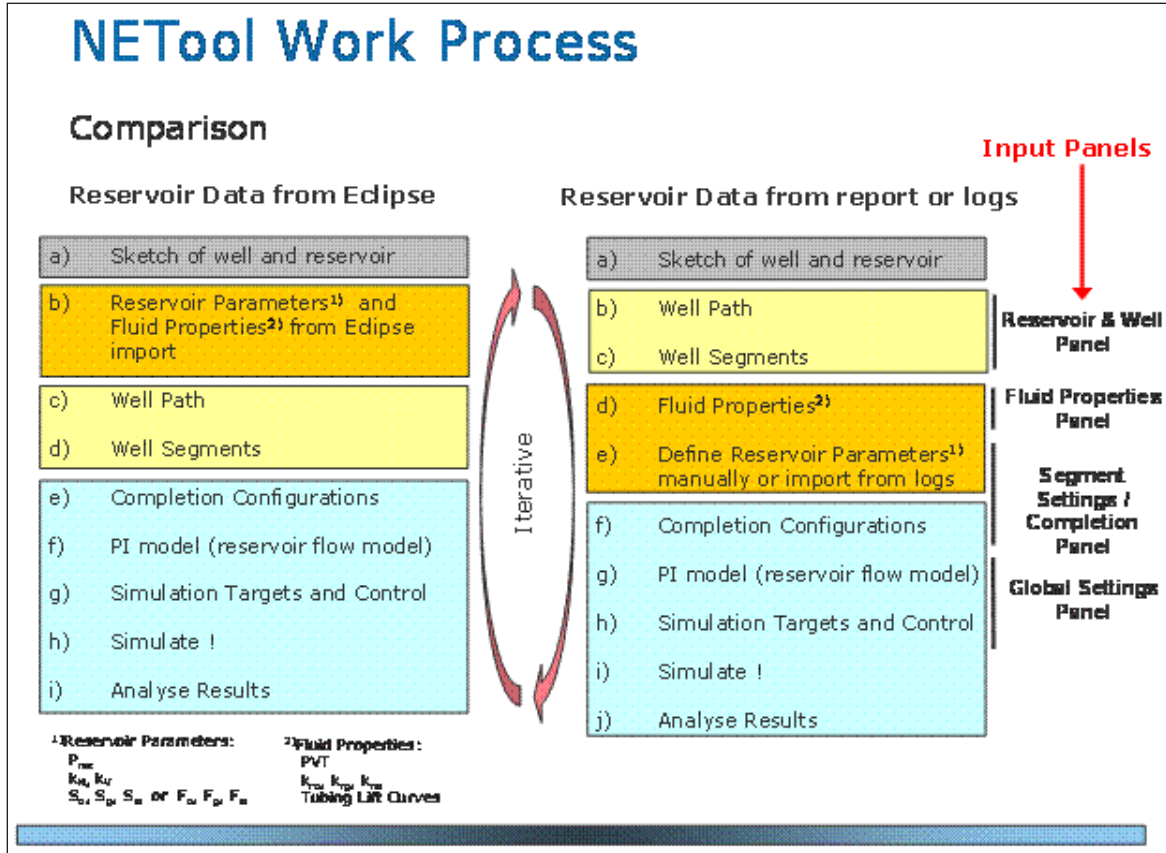
Figure 11. Example of NETool Work Process

The Mathematical Model

Mathematically, NETool is based on conservation of mass and momentum of oil, gas and water components.

The general momentum balance equations are replaced by correlations expressing pressure drop as functions of total volumetric flow rate, phase fractions, flow geometry and fluid properties.

Permeability, Productivity Index & Skin

There are different correlations associated with different parts of the numerical network.

Starting at the upstream end, inside the reservoir, the reservoir pressure drop correlation is based on Darcy's law for flow in permeable media. Permeabilities, PI models and Skins are used to define the pressure drop in the reservoir.

Permeability

A reservoir simulation model contains scattered permeability data, and NETool is able to honor those permeability variations in the reservoir by applying upscaling. This upscaling process is performed in NETool under the assumption of pure radial flow.

The permeability values are essential in PI models unless transmissibilities are supplied directly.

PI Model

The user can choose between PI models for i) steady state or ii) semi-steady state flow conditions.

Vertical wells use standard radial Darcy flow equations.

Horizontal wells use Joshi for steady state flow, and Babu and Odeh for semi-steady state flow.

Deviated wells use a transformation of both the vertical and horizontal formulations.

The PI models are used to represent the reservoir performance, by utilizing local PIs according to the permeability variations along the well trajectory.

There is an option to define the total PI for the well. If enabled, the local PIs are calculated and scaled proportionally to the local reservoir properties to fit the total well PI.

Please be aware it is important to supply a consistent set of reservoir pressures and reservoir boundaries in order to get the PI modeling right.

Skin

Local skin values may also be included in the PI model. Currently, the following skin models are provided:

Damage zone skin

Non-Darcy skin based on Forchheimer coefficient

Additional skin

Flow convergence skin (i.e. skin caused by flow convergence around perforations)

Perforation skin as functions of perforation depth, phasing angle, crushed zone radius, perforation density and gun data

Two different perforations skin models are offered: i) Karakas and Tariq and ii) Furui, Zhu and Hill. The Karakas and Tariq perforation skin model was originally created for vertical wells. In order to make them applicable to horizontal wells, a transform is applied to make an anisotropic medium appear as isotropic.

Pressure & Saturation

Reservoir pressures are required for each segment and must always be supplied.

Saturations are also normally supplied. Saturations are used to define relative permeability values by look-up in Relative Permeability curves and correlations.

The fractional flow feature can be used instead of a set of saturations and relative permeability curves. This feature is a simplified and easy way to define phase fractions and can be used for quick-look purposes.

If a reservoir simulation model has been imported with saturation and pressure data, then saturations and pressures can be retrieved from the grid block(s) the well is passing through. Or alternatively these values may be entered manually or populated from a log.

Completion types

The pressure drop correlations to be employed for the different parts of the completion are interactively selected in the menu system. All the completion types can be fully mixed and varied along the well.

Separate correlations are provided for the following completion types:

Inflow Control Valve (ICV) Device

Slotted liner

Open hole

Perforated cemented liner

ICD – Nozzle ICD

ICD – Tubular ICD

ICD – Channel ICD

Halliburton EquiFlow ICD

Halliburton EquiFlow adjustable ICD

Baker spiral ICD, Troll field

Baker spiral ICD, General

Baker adjustable ICD

Autonomous ICD

Screens

Gravel Pack (as Screen + Gravel in Annulus)

Cased Hole Gravel Pack (as Screen + Gravel in Annulus + PCL)

Blank pipe

Cemented blank pipe

Packer

Tubing Plug/Choke

Completion layers

Well in Netool can have up to four different completion layers:

- Casing/Liner
- Sand Control
- Inflow Control
- Stinger

Each of these layers has certain completion types available depending on the layer's designation. Stinger is the most inner layer, while Casing/liner is the most outer one.

If a segment has more than one layer their diameters should be in the increasing order from Stinger to Casing/Liner. Layers can touch each other making zero annulus clearance but they can't overlap.

However, layers of neighboring segments can overlap. F.ex. Stinger at segment #1 can have larger ID and OD than Sand Control at segment #2.
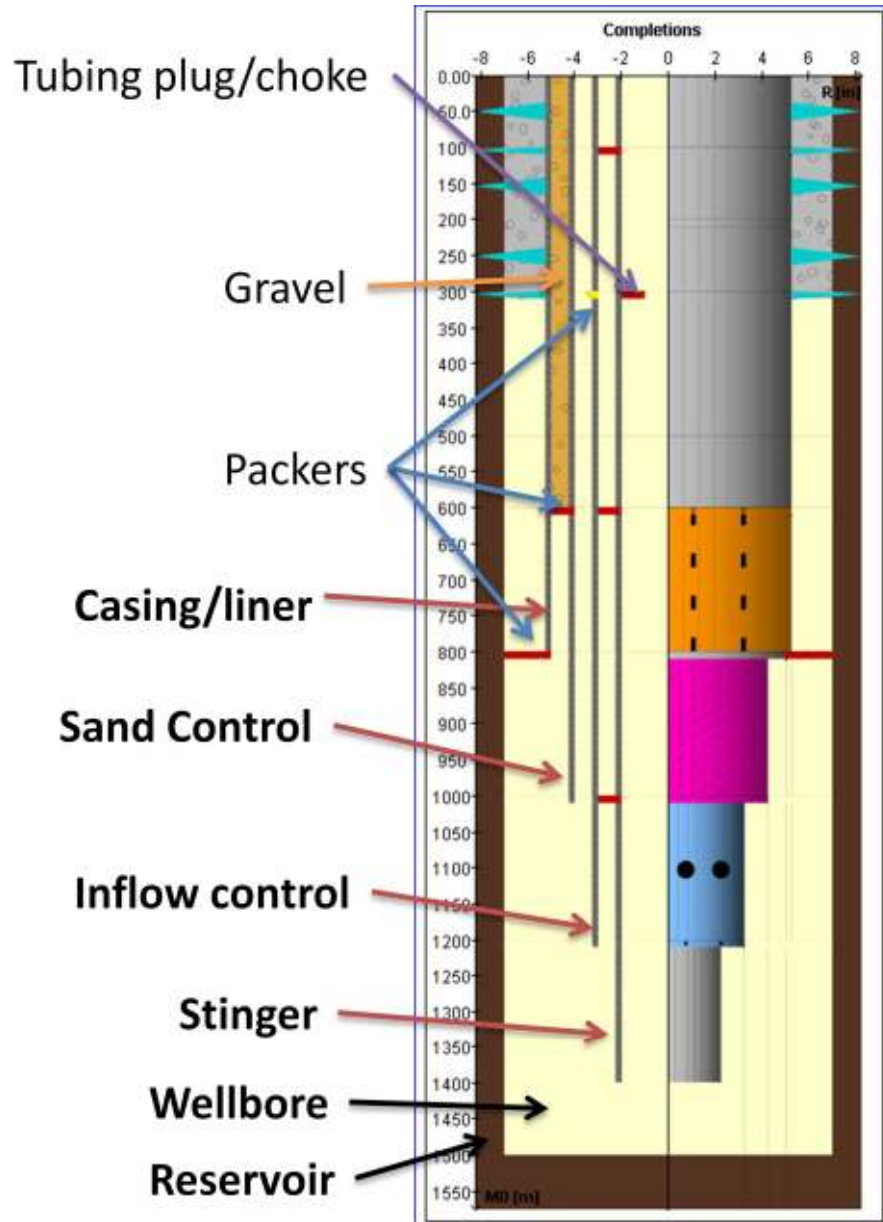
Figure 12. Four annulus layers supported by Netool.

Boundary Conditions

A NETool simulation requires the following boundary conditions:

Outer boundaries:

Reservoir pressure at each reservoir node

Inner boundary - one of the following simulation targets:

Flowing bottom hole pressure

Tubing head pressure

Total volumetric reservoir rate (at reservoir conditions)

Total liquid rate (at standard conditions)

Oil rate (at standard conditions)

Gas rate (at standard conditions)

Water rate (at standard conditions)

Download and Installation

Program Download

NETool is available for download from the Landmark Software Manager (LSM), our Electronic Software Delivery (ESD) tool (http://esd.halliburton.com). Download of releases from the LSM Releases tab is restricted to individuals who are authorized to receive the software. Not all users can download from this area. Additional information about the LSM is located at:

http://css.lgc.com/CustomerSupport/R5000LSMStart.jsp

The LSM offers several advantages:

Notification of new releases or patches that have become available.

Immediate access to new releases or patches.

Ability to review and download Release Notes and Installation Guides prior to download.

History of software products and versions that have been downloaded.

A single location for a holistic view of your Landmark application environment.

Ability to request DVDs where bandwidth issues prevent download.

If you have been given access to receive and deploy NETool, you will need to register your account in the Customer Support Portal, in order to access the LSM. The top of this email contains your PIN and the email address needed to register your account.

Go to the Customer Support Portal at:

http://css.lgc.com/CustomerSupport/CustomerSupportHome.jsp

Under the Please Sign In area, click the Register your PIN here link.

Enter your PIN and email address as provided above.

Follow the remaining instructions to create your password.

Click **Download** for your desired LSM platform.

License control

NETool 5000.0.1.0 requires license codes to be issued by Landmark Graphics Corporation. Licenses issued for version 5000 will work. However, all licenses for 3.x and earlier versions will not.

Compatibility with Earlier Versions

All files created with NETool 3.0 and later versions can be opened in NETool 5000.

The opposite—opening files created with NETool 5000 with earlier versions of NETool — may or may not work depending on the file.

NETool 5000 and NETool 3.xx install in different folders, use different license files, and can be run independently of each other. However the two versions share the files database, please understand the files compatibility if using both versions in parallel.

Getting Started With NETool

Wizard - Define Well Case

Proper work processes are important to utilize NETool's capabilities efficiently.

The easiest way for a beginner is to create the first well case by using the Wizard capabilities. The wizard starts from **File→Open From Wizard**.

It guides users stepwise through data import and definition processes up to the point where detailed completion and reservoir details need to be supplied.

After using the wizard several times the user usually gets acquainted with the NETool work process and menus, and will prefer to create and modify well cases without a wizard.
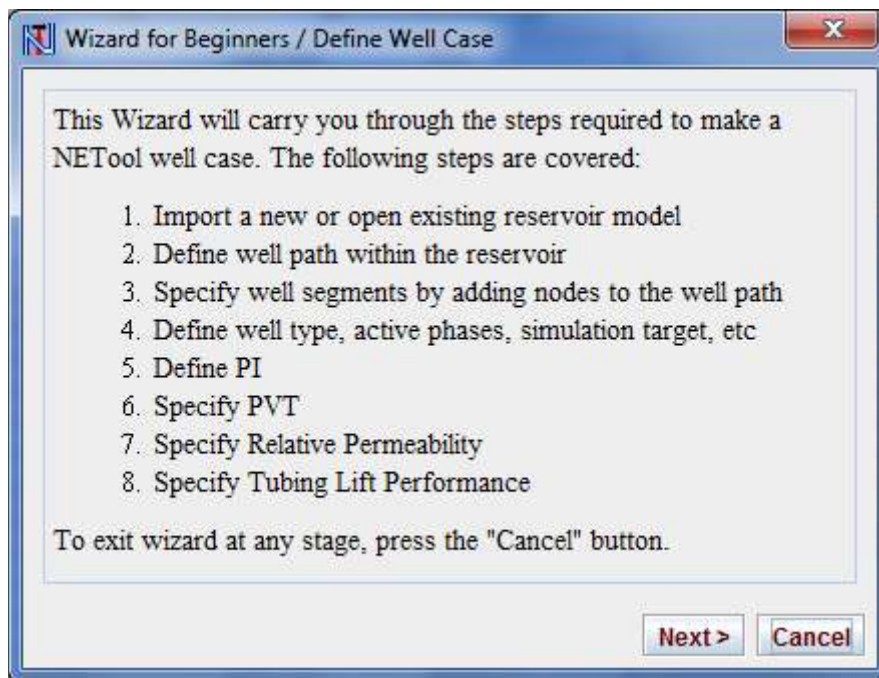
Figure 13. Input Wizard First Screen

Manage Reservoirs, Projects and Well Cases

- This section describes the organization of NETool data. The data is stored in a file repository which constitutes the NETool's database.

- NETool simulations are grouped into Projects. One project is typically for the same reservoir and the top grouping level is therefore named **Reservoir/Project**.

- NETool simulations are called **Well cases**. A **Reservoir/Project** group usually contains many wellcases.

- The operations to organize NETool data are gathered in the **File** and **Edit** dropdown menus which are found on the top menu bar.

Data File Repository

NETool maintains a file repository where data is saved and loaded. The user should <u>not</u> manually operate the repository outside NETool User Interface.

Repository Location

The default location for the repository depends on the operating system:

Windows XP®, Windows 2003®	C:\Documents and Settings\<<username>>\Application Data\NETool\
Windows Vista®, Windows 2008®, Windows 7®	C:\Users\<<username>>\AppData\Roaming\NETool\
Linux®	/home/<<username>>/.netool/

The default location can be altered by user configuration as described in the following sections. When NETool starts, the information about the repository location is printed out into NETool console window.

Change Repository Location

In some cases, it is more convenient to define the NETool repository at another location than the default. The repository location can be configured interactively or defined in a configuration file as described in the next sections.

A network drive can for example be convenient when:

- several personnel need to access the same NETool repository from different computers
- automated company file backup is required
- you want different repositories for different projects

It can also be convenient to operate with several repositories a user wants to alternate between.

Changing the repository location does not relocate NETool repository files by itself. If it is desired to transfer the content of the previous repository location to the new location then the files and folders must be moved manually. It is recommended that IT personnel perform repository move or copy operations in order to minimize risk of losing files or corrupting the repository.

Repository location can be changed by the following actions:

Modifying the configuration file as described below. If the NETool installation is shared between several users this would define new default repository location(s) for all users.

A user can interactively change the repository location in the user interface. Go to **Edit→Preferences→Folder for User Files** menu and select/enter the new database location. Entering a blank value will restore the default.

Repository Content

The subdirectories in the repository make an organized hierarchy of user files.

When NETool starts, it reports into the console window the location of the file repository:

Figure 14. Repository location.

Note: Under Windows XP and Vista some directories are hidden by default. In order to view it may be necessary to allow viewing hidden directories/files under Windows. The hidden directories/files may be viewed by right clicking on the relevant directory, choosing "properties" and unchecking the "Hidden" box.

Under the main directory of the file repository, a subfolder **reservoirs** is created. This directory contains various reservoirs in separate subfolders of **reservoirs**. Each reservoir directory includes the following subfolders:

grid – a copy of the original files used when the reservoir was imported (e.g. .INIT, .EGRID, etc.)

netool_grid_cache – grid and reservoir data in NETool format with small file size for faster loading

wellcases – the case files for each well

Terminology: Reservoirs/Projects and Well Cases

Reservoirs/Projects

- NETool simulations are grouped into Projects. One project is typically for the same reservoir and the top grouping level is therefore named *Reservoir/Project*.

- Three main approaches to establish a Reservoir/Project:

Create Gridless Project. Does not require a reservoir simulation model. Data is imported from Spreadsheets, Logs and/or Manual Entry.

Create Project based on Imported Reservoir Model. A collection of reservoir model files containing grid block dimensions, permeability, porosity, net-to-gross, PVT data and other reservoir properties can be imported.

Combinations of the above. Override or supplement data in the Reservoir Model.

NETool supports reservoir grid files in formats created by the Eclipse reservoir simulator. Once a collection of Eclipse files is imported into NETool (see below) a *Reservoir/Project* has been created.

Well Cases

Within each Reservoir/Project multiple Well Cases can be created. Each Well Case contains the following information:

well path

well segmentation

PVT/RelPerm/Lift tables and other fluid properties data (see **Fluid Properties View**)

properties of each segment in the well path (see **Segment Settings/Completion View**)

general model parameters (see **Global Settings View**)

other data like the name of the reservoir, preferred unit system, etc.

In other words, a well case is a collection of all the parameters that a user can modify within NETool and describes the full work done by the user.

After a reservoir is imported and a well case is created, a NETool simulation of the well case can be performed.

Manage Reservoirs/Projects and Well Cases

Reservoirs/Projects and well cases can be managed by launching **File➔Open** to open the **Open Reservoir and Well Case** window: The same window is also always opened automatically when NETool is launched.

The window consists of three sections:

Select Reservoir or Project section – shows a list of reservoirs and gridless projects currently in the repository. The Add Reservoir button is used to add a new gridless project or a new reservoir.

Select Restart File section – offers the user to import associated Eclipse restart file(s) if an Eclipse simulation model already has been imported.

Well case section – This section offers the user three main alternatives:

Start a new well case

Open an existing well case

Load a well case from another location than the current NETool repository
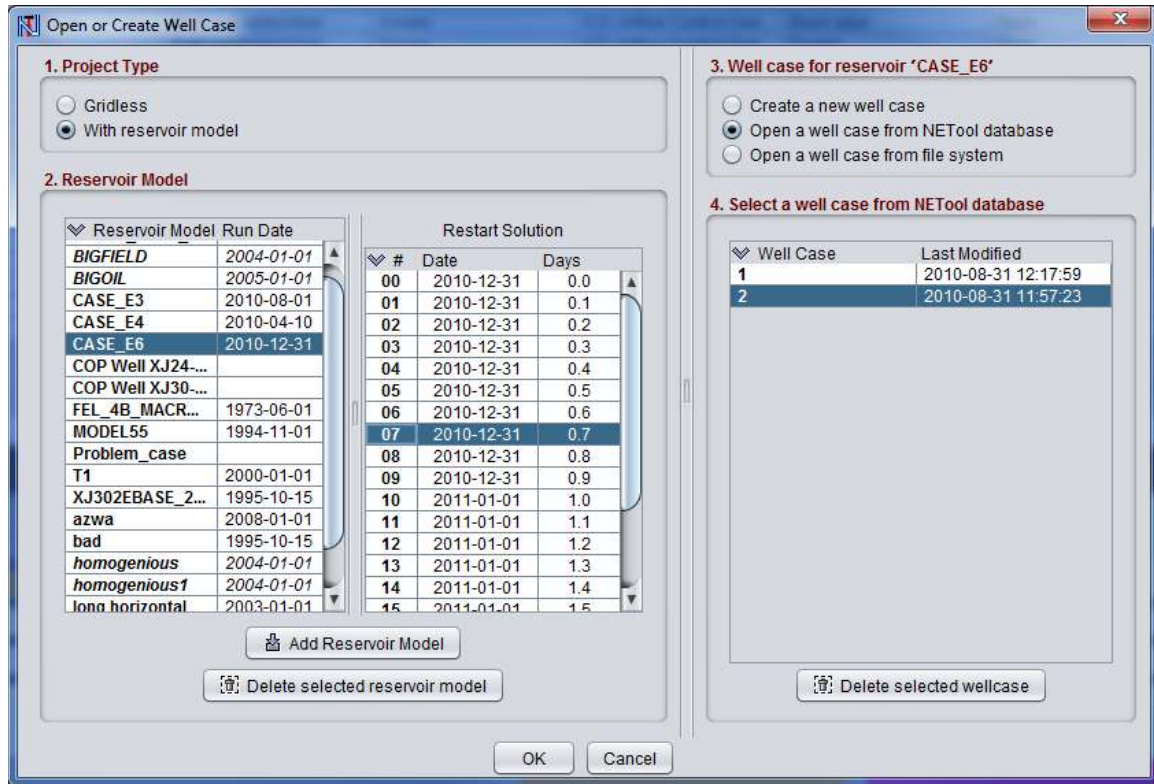
Figure 15. Example of Open Reservoir and Well Case Window

NETool is shipped with some predefined sample reservoirs (BIGOIL, BIGFIELD and Homogeneous) that will be listed also when NETool starts for the first time.

These predefined reservoirs can be used if one for reason does not want to create a Gridless Project. In that case remember to override the reservoir parameters and fluid properties within the model by populating the NETool model with log data, manual input and/or utilize NETool's built-in PVT and Relative Permeability correlations.

Creating a Gridless Project

A Gridless project can be created in the Open Reservoir and Well Case window and by choosing **Gridless project**.

For a gridless project all reservoir properties must be supplied by the user. The data can be specified manually, copy-pasted from spreadsheet or other applications, or imported from logs, or alternatively created by NETool's built-in correlations.

The built-in correlations can generate PVT and Relative Permeability data based on limited information.

Importing Eclipse Reservoir

A Reservoir/Project based on a reservoir simulation model can be created in the Open Reservoir and Well Case window by choosing **Add reservoir →Import reservoir model in Eclipse format**.

The **Import Grid Wizard** is launched and the user is requested to submit reservoir simulation model files sequentially.

Please note that when importing an **Eclipse .INIT** file, a corresponding Eclipse binary grid file is required. The Eclipse binary grid file can be in the extensible grid file format (**EGRID**) or in plain Eclipse grid file format (GRID). If both **.EGRID** and **.GRID** files are supplied, the **.EGRID** file will be used.

However, some Eclipse reservoir simulation features are currently not supported in NETool, e.g. the present version of NETool **cannot** import:

Eclipse grids without inactive cell data. This applies to **.GRID** files only, and the problem is avoided by specifying the record
```
GRIDFILE
2 /
```
in the **Eclipse .DATA** file.

If Eclipse restart files (**.UNRST** and **.Xnnnn** extensions) are present in the folder with the **.INIT** file, they are highlighted and the option to import restart files is presented to the user. Restart files contain e.g. pressure and saturations for each grid block for specific points in time.

Please refer to Eclipse documentation for more details on Eclipse file formats.

After a reservoir model is fully imported, it will appear in the list of Reservoirs/Projects, and it is saved as a NETool internal binary format. The internal binary format is used to reduce file size and load speed compared to the voluminous Eclipse files.

Exporting Reservoir

If you would like to share a NETool reservoir with other users, choose ⬆**Export NETool binary grid** from the File menu. This will export your reservoir grid with properties and save it as a file (with suffix **.ntlgrid**) that you can submit e.g. by email.

Since exported NETool grids are binary files, it is not useful to open these files with a text editor.

These NETool binary grid files can be imported into NETool with **File → Open → Add reservoir model.**

Creating, Opening and importing Well Cases

Select the correct Reservoir/Project in the list of Reservoirs/Projects. Three options will then appear at the right hand side of the dialogue box**:**

Start a new case. This will create a new well case with empty well path, default values in **Global Settings**. In most cases, these default values should be treated as <u>not</u> being reasonably describing your own specific data. There are no universal defaults good for all situations!

Open the **Reservoir & Well** screen to define the well path and subdivide the well into segments.

The next step is to populate NETool with reservoir properties and completion configurations by using the **Segment Settings / Completion** screen.

If available in the Eclipse file dataset, PVT and relative permeability tables will be imported if Eclipse files are imported. If not imported from the Eclipse file dataset, some default tables will appear. However, your real PVT and relative permeability data can be generated from other sources and copied into NETool through **Fluid Properties**.

The PI model and simulation control parameters are defined in the **Global Settings** screen.

To save the new well case choose **File ➔ 💾 Save** or a corresponding button (💾) in the toolbar. Once the well case is saved you will be able to see it in the list of well cases in the **File Open** dialog box.

Open saved well case. Open a saved well case.

Import a well case. In order to facilitate exchanging of data between users there is an option in NETool save a well case into a user-specified folder instead of Netool database. These files have a suffix `.ntlcase`. If you have a well case exported from NETool as described above, import it into your NETool setup with the **File ➔ Open**. Please note that it makes sense to import a well case only if it has been created for the same reservoir that appear on the left hand side part of the File Open dialog box.

Note: NETool will try to enforce this by comparing the names of reservoirs. For example if a user Joe has a reservoir "JoesField", then he creates a well case and sends it by email to Mary. If Mary has a reservoir "MarysField", selects it and tries to import the wellcase NETool will show a confirmation dialog "The reservoir name stored in well case is "JoesField " which is different from " MarysField". Are you sure to proceed? " If MarysField is the same reservoir as JoesField, just named differently, Mary should choose **OK**, otherwise **Cancel**.

Deleting Reservoirs, Projects or Well Cases

Delete Reservoir/Project

To delete a Reservoir/Project, select it from the list and click the **Delete Selected Reservoir** button. If the Reservoir/Project was created by the user it will be deleted completely and will disappear from the list of available reservoirs.

If you are trying to delete a sample reservoir that was shipped with NETool or a reservoir created by your system administrator, then only well cases and custom PVT/RPT tables will be deleted. The "grid files" will remain and the reservoir will still be listed as an available reservoir.

Delete Well Case

To delete a well case, select the well case and click **Delete Selected Well Case**.

Saving Well Cases

To include the current well case to the NETool repository use **File→Save Well Case As or File→Save Well Case**.

It is important to conduct the save operation since a well case is not part, or updated, in the NETool repository until the save operation is performed.

Exporting Well Cases

A well case can be shared with other users by saving it into a user specified folder. This will export your current well case and save it as a file (with suffix **.ntlcase**) into your chosen folder location.

Please be aware this features do not save the well case to the current NETool repository.

Running NETool in Simplified (Training) mode.

It is not easy for a beginner to get acquainted with NETool because of the volume of different input and output parameters. To aid with start-up a simplified mode has been developed. It can be launched through a shortcut in windows "Start" menu (or by adding "-s" key to run_netool.exe.).

In this mode most of the regular input and output parameters are hidden from the user. Although it creates lots of functionality restrictions it helps beginner to understand fast the basics of the software. This mode is especially helpful for training classes.
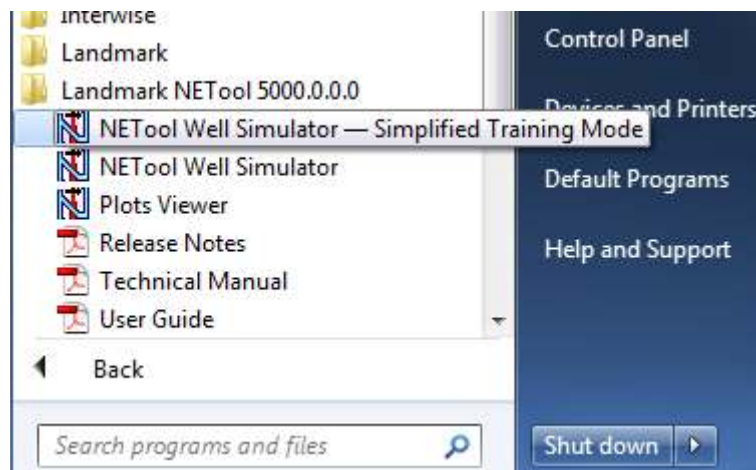
Figure 16. Launching NETool in Training mode.
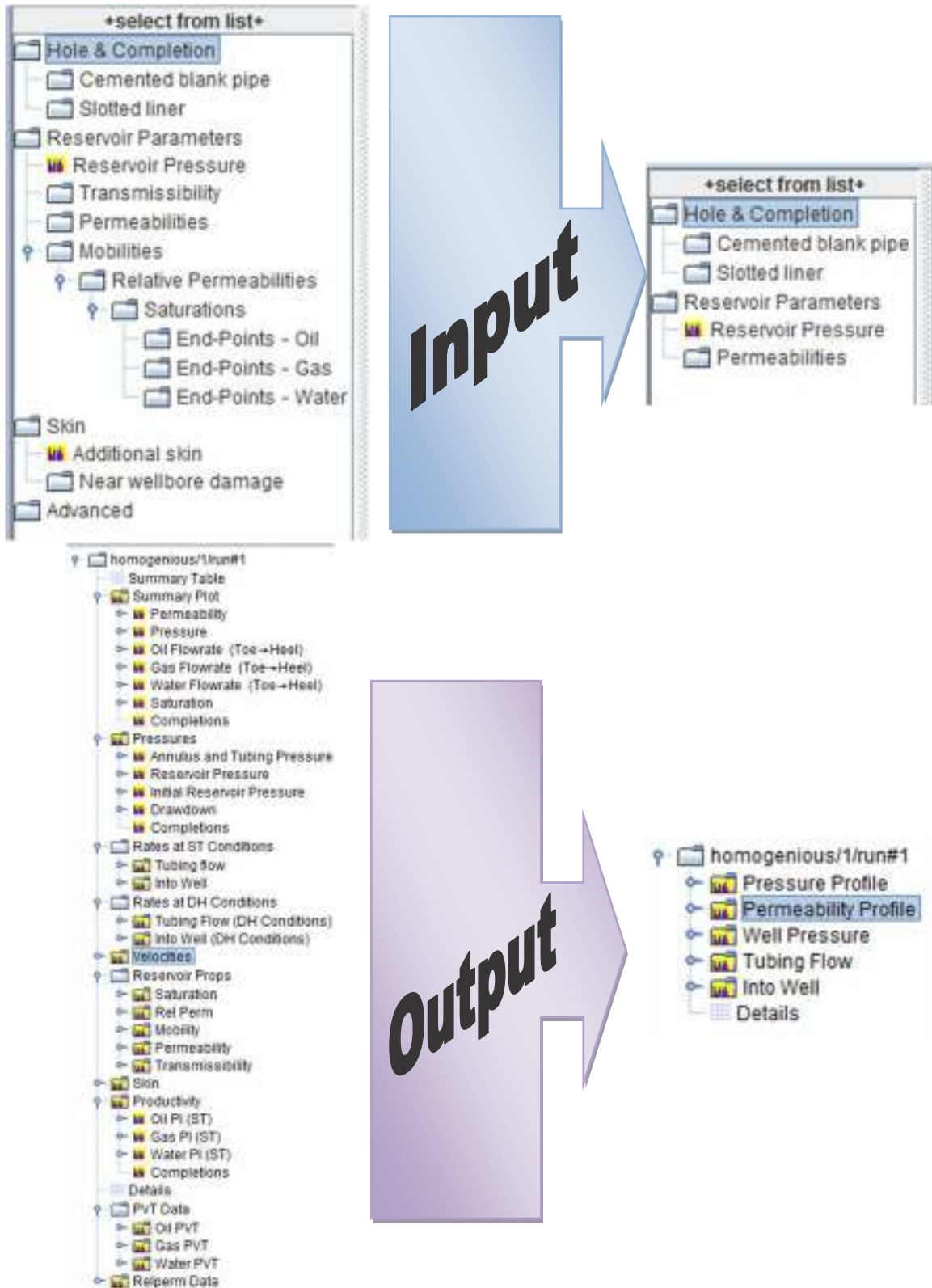
Figure 17. Simplification of input and output in "Training" mode.

Reservoir & Well Menu

- This section describes the features within the **Reservoir & Well** Menu. The menu is displayed when the **Reservoir & Well** tab is selected in the bottom left corner. An example is given in **Figure 5**.

Introduction

- The menu visualizes well and reservoir data, and facilitate entry of well paths and well segmentation:

Well paths can be defined by three alternative methods: i) import of deviation survey data, ii) import of UTM coordinates or iii) interactively by clicking and dragging with the mouse in the graphical views. The well path is represented by a collection of **Trajectory Points**.

Well segmentation subdivides wells into segments. The segment boundaries are defined by **Nodes** within NETool. A segment contains constant reservoir and completion properties within the entire segment. Properties can be different from segment to segment.

The menu can be operated in two main modes. This is controlled by the **Show** dropdown box:

Trajectory must be selected to work with well path definitions

Segments must be selected to work with well segmentation

The menu display has four main areas:

The section on the left contain details of well paths (trajectory points). This is the main area for entry and maintaining well paths.

The uppermost graphical panel shows a lateral, birds-eye view of the reservoir and the well path. If a reservoir grid is present the view will show reservoir simulation grid layers and their respective reservoir properties. The view can also be used to interactively move, add and delete well trajectory points and nodes horizontally using the mouse.

The middle graphical panel illustrates a cross section of the reservoir along the defined well trajectory. If a reservoir grid is present the view shows detailed property distributions for the cross section along the well. The view can also be used to interactively move well trajectories vertically using the mouse. Nodes can be altered similarly along the well path.

The lowermost graphical panel displays completion types used in the well. If no completions are defined yet (no segments) the panel is empty.

The section on the right is present if a reservoir grid is loaded. It lists the reservoir grid blocks intersected by the well. The section is not available in a **Gridless Project**.

Graphical Panels

The two graphical views can display the well path, the well segmentation and the reservoir grid with properties. If the **Well Case** is for a **Gridless Project** only the well path and well segmentation will be displayed.

The upper graphical panel shows a horizontal, birds-eye view of the well and reservoir. The lower panel shows similarly the vertical cross-section along the well path.
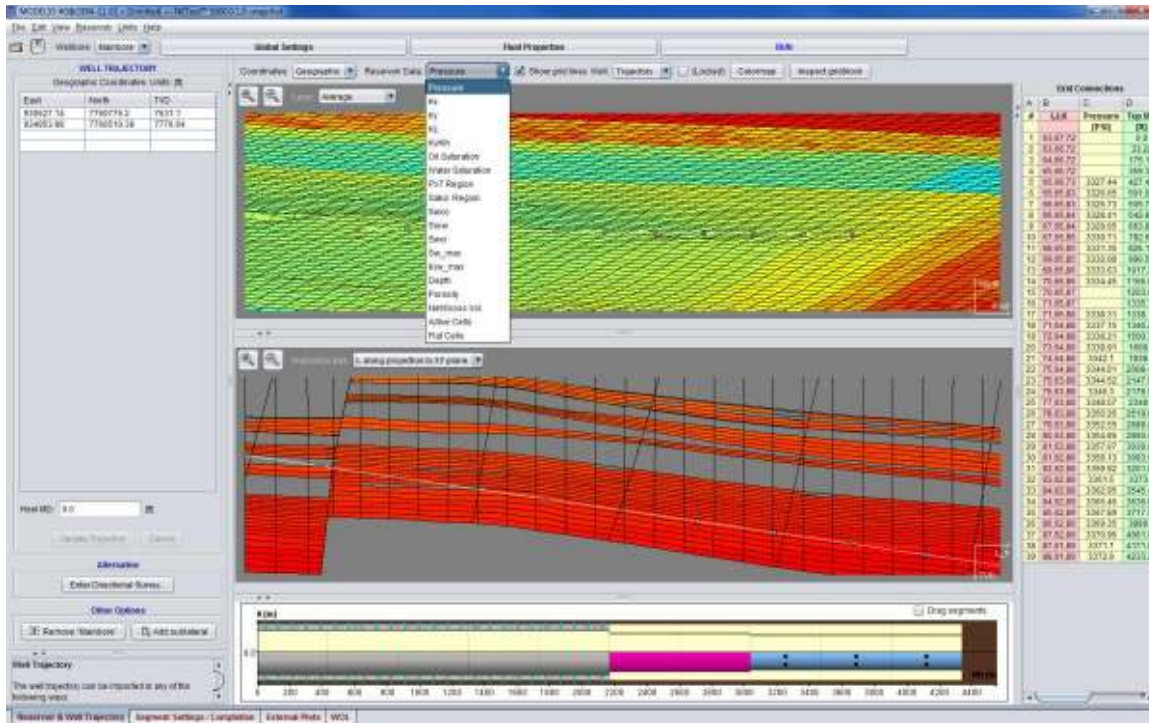
Figure 18. Example of Graphical Panel

A reservoir property such as permeability, porosity, saturations, pressure, active cell and so forth (depending on the content of the imported reservoir simulation model) can be displayed in the graphical panels by colored reservoir grid.

The available properties for display can be selected in the **Reservoir Data** dropdown menu.

The reservoir properties can be presented as the average along the Z-axis or for each individual layer by using the *Layer* drop down menu.

Zooming is possible for both graphical views. The zooming actions act independently for each view. Use the **right** mouse button to define a rectangular area and then click ⚲ **Zoom to selection**. The view will be updated so that the selected area will occupy the whole screen. To view the whole reservoir, click ⚲**Zoom to Screen**.

Inspecting grid.

Due to MAPAXES keyword in Eclipse files grid coordinates exported from Eclipse or other software might not match the real UTM (or Geographic) coordinates well trajectory is in. For example, in GRID coordinates the origin can be at the beginning of the reservoir and all reservoir blocks will have 2-3 digit coordinates, while in Geographic coordinates the origin is often hundreds kilometers away causing grid blocks to have 5-6 digit X, Y, Z.

To switch between short (GRID) and long (Geographic) formats use "Coordinates" switch on top of the reservoir panel.

"Inspect gridblock" button allows to further investigate coordinates and properties of any grid block. "Top" section of the coordinates corresponds to the upper four points in Corner Point Geometry: I, I+1, J, J+1. The "Bottom" contains the same corners except at K+1 instead of K.
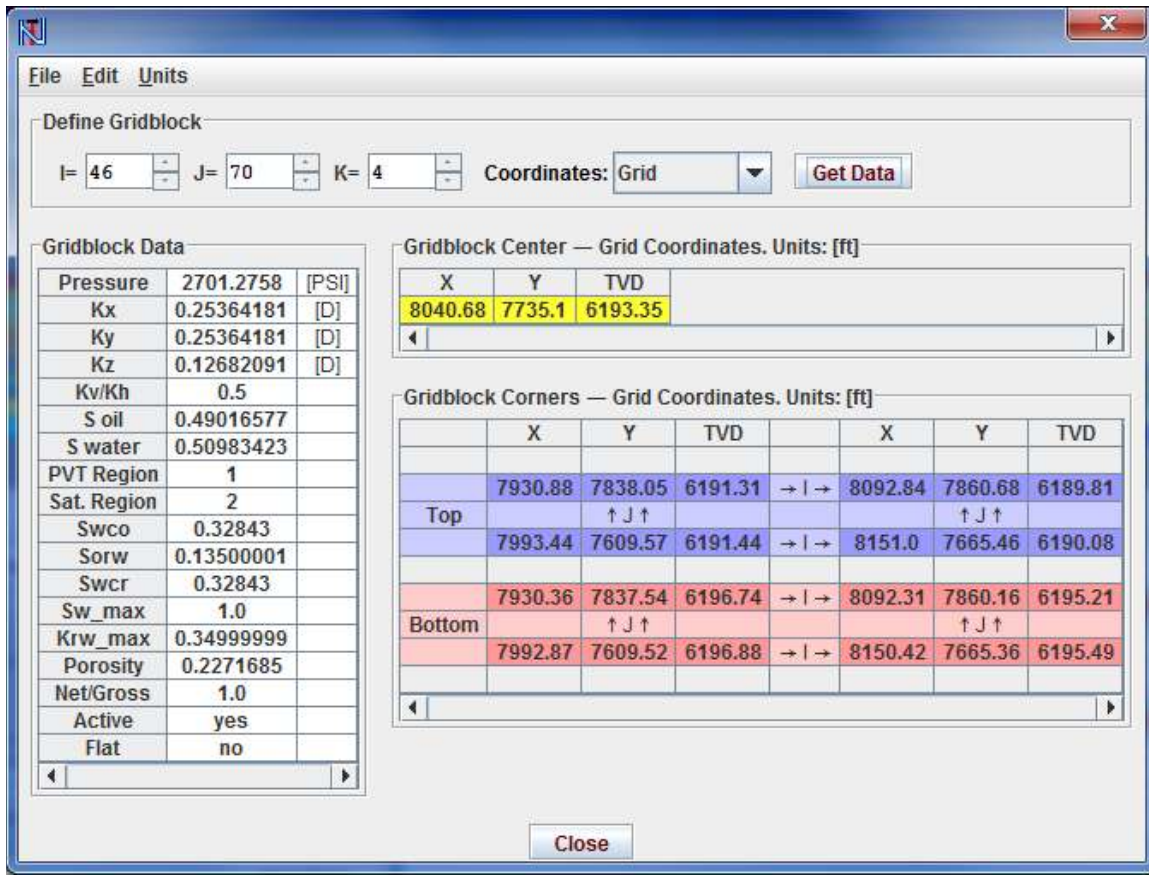
Figure 19. Inspecting grid blocks in detail.

Note: Netool does not have any editing capabilities for grid files. Grid is imported and stored as read-only. If user needs to use values different from the grid at some location (f.ex. permeability) he should switch that property to "Manual" instead of "From grid" in "Segment Settings->Reservoir Parameters" and enter desired value.

Working with Well Paths

Definition of the well path is performed in **Trajectory** mode. **Trajectory** mode is activated by the **Show** drop down box located on the top left in the **Reservoir & Well** screen. The well path cannot be modified in the **Nodes** mode (which is the other alternative in the **Show** dropdown box).

The well path constitute of **trajectory** points linked together. There are three ways to define a well path in NETool:

- Interactively with a mouse and/or keyboard
- Copy-pasting deviation survey

After the well path is defined consider to activate the **Locked** checkbox to make sure that the well path will not be altered accidentally.

Creating and Changing Well Path Interactively

Well paths are created by clicking within the grid in the top window to create trajectory points. Simply click (left mouse button) several points and the well is created. Trajectory points can also be moved by dragging the points - click and hold down left mouse button, move mouse and release mouse button when the trajectory point is in the appropriate location.

Moving the trajectory point in the upper window moves the trajectory point horizontally (x-y directions). To adjust the depth (z direction) of the trajectory point, drag the trajectory point vertically in the lower window.

Additional trajectory points can be added between existing trajectory points by clicking on the white line in between defining the well path.

Right clicking on a trajectory point will bring up a menu which allows the user to:

Remove Trajectory Point – remove that point from the well trajectory. NETool will then calculate a new trajectory between the remaining points, or shorten the well if this is the last trajectory point.

Start New Lateral – a new lateral will be started off of the current trajectory point.

Importing Well Trajectory

Two alternatives are available to import well paths:

Supply **UTM coordinates** -   equivalent to i) x,y,z or ii) easting, northing, TVD

Supply **Directional Survey data** - as MD, inclination, azimuth

The well path import is accessed on the left side of the **Reservoir & Well** panel or from the menu **Edit → Well Trajectory & Nodes**.

UTM Coordinates

Step 1 – Define and Edit Well Trajectory

The Well path can be defined by i) typing or ii) pasting the x,y,z coordinates of trajectory points into the **Trajectory** text field.

Values turn blue when they are edited.

When a reservoir grid is used it is important to use the same coordinate offset for the well path and the reservoir grid such that the trajectory will be at the correct location within the grid.

Step 2 – Update Trajectory

After the trajectory is entered or edited, push **Update Trajectory** button to update the trajectory information.

Step 3 – Define MD Reference Depth (Heel MD)

The measured depth for the first x,y,z coordinate point in the NETool model is defined in the **Heel MD** text area. NETool treats this first point as the top of the NETool model.

Deviation Survey

Select **Enter Directional Survey** to open the directional survey import screen. This screen will define the directional survey and calculate the well path based on the survey data.
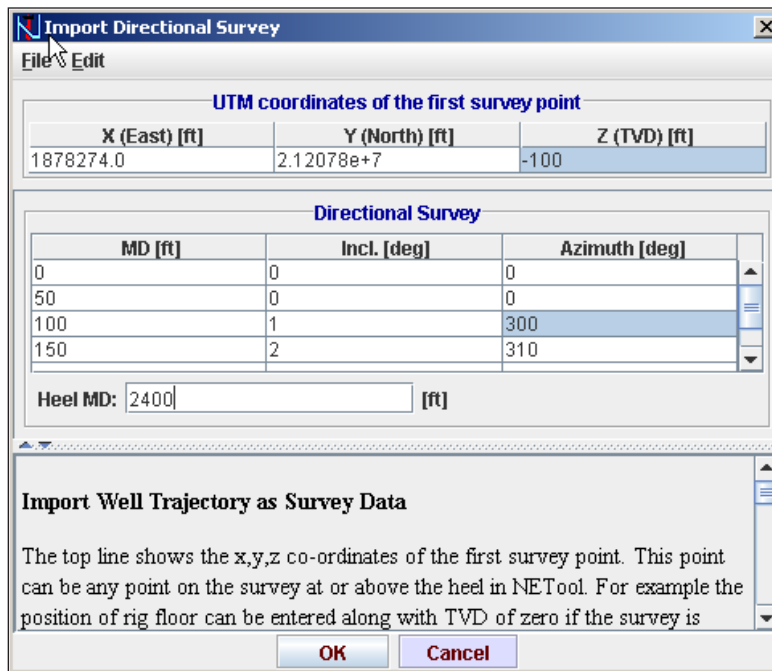
Figure 20. Example of Import Directional Survey Window

Directional survey data is a collection of (MD, Inclination, Azimuth) at different depths. MD can be in either *[ft]* or *[m]*. To change units, exit this menu and choose *Units* from the top menu bar. Inclination and azimuth must always be in degrees (azimuth is in 0-360 degree format).

Note: When using data derived from actual logs, importing the well path as a directional survey keeps the logs on depth. If the well trajectory is later modified using the graphical facilities of the reservoir view, logs may no longer be on depth.

Step 1 – Define location of the first survey point

Enter the x,y,z coordinate of the first survey point at the top line in the window.

An example, if the rig floor is the position of the first survey point then the coordinates of the rig floor must be supplied. If this is an offshore rig the z coordinate typically will be negative since the sea surface normally is at $z = 0$.

It is very important to get this right such that the well is located correctly in the reservoir. A wrongly defined top UTM coordinate leads to wrong position of the well in the reservoir.

The x,y,z position of the well will default to the centre and top of the reservoir (for convenience when using the BIGOIL or other predefined reservoir grids in the absence of your own Eclipse grid).

If you are using an Eclipse file from your reservoir, the position of the top node should be entered with the offset included in the Eclipse file.

Step 2 – Define all survey points
Supply the entire deviation survey set (MD, inclination, azimuth) by either i) file import or ii) copy-paste into the text area.

Step 3 – Define Heel MD of deviation survey
A complete survey dataset is necessary to locate the well correctly downhole. However, NETool's model region should only constitute the part of the well that covers the reservoir section (NETool is currently not designed for detailed simulation of flow form reservoir to the surface). Consequently the trajectory from the top down to the reservoir should be removed after the well location downhole has been defined.

In order to remove the trajectory data above the reservoir section the **Heel MD** needs to be entered. **Heel MD** in this context will define the top of NETool modeling region, and it should be approximately at top of the reservoir.

To sum up: Survey data above **Heel MD** is only used to calculate the x,y,z location down to the heel. The data above the heel will not be included in NETool after the x,y,z coordinates have been calculated..

Step 4 – Calculate Trajectory coordinates
After **OK** is pressed on this screen, the well path in are calculated in x,y,z coordinates. NETool does not save the survey data itself after the x,y,z coordinates have been calculated. It is the remaining x,y,z coordinates that will be kept.

Step 5 – Update Trajectory
In order to update the Trajectory after the x,y,z coordinate calculation has been conducted select **Update Trajectory**.

Define Heel MD (Top NETool Model node)

NETool treats **Heel MD** as the top of the NETool model (the models first point).

The top of the NETool model should normally be quite close to the top of the reservoir since NETool is not designed for detailed modeling of the flow from top completion up to the surface.

This is also the reference location for NETool's report of BHP. It can therefore be a good idea to let **Heel MD** coincide with the bottomhole pressure gauge or the pressure reference depth.

If **Heel MD** is changed only the reported measured depth of the well is changed; the calculated results are unaffected and the well stays at the same location in space.

Create and Change a Multi-Lateral Well Path Interactively

NETool can model complex multi-lateral wells. Multiple laterals are allowed as well as sub-laterals (in an endless fashion actually). All the capabilities to define and model well, completion and reservoir properties for the **Mainbore** apply to **Laterals**.

The lateral to be edited is selected in the **Wellbore** pull down menu located close to the top-left corner in the NETool window. When a well case is created there is always only one **Wellbore** named "Mainbore".

To create a lateral, just right click on the trajectory point where the junction is located, and choose add lateral. An alternative way to add as lateral is to push the **Add Sublateral** button in the left-bottom corner. NETool will automatically enter into the mode of editing the new lateral. This menu allows the trajectory of the lateral to be entered in a similar manner as for the **Mainbore**.

Lateral can be removed with the **Remove Lateral** button on the left hand side.

Remove Mainbore button deletes the entire well. When viewing a lateral the button display **Remove Lateral x.x** – and then the particular lateral is removed.

Add Sublateral adds a lateral to the active well trajectory as selected under **Wellbore**.

An effort should be made to have the lateral junction at a survey point in the **Mainbore** survey. If the top node of the lateral is not exactly a point on the **Mainbore** survey, NETool will choose the nearest point in the **Mainbore** survey as the junction and modify the position of the top of the lateral, or give an error message if the top of the lateral is not sufficiently close to the **Mainbore**.

When the lateral is imported as a deviation survey, it is best to choose the junction as an MD on the **Mainbore** with a trajectory point available – or assign a trajectory point on the **Mainbore** at the junction. If the top measured depth of the lateral survey is not a trajectory point on the **Mainbore**, NETool will either move the junction to the nearest trajectory point or add a trajectory point to the **Mainbore** at the junction.

Lateral Junctions Configuration

Five different junction types are available in NETool. The junction type for a simulation is chosen in the Segment Settings menu: **Segment Settings➔Hole & Completion➔Lateral Connection**:

To Tubing: lateral tubing is connected directly to mainbore tubing

To annulus (Stinger): lateral tubing is connected to mainbore annulus outside Stinger layer. This option is visible only when Stinger is present at the junction location.

To annulus (Inflow Control): lateral tubing is connected to mainbore annulus outside Inflow Control layer. This option is visible only when Inflow Control is present at the junction location.

To annulus (Sand Control): lateral tubing is connected to mainbore annulus outside Sand Control layer. This option is visible only when Sand Control is present at the junction location.

To annulus (Casing/Liner): lateral tubing is connected to mainbore annulus outside Casing/Liner layer. This option is visible only when Casing/Liner is present at the junction location and has Blank Pipe or Slotted Liner completion type.

Working with Segmentation

The next step after the well path is defined is to subdivide the well into segments. A segment is a part of the well with the same/constant reservoir and completion properties along entire segment length.

Segment boundaries are defined by nodes. Nodes are points along the well path.

Creation and deletion of nodes can be performed numerically or graphically.

The active lateral to be edited is chosen with the **Wellbore** dropdown menu located on the **Reservoir & Well Trajectory** tab in left upper corner.

Well segmentation is performed in the **Segments** mode. The segmentation cannot be modified in the **Trajectory** mode (which is the other alternative in the Show dropdown box).

Nodes should mostly be created and deleted from the **Segment Settings / Completion** panel, but it is also possible to create and delete nodes from:

- Graphical panels showing well trajectory
- Completion plot
- Log views.
- **Create Segments** button

Once the segments have been created, they appear in the **Segment Settings/Completion** sheet.

Well Segments panel allows the following operations:

- Modification of segment MD (select desired row and type a new number)
- Deletion of a segment interval (select row and press "delete" button)
- Insertion of a new segment (type desired MD in an empty row at the bottom)

Update MD must be pressed after each of these operations.

Nodes can be moved along the well path interactively in the graphical views or completion plot by dragging. Dragging is performed by left clicking and holding on a node, move the node to the desired position and release the mouse button.

After the segmentation is defined, consider to activate the **Locked** checkbox to make sure that the well path and segmentation will not be altered accidentally.

Well segments

Segments MD [m]
0.0
594.53
1189.07
1783.6
2378.13
2972.67
3170.83

Update MD   Cancel

Creating segments automatically.

Create Segments button is located in lower-right corner of **Segment Settings/Completion** screen. It allows to:

- Create a quantity of equally spaced nodes
- Create nodes at even spacing

With each of the automatic node creation options, the user has the option to delete the previous nodes (default option) or to retain the existing nodes and add additional nodes.

2. Once the nodes have been created, choose Update Nodes MD to assign these nodes in NETool. You can also click **Cancel** if you accidentally create an undesirable node.

Packers button allows to automatically add/remove packers on different completion layers between all segments. This option is useful when user needs to create a large amount of packers. It is usually easier to delete unnecessary packers afterwards then to create everything manually.

Create

◉ 10   segments

of the same length

◯ 100   m

long segments

Create Segments

☑ (Delete existing)

Packers...
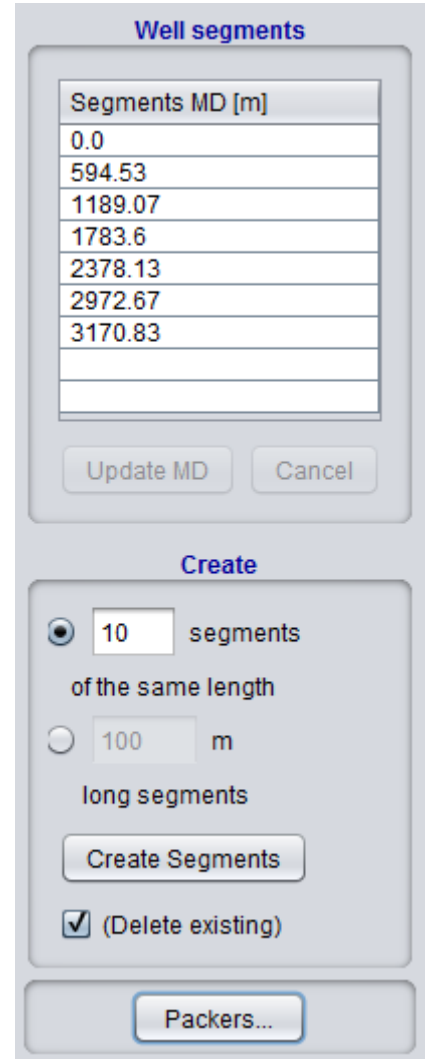
Global Settings Menu

<u>**General**</u>

The General tab is used to define these settings:

<u>**Well type**</u> – oil producer, gas producer or injector

<u>**Phase Mode**</u> – basic combinations of oil, gas and water

<u>**Boundary condition**</u> (or Simulation Target) – bottom hole pressure, tubing head pressure, various flow rate targets, IPR

<u>**Hydrostatics**</u> – Specify hydrostatic effects

<u>**Reservoir Pressure**</u> – Specify how to calculate hydrostatics in reservoir.

<u>**Pressure drop in tubing**</u> – correlations to be used inside the wellbore Homogeneous, Beggs and Brill, Aziz, Duns & Ros or OLGAS

Figure 21. Example of a General tab in the Global Settings Menu

Well type

Three options are available:

<u>Producer</u> – a producing well

<u>Injector, water</u> – a well injecting single-phase water.

<u>Injector, gas</u> – a well injecting single phase dry gas

Note: For the two injection well types, it is assumed pure gas or water flow, and that the relative permeability to the flowing phase is 1 (one). Mobilities under Reservoir Parameters on the Segment Settings/Completion Screen are not used and are unavailable when injection well is selected.

Phase mode

Any combination of oil, gas and water phases can be selected for a Producer.
Target for producers
NETool allows several options of Target rate/pressures as the outlet boundary condition:
<u>Flowing BH Pressure</u>: Specify the flowing Bottom Hole Pressure at the heel node.
<u>Tubing Head Pressure</u>: Specify the flowing Tubing Head Pressure. The lower bottom hole pressure limit and gas must be defined. Lift curves must be in the in the Eclipse type format and shall be entered in Fluid Properties -> VFP.
<u>Total Reservoir Rate</u>: Specify a total volumetric downhole flow rate in reservoir.
<u>Total Liquid rate</u>: Specify the desired flow rate of liquid in volumes at standard conditions.
<u>Oil Flow Rate</u> – Specify the desired flow rate of oil in volumes at standard conditions.
<u>Gas Flow Rate</u> – Specify the desired flow rate of gas in volumes at standard conditions.
<u>None, Make IPR</u> – NETool will generate an IPR curve for the pressure range specified in the number of step specified. With this mode, pressure and flow rate charts are not generated in the output.
<u>Calculate THP.</u> This check box is available when Target is other than Tubing Head Pressure (THP). If checked NETool will use VFP tables to calculate THP corresponding to the BHP or rate target solution. The calculated THP is then shown in the Results Summary Table.

Table 1: THP Results Summary

	Summary Table									
C	D	E	F	G	H	I	J	K	L	M
Phase mode	Oil rate	Gas rate	Water rate	GOR	WCUT	LGR	Q res. total	BHP	THP	Gas Lift Rate
Three phase	3562.39716	1.57522744	0	442.181871	0	0.00226151289	9216.38773	268.8953151	82.24772449	0
	[Sm3/d]	[MMSm3/d]	[Sm3/d]	[Sm3/Sm3]	[%]	[Sm3/Sm3]	[Rm3/d]	[Bar]	[Bar]	[MMSm3/d]

With some Targets an additional THP vs. BHP plot will be generated if the Calculate THP box is checked.

For tubing head pressure calculations, lift curves in the format generated by Prosper™ must be entered in Fluid Properties -> VFP.

Gas Lift Rate. This input field is available when i) Target is Tubing Head Pressure or ii) the Calculate THP box is checked. Gas Lift rate can be used with VFP curves includes gas lift. This input value is ignored if the VFP information does not include gas lift.

Hydrostatic

Several check boxes are used to control NETool hydrostatic pressure effects due to varying vertical depth TVD along the well.

Use Hydrostatic - when checked, the effect of vertical differences along the well are included in the NETool calculations.

If reservoir grid is not present or contains inconsistent values user may want to use reservoir pressure at a certain point and let NETool recalculate hydrostatic difference. This functionality is included for convenience so that the user does not need to manually adjust every reservoir pressure for hydrostatic differences along the well.

There are several options available through Reservoir Pressure from switch (It appears only when Use Hydrostatic is ticked.):

- Defined Along the Well for Each Segment - Netool will use values from 'Segment Settings→Reservoir Parameters→Reservoir Pressure'. It assumes proper reservoir pressures are entered along the well.

- Single Value at Heel + Hydrostatics - NETool takes pressure at BH location in reservoir and recalculates reservoir pressure profile based on vertical difference from the heel and fluid densities flowing into the wellbore.

- Single Value at Some MD + Hydrostatics - NETool takes pressure at some MD defined by user and recalculates reservoir pressure profile based on vertical difference from the heel and fluid densities flowing into the wellbore. This option is useful if vertical well penetrates a thin production zone.

Pressure Drop in Tubing and Annulus

NETool currently offers five options to calculate fluid flow performance in the tubing and open annulus:

Homogenous – uses a single phase flow correlation as commonly used throughout the oil industry using average properties of the phases present in that section of the well.

Beggs & Brill – uses the two phase (gas/liquid) flow correlations as published by Beggs and Brill. This correlation is the best for horizontal pipes.

Aziz, Govier, Fogarasi is based on publication "Pressure Drop in wells Producing Oil and Gas" by the same authors in the Journal of Canadian Petroleum 72-03-04.

Duns, Ros is based on "Vertical flow of gas and liquid mixtures in wells" by

H. Duns Jr. and N. C. J. Ros WPC 10132 1963.

OlgaS – the OLGA-S options only appear when the OlgaS.dll from Scandpower is available. This dynamic link library is an add-on feature to NETool.

Aziz and Duns&Ros models are applicable only to vertical flow in a pipe if liquid and gas are present. Therefore if selected by user they will be applied only to segments which satisfy both conditions:

Inclination is 45-90° to horizontal.
No annulus flow.

If a segment does not satisfy these conditions it will use "Homogenious" pressure drop model.

BJ	BQ	BR
Well	Pressure drop	Grid I,J,K
Inclination	correlation	
[°deg]		
26.5763	Duns Ros	0,0,0
26.5763	Duns Ros	0,0,0
29.5598	Duns Ros	0,0,0
29.5598	Duns Ros	0,0,0
29.5598	Duns Ros	0,0,0
29.5598	Duns Ros	0,0,0
32.5363	Duns Ros	0,0,0
32.5363	Duns Ros	0,0,0

Figure 22. Pressure drop correlation is reported in "Details".

In order to simulate vertical lift performance using these models user should set "Target" to BHP or any Flowrate. Setting "target = Tubing Head Pressure" will enable imported lift table usage from the heel to the surface.

PLT

An option to enter PLT data (water flowrate, total downhole flowrate and pressure as a function of MD) is available. Toggle on **Overlay output with PLT data** check box and click the **PLT Data** button, enter data or copy past from other sources as for instance a spreadsheet. PLT results are presented together with simulated results for the tubing in the **Plots** viewer.

<u>**Inflow (PI Models)**</u>

Productivity Index models are selected and configured in the screen **Global Settings→Inflow.** PIs are normally calculated locally for each segments based on individual segment's properties.

This section presents an overview of the PI model options and configurations. Vertical, horizontal and deviated PI models are discussed in more detail in the NETool Technical Manual.

<u>Well PI</u> dropdown menu

<u>Auto</u> is the default option, which facilitate calculation of all the local PIs based on reservoir properties and configuration of PI models

Manual is an option where the well's total PI is supplied by the user. Local PIs for each well segment will be scaled according to local permeabilities so that the sum of local PIs is equal to the total PI entered. This option can be good if you want to tune the NETool PI to an observed PI from well test. The sum of local PIs are scaled to match the total PI.

Set PI model for each segment checkbox

Box unchecked (default): PI parameters for the whole well are defined as described below.

Box checked: PI parameters can be specified for each segment individually in the

> **Segment Settings/Completion** menu using the folder **Reservoir Parameters →Transmissibility→PI model**.

PI Model Type dropdown menu:

Steady State: This mode is appropriate when there is full pressure maintenance of the reservoir. It assumes constant reservoir pressure versus time and the models assume constant pressure boundaries.

Semi-Steady State PI Model: This mode is appropriate for reservoirs experienced pressure depletion. It assumes a constant pressure drop versus time throughout the reservoir.

A summary of the input parameters for vertical and horizontal PI models is supplied in the **Table 2** and **Table 3**.

Detailed configuration of the PI models is covered in the next sections.

Table 2: PI Models for Vertical Wells - Input Parameters

	Reservoir State	
Input items	Steady state	Semi-steady state
Reservoir Dimension	Radial extent	Circular region: Radial extent Irregular region: Size of area & shape factor
Pressure Status	Pressure at reservoir boundary, or Average reservoir pressure	Pressure at reservoir boundary, or Average reservoir pressure

Table 3: PI Models for Horizontal Wells - Input Parameters

	Reservoir State	
Input items	Steady state (Joshi)	Semi-steady state (Babu & Odeh)
Reservoir Dimension	Reservoir Thickness in the vertical plane and ellipsoid in the horizontal plane where the width of the ellipsoid is given	Reservoir Length Reservoir Width Reservoir Thickness
Well Position	In the center of the reservoir Thickness and the ellipsoid	Length position of well Width position of well Depth position of well
Pressure status	Pressure at reservoir boundary (ellipsoid)	Average reservoir pressure of drainage volume

PI for Deviated Well Sections

NETool combine PI models for horizontal and vertical wells to account for the full range of well deviations.

The following scheme when defining the PI model for the different well segments:

0-10° - Vertical Well PI model

10-80° - Deviated Wells PI - Combination of Vertical and Horizontal PI Models

80-90° - Horizontal Well PI Model

Steady State Models

Vertical PI Model

The Steady State Vertical PI model is used for vertical sections of the well, and assumes:

Constant pressure boundary at radial distance from the well

Radial Darcy flow

Required input parameters:

Radial extent of reservoir: defines the distance to the constant pressure boundary.

Vertical PI based on: defines whether the vertical PI is based on average pressure or external boundary pressure

Pres at Re: Vertical Well PI is calculated using Reservoir Pressure values as pressures at the radial extent of the reservoir. This option is appropriate when manually entered reservoir pressures are pressures at the radial extent of the reservoir, or when pressures populated from the grid are pressures at initial conditions.

Average Reservoir P: Vertical Well PI is calculated using Reservoir Pressure values as the average pressure. This option is appropriate when manually entered reservoir pressures are the average reservoir pressures.

Discussion about configuration.

The appropriate value is dependent on i) type of supplied pressure and ii) size of the reservoir.

Manually entered Reservoir Pressure Values – it is recommended to enter the radial distance to the constant pressure boundary for the well.

Reservoir Pressure taken from the grid at initial conditions - this is the case for making time=0 simulations of the well based on reservoir pressure values from the grid - the well has not yet caused localized drawdown around the well. If the reservoir conditions are such that pressure in the reservoir is well supported (steady state conditions), it is then recommended to enter the actual drainage radius of the well, and set the choice Vertical Well PI based on: to Pres at Re.

Reservoir Pressure taken from the grid/restart files after initial conditions - it is often more appropriate to use the semi-steady state PI model for these conditions.

Horizontal PI Model

The Steady State Horizontal PI model is using the Joshi model for horizontal wells.

The Joshi model assumes that the well is centered in the reservoir. NETool distributes the total PI along the well according to individual segment properties.

Configuration parameters for steady state horizontal wells are illustrated in **Figure 23**.

Reservoir Thickness: Is the vertical thickness of the reservoir (contributing to flow).

Reservoir Width: Joshi assumes a constant pressure boundary with a shape as an ellipse in the horizontal plane where the well is located symmetrically. The reservoir width is the width of ellipse forming the constant pressure boundary in the horizontal plane.

Netool assumes that well length is equal to reservoir length. (In other words Rend = 0).

Reservoir Pressure: Is the pressure at the constant pressure boundary.
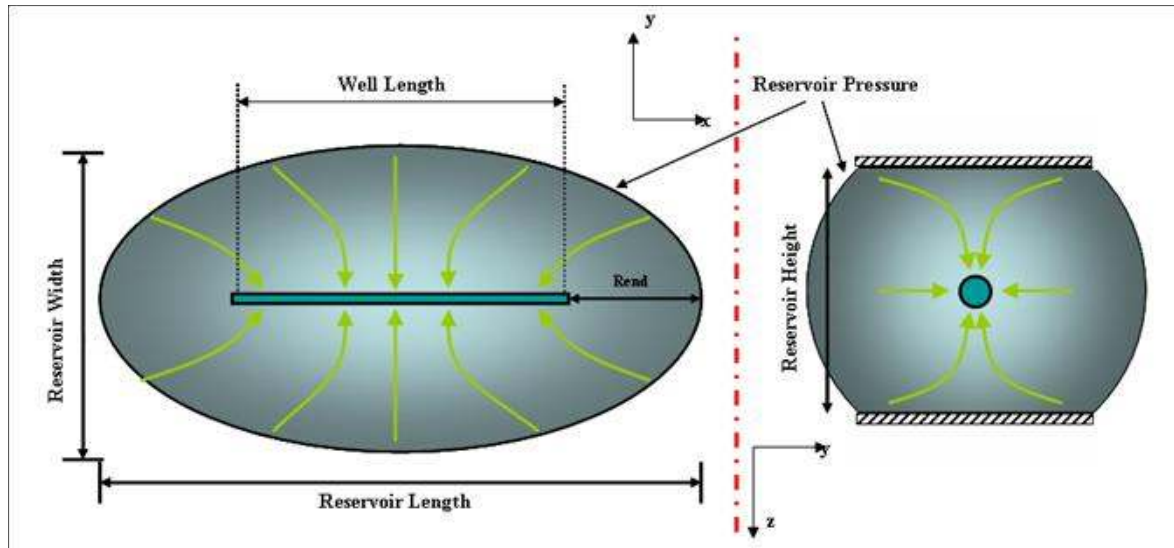
Figure 23. PI Model Parameters for Steady State Horizontal Well (Joshi)

<u>**Semi-Steady State Models**</u>

Vertical PI Model

The Semi-Steady State Vertical PI model is used for vertical sections of the well, and assumes constant depletion rate of the entire reservoir with radial Darcy flow equation model for radial reservoirs, or the use of Dietz shape factors for irregular reservoirs

<u>Drainage Region</u> – has the following two options which trigger various sub-options:

<u>Use circular drainage region</u>

<u>Radial extent of the reservoir</u>

<u>Vertical PI based on:</u>

<u>P at Re.</u> Vertical Well PI is calculated using Reservoir Pressure values as pressures at the radial extent of the reservoir. This option is appropriate when the reservoir pressures entered manually are pressures at the radial extent of the reservoir, or when taking pressures from the grid at initial conditions.

<u>Average Reservoir P.</u> Vertical Well PI is calculated using Reservoir Pressure values as the average pressure. This option is appropriate when the reservoir pressure entered manually is the average reservoir pressure, or when taking the reservoir pressure from the grid when using a restart file at a future time.

<u>Use irregular region</u>

<u>Drainage area</u>

<u>Drainage area shape - (Dietz) factor</u>

Discussion about configuration.

The Semi-Steady State PI model assumes a constant pressure drawdown through the reservoir.
The appropriate value to enter here is dependent on where the reservoir pressure is taken from and the size of the reservoir.

Manually entered Reservoir Pressure Values - it is appropriate to enter the radial distance to the reservoir boundary. Also, ensure the reservoir pressure values are appropriate.

Reservoir Pressure taken from the grid with grid at initial conditions - this is the case for making time=0 simulations of the well based on reservoir pressure values from the grid - the well has not yet caused localized drawdown around the well. If the reservoir conditions are such that pressure in the reservoir is not supported (semi-steady state conditions), it is then appropriate to enter the actual drainage radius of the well, and set the choice Vertical Well PI based on: to Pres at Re.

Reservoir Pressure taken from the grid from a restart file at future time - this is the case for making future time simulations of the well with reservoir pressure taken from the grid. In this case, there will be localized drawdown around the well. NETool will be using as reservoir pressure the average pressure of the gridblock the well passes through. It is appropriate then to set Vertical Well PI based on: to Average Reservoir P. The drainage radius should be set to the Peaceman radius of the gridblock.

Horizontal PI Model

The Semi Steady State Horizontal PI model is using the Babu & Odeh formulation for horizontal wells.

The Babu & Odeh model considers the position of the well inside a rectangular reservoir box. The well is parallel to rectangular reservoir box but not need to be centralized in the box. See illustration in **Figure 24**. Configuration parameters for Semi Steady State horizontal wells are illustrated in **Figure 24**.

Reservoir Dimensions:

Reservoir Length – is the length of the reservoir in the direction parallel to the well.

Reservoir Width – is the width of the reservoir in the direction perpendicular to the well

Reservoir Thickness - is the vertical thickness of the reservoir (contributing to flow).

Well Position - the following items describe the Well Position in the reservoir:

Length position of well - is the distance from one of the end of the reservoir to the middle of the well in the direction parallel to the well.

Width position of well - is the distance from the well to the extent of the reservoir in the horizontal plane perpendicular to the well direction.

Depth position of well – the depth position is relative to the reservoir thickness, i.e. TVD from top of reservoir.

Reservoir Pressure – Is the average pressure
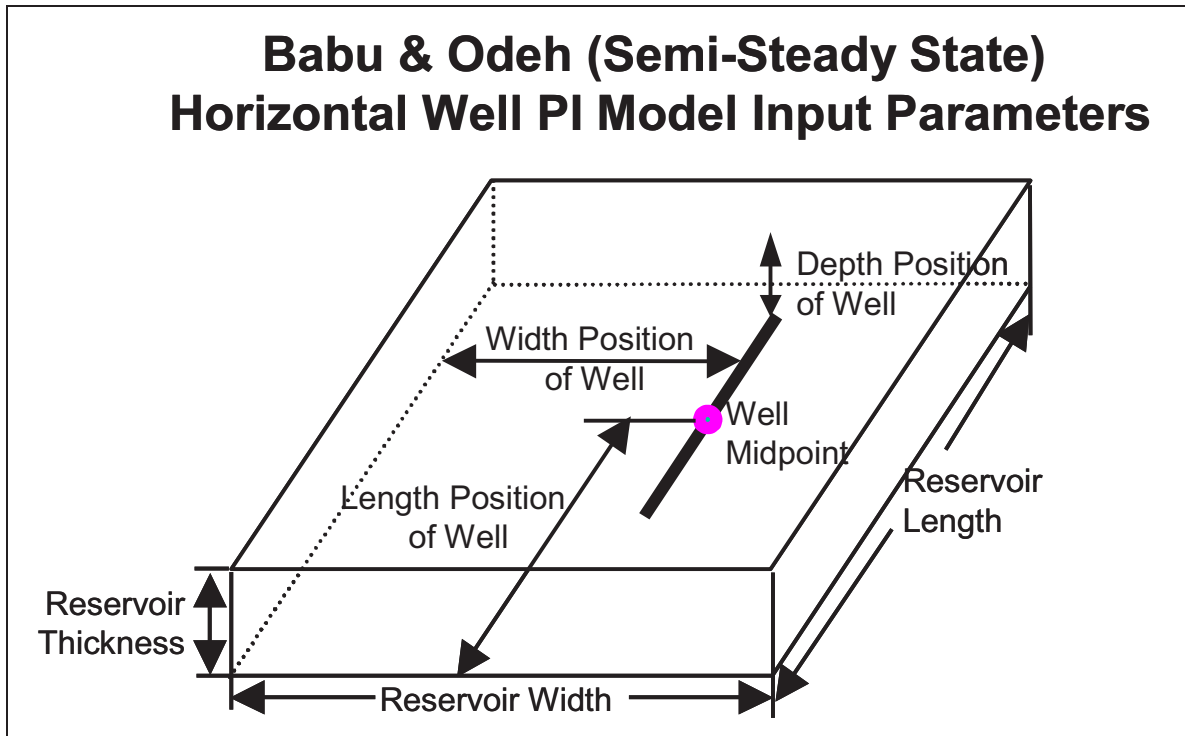
Figure 24. Model Parameters for Semi-steady State Horizontal Well (Babu & Odeh)

Advanced

This menu controls numerical parameters and is important for defining the accuracy and handle potential numerical problems. Manipulation of the parameters below may help solving cases that otherwise wouldn't converge.

Precision of calculations

This sets the allowable numerical error. Reducing this term will make NETool more accurate, but leads to longer run times and may lead to convergence problems.

Stability

The NETool solver cannot accept flow rates of zero in any part of the numerical network being solved (all flow rates in NETool are limited to a small non-zero value). The stability term sets the minimum flow rate allowed in NETool. This is the local flow rates (local influx rates, local annulus flow rates, local tubing flow rates, local flow rate across the completion, etc). For some very low flow rate cases, it is appropriate to reduce this number as the very small flow rates may change the results.

A bit of thought on number of intermediate nodes for the well, such that areas of low influx do not have large numbers of intermediate nodes can also help with both "Problems with Decomposition" and "Unable to determine flow direction" errors.

A Stability value of $1 = 1\text{E-8 Sm}^3/\text{sec}$.

Flow can change directions

NETool automatically solves for flow directions inside the well. For some pressure distributions, for example cross-flow situations can exist. To prevent NETool from looking for such solutions uncheck appropriate check boxes.

Well Pressure Range

Three parameters are available:

Well Pressure Limit- Auto or Manual

Min well pressure (only available if Well Pressure Limit is Manual)

Max well pressure (only available if Well Pressure Limit is Manual)

By default, NETool automatically chooses the allowed well pressure range. This operation is necessary for numerical convergence considerations. However, for cumbersome cases with multiple laterals and cross-flow this may not work satisfactory.

If convergence problems is experienced, one remedy could be to define minimum and maximum pressures manually (i.e. bottom-hole pressure as lower and maximum reservoir pressure as upper limit) in order to assist in solving the case.

Momentum balance

Momentum balance has the following options:

Improved momentum balance – enables compressible Bernoulli model for completions

Max Mach number – sets up the limit on how close to the speed of sound subcritical flow can be.

Bernoulli for diameter variations- enables compressible Bernoulli model for annulus and tubing diameter variations. Uses either upstream density (Approximate option) or average of upstream and downstream (Almost precise)

Transitional flow regime

Low Reynolds – Lower limit for transitional flow in terms of Reynolds number. A reasonable number is 2000.

High Reynolds – Upper limit for transitional flow in terms of Reynolds number. A reasonable number is 3000.

Convergence Tuning

Options to control numerical calculations and convergence are:

Iteration step dumping – limits how fast variables can change between iterations.

Water derivatives - Controls water derivatives in the numerical solver. Using *Switching* or *Off* will help to converge cases with high oil viscosity and uneven water inflow.

Smooth Pres over - Defines window size for moving average filtering applied to reservoir pressure. Reservoir pressure usually exhibits unphysical jumps between Eclipse grid blocks. This pre-processing makes pressure profile more physical and improves convergence. Suggested to be 4-20 the size of a grid block.

Local density in hydrostatics - May help to reach convergence for cases with oscillating wellpath or wrong reservoir pressure coming from large grid blocks.

Simplified Rs/Rv handling - Enables an old model for Rs and Rv used in Netool v5000.0.0.1 and earlier. This model assumes a single average Rs and Rv along the well. It's recommended to keep this checkbox selected as the new model is in testing stage. Unselect this checkbox if your model contains more than one production zone.

PVT derivatives

PVT derives in MatBal

PVT derives in inflow acceleration

PVT derives in friction

PVT derives in completion

Enabling or disabling these options may help to reach convergence in hard cases.

Output

This menu controls which plots and tables that should be produced by default, most of the output items are turned on.

Annotation

This field is useful to provide detailed description of the well case. It is usually a good practice to make an expanded description of important well cases for future documentation and understanding.

Fluid Properties Menu

The Fluid Properties Menu is used for viewing, adding or modifying:

- PVT data (required)
- Relative Permeability data (optional)
- VFP (Lift curves) data (optional)

The Fluid Properties Menu contains the tabs PVT, RelPerms, Lift to for selection of the respective property submenu.

PVT data must always be present while relative permeability and lift curves are optional and may be left out.

Relative permeability is required if mobilities are to be calculated from relative permeabilities and saturations.

Lift curve(s) is required if the simulation is produced against the tubing head pressure target.

PVT data and relative permeability data will be automatically imported if an Eclipse file dataset is imported.

The Lift Curves (VFP tables) are normally generated with software such as Prosper. The Lift curves can be copy-pasted into NETool.

PVT

NETool supports black-oil PVT formulation with dissolved gas, vaporized condensate and multiple bubble/dew points.

Fluid properties can be supplied in two different formats: i) NETool's PVT tables or ii) Eclipse style PVT tables.

PVT data can be viewed, edited and entered from the PVT tab of the Fluid Properties Menu.

PVT data can be prepared outside NETool for copy-pasting into the NETool input window, or alternatively saved as a file and imported into NETool.

The PVT window has two tabs:

Oil-Gas PVT data

Water PVT data

The content of these tabs will depend on the chosen item in the "PVT data from" pull down menu:

From PVT table – a table is shown. **It is initially populated with an example set of values**. User is supposed to fill it correct data i) by copy-pasting from other program or ii) by using PVT correlations.

From Eclipse INIT file – NETool will extract PVT data from the Eclipse INIT file and present the content in the text area as Eclipse type PVT format.

<u>From PVT data in Eclipse format</u> – shows a blank text area which allows user to enter or paste data in Eclipse type PVT format.

PVT data used by NETool can be inspected graphically. To view PVT data plots choose <u>View</u>➔<u>Visualize PVT and RelPerms</u>. Additionally, PVT data is always shown in the <u>output plots after a simulation.</u> **It is recommended to look at the PVT plots for every new PVT table to have a first glance if they look physically reasonable. Wrong or inconsistent PVT data can also lead to convergence problems in addition to wrong calculations**.
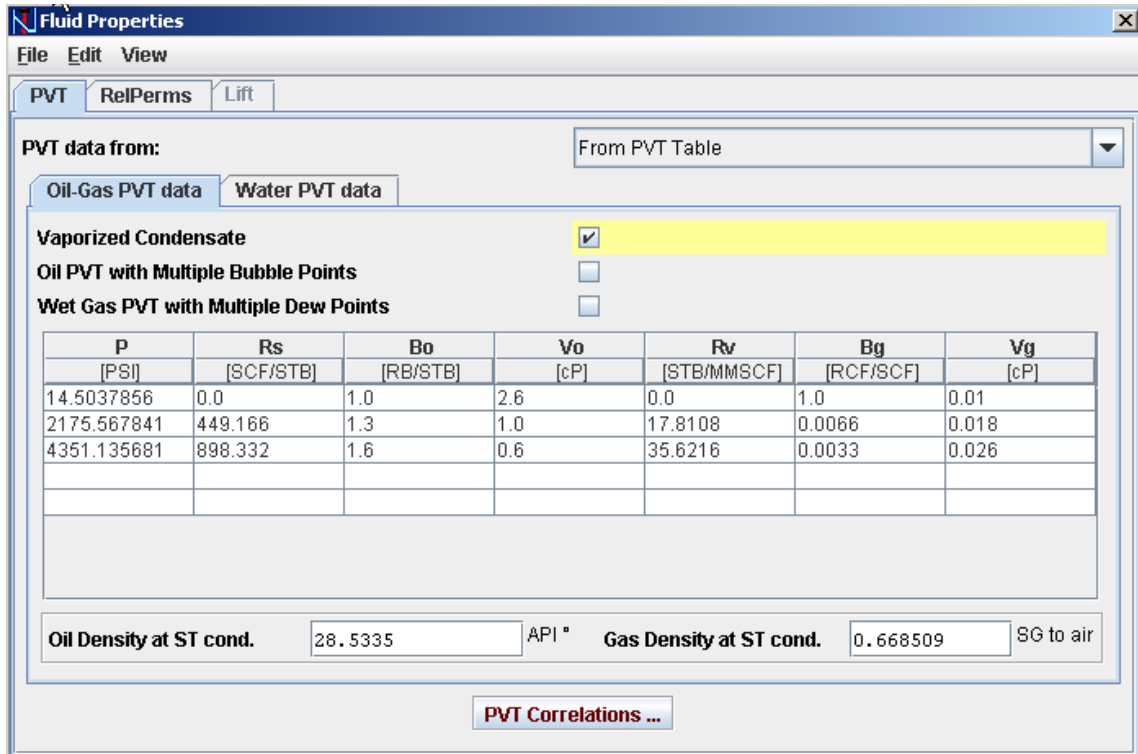
Figure 25. PVT Input Screen

PVT Correlations

The PVT correlations capability provides generation of PVT data based on limited PVT information. Minimum required information are oil and gas gravities plus some default water properties.

The facility is available when working in the *From PVT Table* mode. The PVT correlations button is located below the PVT table. Selecting the button opens a new window which allows the user to specify and configure the correlations.

After configuring the correlations PVT data can be generated by pushing the Generate PVT button. The generated PVT data populates the PVT table.

Table 4 shows the simple industry-standard PVT correlations available.

Table 4: PVT Correlations

PVT Parameter	Correlation
Solution Gas Oil Ratio	Standing
Bubble Point pressure	Standing
Oil Formation Volume Factor ($P > P_{bp}$)	Standing
Oil Formation Volume Factor ($P < P_{bp}$)	Standing
Oil compressibility	Standing
Dead oil viscosity	Glasø
Saturated Oil Viscosity	Standing
Undersaturated Oil Viscosity	Standing
Pseudo Critical Temperature	Sutton
Pseudo Critical Pressure	Sutton
Gas Viscosity	Carr et al., Dempsey, Standing
Z-factor	Hall and Yarborough
Gas Formation Volume Factor	Equation of State
Water compressibility	Rowe and Chou
Water Viscosity	Kestin
Water Density	Rowe-Chou
Water Formation Volume Factor	Rowe-Chou

Oil and Gas PVT Properties

Oil and gas PVT are based on black-oil formulation. It also allows for vaporized oil, condensate, wet gas, multiple bubble points and multiple dew points if desired.

Capabilities for i) vaporized oil, ii) multiple bubblepoints and iii) multiple dewpoints are controlled by the checkboxes shown in **Table 4**.

More PVT calculation details are covered in NETool Technical Manual.

PVT data can be produced graphically by **Fluid Properties→View→Visualize PVT & Relperms** for inspection. The same plots will also be produced by NETool in the output plots after a simulation.

Oil PVT with Multiple Bubble Points

The check box Oil PVT with Multiple Bubble Points can activate multiple bubble point capabilities.

When the option is activated the columns in the PVT table are split to distinguish saturated and under-saturated regions for Oil Formation Volume Factor (B_o) and Oil Viscosity (μ_o).

R_S, $B_{O,SAt}$ and $V_{O,SAT}$ columns in the table define **saturated** data as a function of pressure.

$C_{O,U-SAT}$ and C $V_{O,U-SAT}$ columns define compressibility and viscosibility of **undersaturated** oil.

Also the input for Rs will appear in the **Segment Settings→Reservoir Properties** sheet.

When the *Multiple Bubble Points* option is not activated, NETool assumes that the R_S, B_O and V_O columns in the table define the **actual** oil PVT data as a function of pressure. R_S must be increasing up to the bubble point and must be constant at higher pressures than the bubble point.

Vaporized Condensate

The check box *Vaporized Condensate* adds the R_V column PVT data. R_V contains vaporized oil as function of pressure.

Wet Gas PVT with Multiple Dew Points

The check box *Wet Gas PVT with Multiple Dew Points* can activate multiple dew point capabilities.

When the option is activated, the columns in the PVT table are split to distinguish wet and dry gas for Gas Formation Volume Factor (B_G) and Gas Viscosity (V_G).

R_V, $B_{G,WET}$ and $V_{G,WET}$ columns in the table define **saturated** data as a function of pressure.

$B_{G,DRY}$ and V_g,dry columns define properties of **dry** gas.

Also the input for R_V will appear in the Segment Settings→Reservoir Properties Sheet.

When the *Multiple Dew Points* option is not activated, NETool assumes that the R_V, B_G and V_G columns in the table define the **actual** wet gas PVT data as a function of pressure. R_V must be increasing up to the dew point and must be constant at higher pressure than the dew point.

PVT Tables Summary

Oil and Dissolved Gas

[black units] indicate units in METRIC.

{blue units} indicate FIELD units.

PVT table for oil with dissolved gas with one bubble point:

Table 5: Oil and Gas Dissolved

P Pressure [Bar] {PSI} Must be increasing down the column	Rs Solution Gas/Oil Ratio [Sm3/Sm3] {SCF/STB}	Bo Oil formation volume factor [m3/Sm3] {RB/STB}	Vo Oil viscosity [cP] {cP}	Bg Gas formation volume factor [Rm3/Sm3] {RCF/SCF}	Vg Gas viscosity [cP] {cP}
...	...	...	...	...	...

Oil above bubble point with constant Dissolved Gas

The simplest possible black oil PVT table is for undersaturated oil produced above bubble point. You get this configuration if you uncheck *Gas* but keep *Oil* under *Global Settings→General→Phase Mode*.

Table 6: Oil and Bubble Point With Constant Dissolved Gas

P Pressure [Bar] {PSI} Must be increasing down the column	Bo Oil formation volume factor [m3/Sm3] {RB/STB}	Vo Oil viscosity [cP] {cP}
...	…	...

This mode requires that you enter *Bubble Point* and *Dissolved Gas* (constant R_S value) in the inputs below the table.

Oil and Wet Gas

In case vaporized condensate is activated an additional column – R_V (highlighted below) – is added to the PVT table:

Table 7: Oil and Wet Gas

P Pressure [Bar] {PSI} Must be increasing down the column	Rs Solution Gas/Oil Ratio [Sm3/Sm3] {SCF/STB}	Bo Oil formation volume factor [m3/Sm3] {RB/STB}	Vo Oil viscosity [cP] {cP}	Rv Vaporized Condensate/Gas Ratio [Sm3/Sm3] {STB/MMSCF}	Bg Gas formation volume factor [Rm3/Sm3] {RCF/SCF}	Vg Gas viscosity [cP] {cP}
...	...	...	...	…	...	...

Dry Gas

When modeling dry gas this PVT table is presented:

Table 8: Modeling Dry Gas

P Pressure [Bar] {PSI} Must be increasing down the column	Bg Gas formation volume factor [Rm3/Sm3] {RCF/SCF}	Vg Gas viscosity [cP] {cP}
...	…	...

Wet Gas with Multiple Dew Points

Modeling vaporized condensate with multiple dew points PVT table is presented (Vaporized condensate + Wet gas PVT with multiple dew points turned ON):

Table 9: Vaporized Condensate with Multiple Dew Points

P Pressure [Bar] {PSI} Must be increasing down the column	Rs Solution Gas/Oil Ratio [Sm3/Sm3] {SCF/STB}	Bo Oil formation volume factor [m3/Sm3] {RB/STB}	Vo Oil viscosity [cP] {cP}	Rv Vaporized Condensate/ Gas Ratio [Sm3/Sm3] {STB/MMSCF}	Bg, wet Wet gas formation volume factor [Rm3/Sm3] {RCF/SCF}	Bg, dry Dry gas formation volume factor [Rm3/Sm3] {RCF/SCF}	Vg, wet Wet gas viscosity [cP] {cP}	Vg, dry Dry gas viscosity [cP] {cP}
...	...	...	...	...	...	...	...	...

Oil with Multiple Bubble Points

Modeling oil with multiple bubble points gives this PVT table (Oil PVT with multiple bubble points). Undersaturated oil properties (B_O and viscosity) are calculated by NETool by the use of oil compressibility and viscosibility.

Table 10: Oil with Multiple Bubble Points

P Pressure [Bar] {PSI} Must be increasing down the column	Rs Solution Gas/Oil Ratio [Sm3/Sm3] {SCF/STB}	Bo, sat Saturated oil formation volume factor (at bubble point) [Rm3/Sm3] {RCF/SCF}	Co, u-sat Under-saturated oil compressibility (above bubble point) [1/bar] {1/PSI}	Vo, sat Saturated oil viscosity (at bubble point) [cP] {cP}	CVo, u-sat Under-saturated oil viscosibility (above bubble point) [1/bar] {1/PSI}	Bg Gas formation volume factor [Rm3/Sm3] {RCF/SCF}	Vg Gas viscosity [cP] {cP}
...	...	...	...	...	...	...	...

Oil and Wet Gas with Multiple Bubble Points and Multiple Dew Points

Depending on the PVT options chosen, oil PVT table may have between six and eleven columns.

Vaporized Condensate	✔
Oil PVT with Multiple Bubble Points	✔
Wet Gas PVT with Multiple Dew Points	✔

P	Rs	Bo,sat	Co,u-sat	Vo,sat	Cvo,u-sat	Rv	Bg,wet	Bg,dry	Vg,wet	Vg,dry
[PSI]	[SCF/ST...]	[RB/STB]	[1/PSI]	[cP]	[1/PSI]	[STB/M...]	[RCF/S...]	[RCF/S...]	[cP]	[cP]
14.5037...	0.0	1.0	1.72368...	2.6	1.37895...	0.0	1.0	1.0	0.01	0.01
2175.56...	449.166	1.3	1.72368...	1.0	1.37895...	17.8108	0.0066	0.0070	0.018	0.017
4351.13...	898.332	1.6	1.72368...	0.6	1.37895...	35.6216	0.0033	0.0035	0.026	0.025

Figure 26. PVT Layout

Figure 21 shows PVT table layout with all possible columns (multiple bubble and dew points).

<u>**Water PVT Properties**</u>

Water PVT Properties are entered in the **Fluid Properties Menu→ PVT → Water PVT** data tab. NETool requires:

ρ_w at stock tank conditions
P_{ref}
$B_w (P_{ref})$
$\mu_w (P_{ref})$
c_w
Water viscosibility
NETool provides reasonable default values for these parameters.

Eclipse PVTW keyword supplies water properties when *From PVT data in Eclipse format* is chosen in the *PVT data from* dropdown menu.

<u>**Eclipse PVT Format**</u>

PVT data in Eclipse format may be entered into PVT text area as an alternative to NETool PVT table input.

NETool will accept Eclipse PVT data in either oilfield units or metric units. The units of the PVT file are designated by a keyword **<u>at the top of the file</u>**. Two unit sets are supported by NETool defined by the keywords METRIC or FIELD. **Please note that Eclipse FIELD units are slightly different from NETool FIELD units**

Following Eclipse PVT keywords are recognized by Netool:

Units system: **METRIC** or **FIELD**.

The units keyword must be at the top of the file — it must be the first keyword.

Densities at stock tank conditions: **DENSITY**.

This keyword must always be entered.

Oil properties:

Live oil: **PVTO** or **PVCO**

Dead oil: **PVDO** or **PVCDO**

Exactly one of these three keywords must be entered if oil phase is present.

Gas properties:

Dry gas: **PVDG**

Wet gas: **PVTG**

Exactly one of these two keywords must be entered if gas phase is present.

Water properties: **PVTW**.

This keyword must be entered if water phase is present.

RSCONST keyword is recognized in phase modes with no free gas (1-phase oil or 2-phase oil+water).

Units used by Eclipse are as follows:

Quantity		METRIC unit	FIELD unit
P	Pressure	[Bar]	[PSI]
Rs	Solution Gas/Oil Ratio	[Sm³/Sm³]	[MSCF/STB] [1]
Rv	Vaporized Condensate/Gas Ratio	[Sm³/Sm³]	[STB/MSCF] [1]
B_o	Oil Formation Volume Factor	[Rm³/Sm³]	[RB/STB]
B_g	Gas Formation Volume Factor	[Rm³/Sm³]	[RB/MSCF] [1]
B_w	Water Formation Volume Factor	[Rm³/Sm³]	[RB/STB]
V, µ	Viscosity (oil, gas or water)	[Cp]	[Cp]
C	Compressibility (oil, water)	[1/Bar]	[1/PSI]
CV	"Viscosibility" (oil, water)	[1/Bar]	[1/PSI]
ρ	Density at Stock Tank Conditions (oil, gas and water)	[kg/m³]	[lb/ft³][1]

[1] Different from NETool FIELD unit !

The option PVTNUM from will only appear when PVT data is supplied in Eclipse format. The Eclipse PVT format allows for multiple PVT regions and the **PVTNUM** assigns the correct PVT dataset to the region.

<u>**Relative Permeability**</u>

Relative permeability is part of the Mobility term used in reservoir flow equations.

Relative permeability is determined as a function of fluid saturations, the functions can be either curves and/or correlations.

The RelPerm data from dropdown menu selects the option to be used when determining relative permeability.

Five options are available:

From Relperm table – presents relative permeabilities in tables. The number of tables and table types are defined by the Two-phase Oil Relperms dropdown menu.

From Wyllie's CS, OL correlation - Wyllie's correlations for cemented sandstone, oolitic limestone is used. NETool Technical Manual provides more details.

From Wyllie's US, WS correlation – Wyllie's correlations for unconsolidated sand, well sorted is used. NETool Technical manual provides more details.

From Eclipse INIT file – window shows a text area with the relative permeability table(s) extracted from the Eclipse simulation model, if such data is available in the INIT file. The data is presented in Eclipse PVT format. The content is not open for edit.

From Relperm data in Eclipse format – has the same look as the option above with the difference that the text area is by editable and by default empty. User can also copy text from the content in the option From Eclipse INIT file for purpose of further editing.

Relative Permeability curves can be produced by **Fluid Properties➔View➔Visualize PVT & Relperms** for inspection. The same plots will also be produced by NETool in the output plots after a simulation.
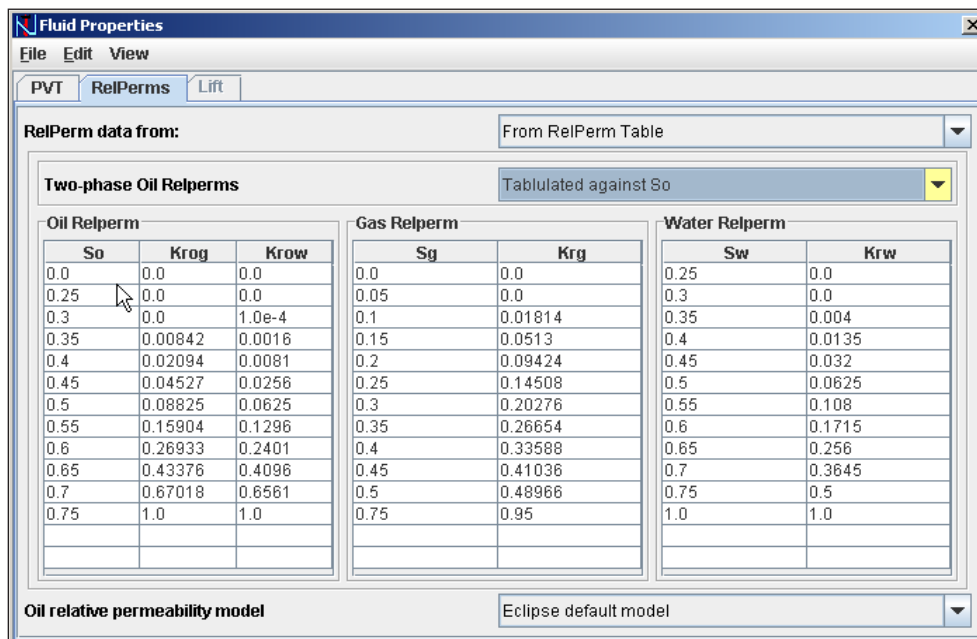
Figure 27. Relative Permeability Input Screen

Gas and Water Relative Permeability

Gas and water relative permeability are calculated in a straight forward manner as a function of gas and water saturation from the values entered in the Relative Permeability Table:

$k_{rg} = F(S_g)$

$k_{rw} = F(S_w)$

Oil Relative Permeability

Four methods of calculating three-phase oil relative permeability k_{ro} are supported.

The selected model is determined by the Oil relative permeability model dropdown menu at the bottom of the Relative Permeability window.

Details of the four models are described in NETool Technical Manual:

Eclipse default model

Stone's 1st model

Stone's 1st model modified by Fayers and Matthews

Stone's 2nd model modified by Geoquest (Eclipse)

Relative permeability Eclipse Format

This section contains the description when using Relative Permeability in Eclipse format.

If several saturation regions exist in the reservoir, relative permeability files may contain multiple tables, one for each region. NETool will use relative permeability tables from the well segments corresponding reservoir regions. The table number to use is determined by **Reservoir&Well→Reservoir Data→Sat. Region** (the field is visible only if **SATNUM** keyword is present in **Eclipse INIT** file).

Following combinations of Eclipse keywords are supported:

Netool **3-phase** mode (oil+gas+water):

SWOF+SGOF

SWOF+SLGOF

SOF3+SGFN+SWFN

SOF3+SGWFN

Netool 2-phase **oil+gas** mode:

SGOF

SLGOF

SOF2+SGFN

SOF3+SGFN   (the k_{row} column of SOF3 is ignored)

Netool 2-phase **oil+water** mode:

SWOF

SOF2+SWFN

SOF3+SWFN   (the k_{rog} column of SOF3 is ignored)

Netool 2-phase **gas+water** mode:

SGOF+SWOF   (the k_{rog} and k_{row} columns are ignored)

SGFN+SWFN

SGWFN

End-point scaling

The end-point scaling allows the user to redefine values for the connate, critical and maximum saturations in the relative permeability table. The table is linearly scaled to

preserve relative permeabilities at saturation end-points. In addition it is possible to redefine values of relative permeability at maximum phase saturations ("vertical scaling").

This is often used for modeling reservoirs with an initial depth variation of either the connate or critical saturations.

The implementation is consistent with the one in the Eclipse simulator with the following omissions:

Only two-point scaling is supported — alternative three-point scaling as defined by the **SCALERCS** keyword is not supported.

Connate gas saturation (**SGL** keyword) is not supported.

Functionality provided by **KRWR**, **KRGR**, **KRORW**, **KRORG** keywords is not supported.

For each end point input the following options are available:

No scaling (from **relperm** table) — do not scale this end point — assume that value of this end point defined by the relative permeability table is correct for this well segment.

From Nearest Blocks — the value of this end-point will be extracted from the reservoir gird from the gridblock(s) the well passes through. This option is hidden unless the data is actually available in the reservoir grid files.

Manual — enter the value of this end-point.

End-point scaling is by default not activated, i.e. No scaling (from relperm table)

10 end-points parameters are recognized as shown in **Table 11**.

Table 11: End-point Scaling Labels Used in NETool and Eclipse

NETool label	Swco	Swcr	Sw_max	Sgcr	Sg_max	Sorw	Sorg	Krw_max	Krg_max	Kro_max
Eclipse keyword	SWL	SWCR	SWU	SGCR	SGU	SOWCR	SOGCR	KRW	KRG	KRO

End-point Scaling by Manual Entry

End-points can be defined manually in **Mobilities → Relative Permeabilities → Saturations** folders on the **Segment settings/Completion** Menu. End-point scaling is not activated by default (option Unscaled).

End-point Scaling using End-Points imported from Eclipse Model

When any of the end-points are present in the Eclipse .INIT file the following appear in the NETool user interface:

The selector **Reservoir Data in the Reservoir & Well** Menu will contain the available end-point keywords. The end-point values can then be displayed in the grid.

The Mobilities Relative PerSaturations folder in the Segment settings/Completion Menu will show three additional folders for specification of end points for oil, gas and water.
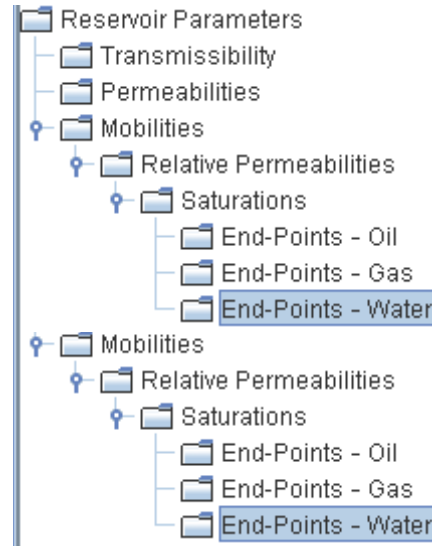
Figure 28. Reservoir Parameters and Mobilities

VFP (Vertical Flow Profile) - Lift Curves

The VFP (Vertical Flow Profile) tables contain information to calculate the relationship between tubing head and bottom hole pressures as a function of rate, water fraction, gas fraction and/or artificial lift, based on fluid flow performance in the tubing. This information is used by NETool to "convert" the tubing head pressure to bottom hole pressure or vice-versa, depending on the simulation Target and Well Type specified in the **Global Settings→General** input screen.

VFP tables are always used and required as input when the Target is set to Tubing head Pressure on the **Global Settings→General** input screen.

When Target is set to

Bottom Hole Pressure

One of the Flow rates (Reservoir, Liquid, Oil, Water or Gas)

None, Make IPR

The VFP information is not required as input.

However if the Calculate THP checkbox is ticked on **Global Settings→General**, NETool will then use VFP tables to calculate Tubing Head Pressure corresponding to the Bottom Hole Pressure, and display the Tubing Head Pressure in the Results Summary Table.

Table 12: Tube Heading Pressure

				Summary Table						
C	D	E	F	G	H	I	J	K	L	M
Phase mode	Oil rate	Gas rate	Water rate	GOR	WCUT	LGR	Q res. total	BHP	THP	Gas Lift Rate
Three phase	3562.39716	1.57522744	0	442.181871	0	0.00226151289	9216.38773	268.8953151	82.24772449	0
	[Sm3/d]	[MMSm3/d]	[Sm3/d]	[Sm3/Sm3]	[%]	[Sm3/Sm3]	[Rm3/d]	[Bar]	[Bar]	[MMSm3/d]

Also, for some of the Targets an additional THP vs BHP plot will be generated if the Calculate THP is ticked.

VFP table format

NETool supports VFP tables (**Lift curves**) in Eclipse format. Other software applications are designed for calculating tubing flow performance and generate such lift curves. Among them is e.g. Prosper which can generate lift curves and then export them in Eclipse format.

When the well is a production well a production **VFP** table (**VFPPROD** keyword) is needed; when the well is an injection well, an injection **VFP** table (**VFPINJ** keyword) is needed. In other words the **VFP** table must match the well type specified on the **General** tab of the **Global Settings Menu**.

Artificial Gas Lift option is supported by NETool and is controlled by the Gas Lift Rate input field on the **Global Settings→General** screen.

Following keywords are currently supported:

VFPPROD keyword:

FLO — flowrate variable

OIL — oil rate

LIQ — liquid rate

GAS — gas rate

WFR - water fraction variable

WOR — water oil ratio

WCT — water cut

WGR — water gas ratio

GFR - gas fraction variable

GOR — gas oil ratio

GLR — gas liquid ratio

OGR — oil gas ratio

THP — tubing head pressure

ALQ — artificial lift quantity

GRAT — gas lift rate

VFPINJ keyword:

FLO — flowrate variable

OIL — oil rate

WAT — water rate

GAS — gas rate

THP — tubing head pressure

VFP Table Units

Units system of the VFP data is designated by a keyword at the top of the file. The keywords **METRIC** or **FIELD** are accepted. This keyword MUST appear at the top of the file — you may need to manually add this keyword.

The Eclipse option to specify the units in item 8 (**VFPPROD**) of record #1 of Table 13 is supported and takes precedence over the keyword unit specification at the top of the file.

Units used by Eclipse are as follows:

Table 13: Units Used by Eclipse

Mnemonic	Quantity	METRIC unit	FIELD unit	
BHP, THP	Pressure	[Bar]	[PSI]	
FLO-OIL, FLO-LIQ, FLO-WAT	Liquid Rate	[Sm³/d]	[STB/d]	
ALQ-GRAT, FLO-GAS	Gas Rate	[Sm³/d]	[MSCF/d]	(different from NETool FIELD unit!)
WFR-WOR, WFR-WCT	Liquid/Liquid ratio (dimensionless)	[Sm³/Sm³]	[STB/STB]	
WFR-WGR, GFR-OGR	Liquid/Gas Ratio	[Sm³/Sm³]	[STB/MSCF]	(different from NETool FIELD unit!)
GFR-GOR, GFR-GLR	Gas/Liquid Ratio	[Sm³/Sm³]	[MSCF/STB]	(different from NETool FIELD unit!)

Table 14: VFP Example

VFPPROD

------------------------ Basic data, record 1------------------------------------

1 7.000E-03 'LIQ' 'WCT' 'GOR' 'THP' ' ' 'FIELD' 'BHP'   /

-----------------------5 flow values, record 2-----------------------------------

1.000E+01 3.000E+02 7.000E+02 1.000E+03   2.000E+03   /

-----------------------2 THP values, record 3---
2.0000E+02 1.000E+03   /
-----------------------1 WFR value, record 4---
0.0 /
-----------------------------3 GFR values--
1.000E+00   2.000E+00   4.000E+00   /
-------------------------------1 ALQ value--
0.0 /
------------------- Record 7 to NTHP*NWFR*NGFR*NALQ+6--------------------------
1 1 1 1   1.93199E+03 1.6585E+03 6.77031E+02 7.15261E+02 8.62436E+02/--record 7
2 1 1 1   2.7366E+03 2.73303E+03 2.7508E+03 2.77323E+03 2.90209E+03 /-- record 8
1 1 2 1   1.7747E+03 4.33303E+03 5.3808E+02 6.30323E+02 9.3909E+02 /-- record 9
2 1 2 1   2.5366E+03 2.38303E+03 2.3608E+03 2.26523E+03 2.28809E+03 / --record 10
1 1 3 1   1.6477E+03 4.41903E+02 6.9528E+02 8.81623E+02 9.3909E+02 / record 11
2 1 3 1   2.466E+03 1.78303E+03 1.808E+03 1.85153E+03 2.04484E+03 / last record
(2*1*3*1 +6= 12)

Segment Settings/Completion Menu

The **Segment Settings/Completion** menu (see Figure 29. Segment Settings and Completion sheet) is used to specify and configure completions, completion components, reservoir parameters of pressure, permeability and fluid components, skin, and some other parameters for each segment along the well.

Some of the parameters can also utilize the capability to import the values from text based log data.

This screen is accessed by clicking on the **Segment Settings/Completion** sheet at the bottom of the screen in main NETool window.
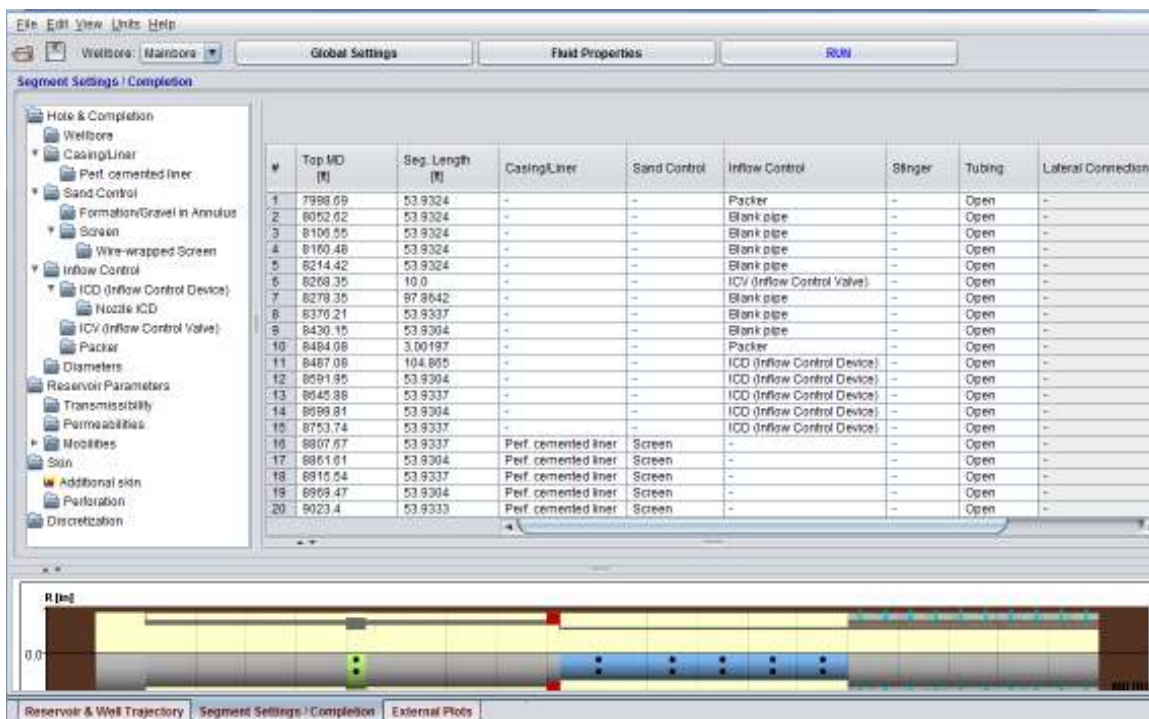
Figure 29. Segment Settings and Completion sheet

The user interface has the following features:

Editable measure depth – both top and bottom measured depth can be edited for each segment. Simply click on the appropriate segment and edit the depth in the window at the top of the screen.

 Notes

• Top depth of top segment cannot be changed from this screen – use **Heel MD** in the **Trajectory** mode on the **Reservoir&Well** screen.

• To remove or add segments, you must use **Well Segments** sheet window on the right. Bottom Depth cannot be changed beyond the end of the well path.

Smart entries – some of the input areas are smart entries. Smart entry items allow both a choice and a numerical value in the same input area. With smart entries, the choice can be made either by right clicking the input box or in the input line at the top of the screen. **Figure 30** shows the smart entry for Reservoir Pressure.

Figure 30. Smart Entry for Reservoir Pressure

Highlighting multiple input areas – multiple entries in a single column can be highlighted simultaneously. To highlight sequential entries click on the entry at one end, hold down shift and click on the entry at the other end of the sequence. Single entries can be added (or removed from the highlighted group by holding **CTRL** while clicking on that entry.

Editing multiple highlighted entries – to edit multiple highlighted entries use the editing bars at the top of the screen.

Cutting and pasting to and from MS Excel – areas of the table can be highlighted and copied and pasted to Excel. Values in Excel can also be cut and pasted into the tables on the **Segment Settings/Completion** Screen

Graying out/Non-appearance of items – input areas which are not used by NETool is grayed out (or that screen does not appear).

Completion Specification and Configuration

Completion specification in Netool is broken down into two parts:

1. Chose the completion type
2. Configure the parameters for that specific completion type.

Technical descriptions of the completions used in NETool are included in the NETool Technical Manual.

The completion is specified by choosing the **Hole & Completion** folder at the left side of the **Segment Settings/Completion** screen. Several columns appear to the right where the completion type for every layer can be specified.

The completion type is chosen in the columns corresponding to four different layers either by using the pull down menu at the top of the screen or the popup menu that appears when right clicking the mouse.

Each completion type has its own specific input section that becomes active as Hole & Completion subfolders when that completion type is chosen as part of the well.

Specific Completions Types

This section discusses the input parameters for the specific completions. A more technical discussion is included in the NETool Technical Manual.

The roughness in the tubing and annulus affects the pressure drop. Formation and Pipe Roughness are available for each completion layer.

Open Hole Completion

The only information required by NETool for an open hole completion is the Wellbore diameter and Roughness – no additional sub folder is required or available for the open hole completion.

Damaged Zone skin or manually entered skin is available for the open hole completion type and discussed in further detail in the *NETool Technical Manual*.

ICV device

Inflow Control Valve device is a completion type which has a valve in the middle. That valve connects annular and tubing spaces. In combination with blank pipes and packers at each side it is usually used for zone isolation. There are different pressure drop models and several parameters to control flow through it. This type of completion is described in more detail in the *NETool Technical Manual*.

The ICV completion is run with an open annulus and the option for a Collapsed Annulus is available.

Required input parameters for ICV are:

Tubing ID – inner diameter of the isolation tubing used

Tubing OD – outer diameter of the isolation tubing used

Pressure drop calc. – defines which pressure drop model to use for the valve

Parameters related to each pressure drop model.

Slotted Liner

Slotted Liner is a liner with slots cut to allow flow. This liner type is not cemented. A Technical Description is included in the *NETool Technical Manual*.

Required input items for Slotted liner are:

Casing/Liner ID – inner diameter of the slotted liner

Casing/Liner OD – outer diameter of the slotted liner

Slots Density – how many slots per ft (or per meter)

Slot Length – length of the slots

Slot Width – width of the slots

Perforated Cemented Liner

The perforated cemented liner input consists of two parts:

Description of the liner and perforations, and

Choice of skin model and skin model input parameters – the perforation skin input items are discussed in the **Skin** section.

Casing/Liner and Perforation Description

The liner/casing and perforations are configured under the **Perf. cemented liner** folder under **NETool window → Hole and Completion** sheet.

The required input items for describing the Casing/Liner and Perforations are:

Casing/Liner ID – inner diameter of the casing/liner

Casing/Liner OD – outer diameter of the casing/liner

Hole Diameter – Entry diameter of the perforation.

Shot Density – number of shots per foot (or meter)

Length of Perforation – penetration length of the perforation as measured from the outer diameter of the casing/liner.

The last three parameters may also be edited under skin folder.

ICD Completions

ICDs or Inflow Control Devices are completion components that create pressure drops between annulus and tubing. These devices are primarily introduced to control the zonal flow contributions to obtain optimized reservoir performance.

ICDs are often associated with horizontal wells and in wells across multiple zones to avoid premature water and/or gas breakthrough.

NETool includes the following ICD models:

- ICD - Nozzle ICD – an ICD implementation for ICS's using nozzles to create the pressure drop.
- ICD - Tubular ICD – an ICD implementation for ICS's using parallel tubes, assuming equal dimensions for each tube to create the pressure drop.
- ICD - Channel ICD – an ICD implementation for ICS's using parallel channels, assuming equal dimensions for each channel to create the pressure drop.
- Halliburton EquiFlow (adjustable) ICD – models provided by Halliburton for EquiFlow ICD completion.
- Baker Spiral ICD, Troll – a pressure drop model for the Baker Spiral ICD based on laboratory tests with the Troll fluids. This model is only valid for Troll fluids. Troll is the largest gas field with a thin oil column located offshore Norway.
- Baker Spiral ICD, General – a general implementation of the Baker Spiral ICD model valid for a variety of fluids.
- Baker Adjustable ICD – a model for the Baker Adjustable ICD.
- Autonomous ICD – a generic equation for modeling autonomous ICD or valve.

All ICD completion types have annulus and tubing node layers. They can be run with open annulus or collapsed annulus. The collapsed annulus facility is available under Hole and Completion to have Collapsed Annulus.

Nozzle ICD

The Nozzle ICD can be used for any ICD type device. Pressure drop is simulated using a nozzle calculation. This option is appropriate for example for Reslink ICD's which use nozzles to generate the pressure drop.

The required input parameters for the Nozzle ICD are

Tubing OD – OD of the ICD device

Tubing ID – ID of the ICD device

Joint Length – the spacing between the pressure inducing devices. Normally they are placed one per joint.

ICD Nozzle Diameter

Flow coefficient

Number of parallel Nozzles per joint length - up to 30 individual nozzles

Further discussion of the Nozzle ICD is included the NETool Technical Manual.

Tubular ICD

For the Tubular ICD, pressure drop is simulated using parallel tubes, assuming equal dimensions for each tube calculation.

The required input parameters for the Tubular ICD are

Tubing OD – OD of the ICD device

Tubing ID – ID of the ICD device

Joint Length – the spacing between the pressure inducing devices. Normally they are placed one per joint.

Tube diameter

Tube length

Number of parallel tubes per joint

Minor Loss Coefficient

Further discussion of the Tubular ICD is included in the NETool Technical Manual.

Channel ICD

For the Channel ICD, pressure drop is simulated using channels tubes, assuming equal dimensions for each channel.

The required input parameters for the Channel ICD are

Tubing OD – OD of the ICD device

Tubing ID – ID of the ICD device

Joint Length – the spacing between the pressure inducing devices. Normally they are placed one per joint.

Channel hydraulic diameter from (manual or calculated)

Channel hydraulic diameter

Channel cross-section

Channel length

Number of parallel channels per joint

Minor Loss Coefficient

Further discussion of the Channel ICD is included the NETool Technical Manual.

Halliburton EquiFlow ICD

This model is based on nozzle and frictional pressure drop models to calculate pressure loss through flow tubes within pressure drop chamber of the EquiFlow ICD. The main input parameters are:

Tubing OD – OD of the ICD device

Tubing ID – ID of the ICD device

Joint Length – the spacing between the pressure inducing devices. Normally they are placed one per joint.

EquiFlow ICD type- allows the user to select one of the predefined settings or manually specify the three tube design parameters (length, diameter, quantity).

The predefined settings notation, like 20 x 100, means the pressure drop for a certain rate across the EquiFlow ICD. The pressure drop is given in psi and the rate in bbl/d for a nominal fluid with viscosity 1 cp and specific gravity of 1. The example 20 x 100 means 20 psi pressure drop at a 100 bbl/d rate.

The **manual** option facilitates flexible configuration of the pressure drop chamber in the EquiFlow ICD. The user can specify length and diameter of the tubes, as well as the number of tubes in parallel.

Tube length (customized) - Length of the tube in the EquiFlow ICD's pressure drop chamber

Tube Diameter(customized) - Diameter of the tube in the EquiFlow ICD's pressure drop chamber

Number of tubes(customized) - Number of parallel tubes in the EquiFlow ICD's pressure drop chamber

Halliburton EquiFlow Adjustable ICD. The pressure drop is created by six tubes of different diameters. The completion has a threaded removable sleeve to allow access to the flow tubes. The flow profile is changed by closing or opening a pre-determined set of tubes and is carried out at the surface.

Baker Spiral ICD, Troll

The pressure drop correlation for the Baker Spiral ICD, Troll is based on tests performed with Troll (offshore Norway) reservoir fluids and this specific item of equipment. The results of this model do not use the PVT data of the fluids supplied to NETool. This model is only valid for the Troll field.

Please contact Baker Oil Tools for further information.

The required input parameters for this model are:

Tubing OD – OD of the ICD device

Tubing ID – ID of the ICD device

- Spiral ICD Type – the input choices are for 5 settings of this device with the number representing the pressure drop in bar at nominal rate with nominal fluid, which are associated with the commonly used settings for Baker Spiral ICD.

Baker Spiral ICD, General

The Baker Spiral ICD, General is a general implementation of the Baker Spiral ICD – unlike the Baker Spiral ICD, Troll, this completion type is affected by the PVT properties of the fluids.

Please contact Baker for further information.

The required input parameters for this model are:

Tubing OD – OD of the ICD device

Tubing ID – ID of the ICD device

Spiral ICD Type – This is a smart input item, it can be set either to one of the predefined settings of the ICD or manually using the Coefficients A and B. The 5 numeric input choices are settings of this device with the number representing the pressure drop in bar at nominal rate with nominal fluid, which are associated with the commonly used settings for Baker Spiral ICD

Manual definition – coefficients A and B are used to determine the pressure drop. This is an advanced option and would normally only be used for the detailed design and after conversations with the vendor.

Baker Adjustable ICD

An adjustable ICD from Baker which can have 2 to 6 ports open.

The required input parameters for this model are:

Tubing ID – ID of the ICD device

Tubing OD – OD of the ICD device

- Adjustable ICD Type – Number of open ports on the ICD.

Autonomous ICD.

This model allows simulating inflow control devices that use "autonomous" concepts. These include Statoil's Rate Controlled Production (RCP) and Halliburton's Equiflow AICD. These devices are characterized by a high dependency of pressure drop on flow rate: $dP \sim q^n$ where n can be in a range [3;5].

Screens.

There are three types of screens available: Generic, Perforated pipe and wire-wrapped.

Screens usually create an insignificant pressure drop between annulus and tubing. All types can have sand pack or filter cake.

Generic.

This type of screen represents a completion with a very large flow area and therefore zero pressure drop across the completion. There are no input parameters.

Perforated pipe.

Represents a pipe with round holes. Input parameters are "Perforation diameter" and "Perforation density". Orifice equation is used to calculate pressure drop.

Wire-Wrapped Screens

This completion uses both the annulus and tubing node layers, hence the option for Collapsed Annulus is available.

The wire-wrapped screens completion includes the option to simulate a filter cake built up on the outside of the screen – this option is available under an additional folder Filter cake/Annulus fill.

Wire-wrapped Screen input data

Figure 31 illustrates the various components in wire-wrapped screens.

The following are the required input items for wire-wrapped screens:

- Screen base pipe ID – Internal Diameter of the Screen Base Pipe. The same as the ID of the screen.

- Screen base pipe OD – **DO NOT USE SCREEN OD HERE**. Screen OD is calculated by NETool. It is the OD of the screen base pipe that should be entered. NETool calculates the Screen OD = Screen Base Pipe Outer Diameter + 2*Height of Flow Channel + 2* Wire Thickness

- Perforation diameter – diameter of the perforations in the screen base pipe.

- Joint Length – length of each screen joint.

- Perf. Density- Perforations per length of unit.

- Width of flow Channel – the width of the space radically between the longitudinal wires of the screen. "w" in the **Figure 31**.

- Thickness of flow channel walls – width of the longitudinal wires (those first laid on the base pipe). "s" in **Figure 31**.

- Height of flow channel – thickness of the longitudinal wires (those first laid on the base pipe). "d" in **Figure 31**.

- Wire Width – width of the outer wires of the screen (those that hold back the sand). "W_w" in **Figure 31**.

- Wire Thickness – thickness of the outer wires of the screen. "h" in **Figure 31**.

- Spacing between wires – the gap between the outer wires of the screen. "y" in **Figure 31**.

Pre-Perforated Base Pipe – tubular pipe with a holes drilled in it

Inner (Longitudinal) Wire – first on welded onto the basepipe, runs along axis of pipe

Outer (Radial) Wire – wound around the pipe, this is the on that holds back the gravel
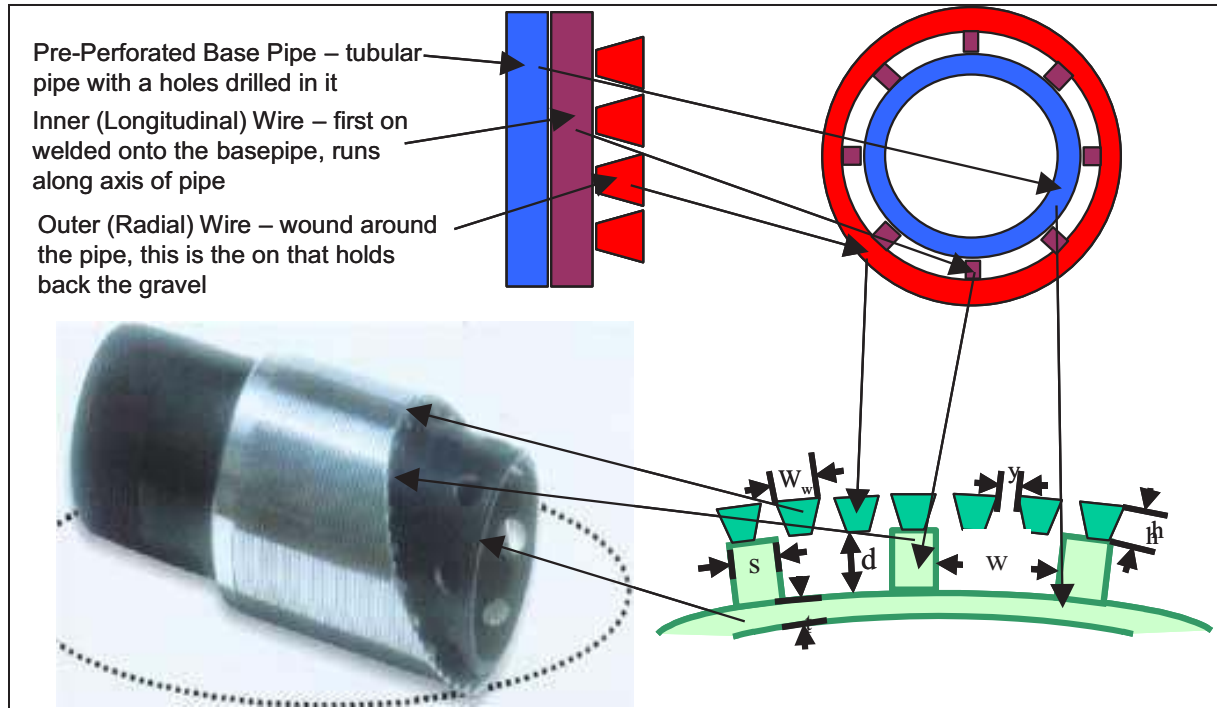
Figure 31. Wire-wrapped Screens Illustration

Filter Cake/Annulus Fill Input

The wire-wrapped screen model includes an option to model a build up of porous media (filter cake) on the outside of the screen and the effect of this filter cake on the productivity of the well.

The filter cake modeling can be used to model gravel packs also – without slip in the annulus – this modeling is identical to checking the box collapsed annulus on the under the Hole and Completion folder. For modeling of gravel pack which includes the slip between gas and fluids please use the gravel pack option.

The following are the input items under Filter cake/Annulus fill:

Sand Pack Thickness – thickness of the filter cake built up on the screens.

If a value of thickness greater than or equal to than annulus thickness is entered, the annulus is considered filled.

For compatibility with older versions, entering a negative number also simulates the annulus filled with porous media. In the current version please use collapsed annulus rather than a negative number.

Sand pack permeability – this is a smart input item. There are two choices under this item:

Manual – enter the permeability of the sand pack/filter cake here.

Use Sand Properties – permeability is calculated base on the following input parameters:

Sand pack porosity – porosity of the sand pack

Sand particle shape factor – particle shape factor.

Sand grain diameter – average diameter of the sand grains in the filter cake.

Gravel Pack

The Gravel Pack completion is used to simulate an open hole gravel pack. Wire-wrapped screens are used to contain a gravel filled annulus against an open hole.

In versions prior to 5000.0.1.0 NETool had "Gravel pack" and "Cased Hole Gravel Pack" completions. From v5000.0.1.0 they are replaced with "Screen+Formation/Gravel in Annulus". To model Gravel pack:

1. Chose desired screen type on "Sand Control" layer.
2. Select "Formation/Gravel in Annulus"
3. For Cased Hole Gravel Pack also select "Perf. Cemented liner" on "Casing/Liner" layer.

The most common screen to be used with Gravel Pack is Wire-wrapped screen. The Wire-wrapped screen folder is used to input the wire-wrapped screen portion of the open hole gravel pack system. For more information on inputting the wire-wrapped screen information, please see the previous section. Filter cake/annulus fill of the wire-wrapped screen section is not used for the gravel pack model.

The required inputs for the gravel pack model (as illustrated in **Figure 32**) in addition to those under the wire-wrapped screen folder are as follows:

Annular Permeability – Permeability of the Gravel Pack Sand.

Phase slip - may be checked/unchecked

Corey Index – used in the slip (simple two phase relative permeability relationship) for slip between gas and Liquids as they flow along the well through the gravel. Acceptable values are equal to or greater than one (1). Two (2) is commonly used.

Phase Slip Correlation – determines to use 2-phase or 3-phase slip model.

Annular Forchheimer beta - for modeling non-Darcy flow effects.

For Cased Hole Gravel Pack under the "Perf. Cemented liner" the following input is available:

Gravel filled perf. length –this input area allows simulations based on how the perforation tunnel has been filled with gravel. Appropriate values range from zero – perforation tunnel completely void to Length of perforations as entered under the Perf. cemented liner sub-folder.
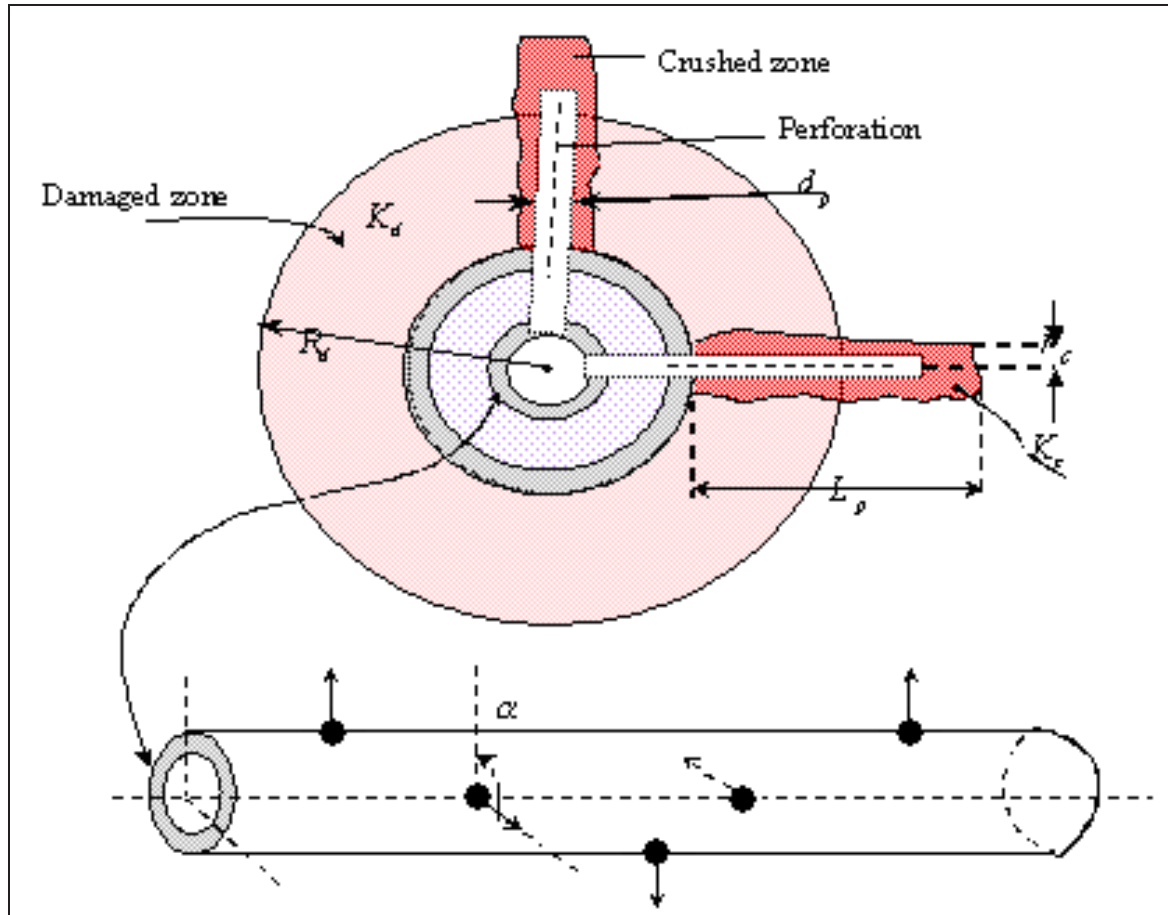
Figure 32. Gravel Pack Illustration

Blank Pipe

The blank pipe completion is used when a section of blank pipe is run to isolate the tubing from the annulus along a section of the well.

For the blank pipe completion type the inner and outer diameter of the blank pipe section are required as input:

Tubing/Casing ID – inner diameter of the blank pipe

Tubing/Casing OD – outer diameter of the blank pipe

Cemented Blank Pipe

The cemented blank pipe completion type is used when no annulus is present and there is not any flow from the reservoir. If it is desirable to simulate a portion of the production tubing, the cemented blank pipe completion type should be used.

The only required input item for cemented blank pipe type completion is the inner diameter of the casing/liner/tubing.

For stability of the NETool solver, the top segment in the well is always cemented blank pipe. If this is not a part of your completion scheme please just make the segment very short (like 3ft or 1m) and it won't affect your results.

Packer

This is a short device which blocks annular space. In NETool model, it is equivalent to Cemented Blank Pipe. However, for user convenience it is included as a separate completion type.

Packer can be sealing – if there's no leak between the packer and wellbore walls and non-sealing (there's a gap fluids can go through). Non-sealing packer is plotted in yellow instead of red on completion plot.

Inline Choke/Plug

This completion type can restrict or completely block flow within the tubing. It has the same pressure drop models to choose from as the ICV completion type.

There is also an option to allow for annulus flow around the restriction/plug.

Reservoir Parameters

The Reservoir parameters folder is used to define reservoir properties for calculations of flow performance between reservoir and wellbore. The modeling is based on PI models.

The flow is calculated per each segment. The basic PI formula is $Q_{phase} = \lambda_{phase} \cdot T \cdot \Delta P$, where λ is fluid mobility, T is transmissibility, ΔP is pressure difference between reservoir pressure (input) and annulus pressure (calculated by NETool).

The following input columns are available under the top level Reservoir parameters folder:

Table 15: Top Level Reservoir Parameters Folder

Reservoir Parameters Folder	
Column	Description
Connected to reservoir	Defines if a segment is connected to reservoir. Checked means it is connected to reservoir. Unchecked has same effect as zero permeability or non-perforated liner.
Reservoir Pressure	Three input options are available for data input:
	Manual. Reservoir pressure values are entered manually here or imported from a log. The appropriate reservoir pressure when entered manually is dependent on the settings for the PI model – for Horizontal wells, with Steady State Model enter Pressure at the boundary of the reservoir and average reservoir pressure for the Semi-Steady State model.
	From nearest blocks. Reservoir pressure values are taken from the reservoir grid from the blocks the well passes through.
	From grid upscaling – permeability value is averaged from the

	grid data
Rs	Rs, Solution Gas Oil Ratio, is the amount of dissolved gas in oil at the current reservoir condition.
	This input column will only appear if i) an Eclipse type PVT table includes the PVTO keyword with multiple Rs values or if ii) NETool format PVT table is given with multiple bubble points for oil. Otherwise, the value of Rs is well defined in the PVT data having a unique pressure relation and this column is not necessary and is hidden.
	Three options are available for data input:
	Manual entry of the Rs value.
	Saturated. Assumes that the oil is saturated and the Rs is a function of Reservoir Pressure
	From nearest blocks. Rs values are populated from the gridblock(s) the well passes through. This requires the imported Eclipse restart file includes Rs values.
Rv	Rv, Vaporized Condensate Ratio, is the amount of condensate liquid that is vaporized in the gas at the current reservoir conditions.
	This input column will only appear if) an Eclipse type PVT table includes the PVTG keyword with multiple Rv values or if ii) the NETool format PVT table is given with vaporized condensate and multiple dew points. Otherwise, the value of Rs is well defined in the PVT data having a unique pressure relation and this column is not necessary and is hidden.
	Three options are available for data input:
	Manual entry of the Rv value.
	Saturated. Assumes that the gas is saturated with vaporized oil and the Rv is a function of Reservoir Pressure
	From nearest blocks. Rv values are populated from the gridblock(s) the well passes through. This requires the imported Eclipse restart file to include Rv values.

Transmissibility

Transmissibility is a part of the PI modeling.

The transmissibility equation can be described as $T = A / (B + S)$, where S is the skin value.

Table 16: Transmissibility Subfolder

Transmissibility Subfolder

Column	Description
Transmissibility	Three options are available for data input:
	From PI model. Transmissibility is calculated by NETool based on upscaled permeabilities, skin and other PI model parameters.
	Manual. Transmissibility is entered manually or as a log. NETool skin model can't be applied in this case since user entered T must already include all skin effects.
	Manual T_A & T_B. Same as previous but allows using NETool skin model. Transmissibility (T) is calculated as follows: T = T_A / (T_B + S)
Transmissibility A	Enter the A factor in the Transmissibility equation in the form: Transmissibility = A / (B + S)
Transmissibility B	Enter the B factor in the Transmissibility equation in the form: Transmissibility = A / (B + S)
Transmissibility multiplier	The transmissibility is multiplied by this value. This capability has been included to ease history matching and quickly run sensitivities.

Skin factor also affects the flow from the reservoir (as calculated by the PI model) and details are discussed in Skin section

PI model

The folder PI model will appear only if the Global Settings->Inflow->Set PI model for each segment is checked.

Permeabilities

Table 17: Permeabilities Subfolder

Permeabilities Subfolder	
Column	Description
Kh	Horizontal permeability. 3 options are available for data input:
	Manual – permeability is entered manually or as a log.
	Use grid upscaling – permeability value is upscaled from the grid data
	From nearest blocks – permeability is averaged from the grid blocks well is passing through
Kv/Kh	Ratio of Vertical to Horizontal Permeability used to define the vertical permeability. 3 input options are available:
	Manual – ratio is entered manually or as a log.

	Use grid upscaling – ratio is upscaled from the grid data *From nearest blocks* – ratio is averaged from the grid blocks well is passing through
Bedding Dip East	is the dip of the reservoir in the eastward direction, angle down from horizontal towards the east. Bedding dip East and Bedding dip North are used to determine the dip angle of the reservoir such that the angle of the well relative to the dip angle of the reservoir can be calculated. The angle of the well relative to the bedding plane of the reservoir is then used to determine which PI calculation is used (vertical well, deviated well, or horizontal well).
Bedding Dip North	is the dip of the reservoir in the north direction, angle down from horizontal towards the east. See more description above.
Upscaling radius	Permeability values of grid blocks inside this radius (measured from the wellbore) are included in upscaling.
Upscaling MD step	Specifies axial step used in upscaling (dZ in cylindrical coordinate system)
N angular steps	Specifies number of angular steps used in upscaling ($360°/d\theta$ in cylindrical coordinate system).

About Upscaling:

NETool uses a proprietary anisotropic upscaling technique which access scattered permeability data from the Eclipse grid and averages it for use in the NETool analytical PI equations.
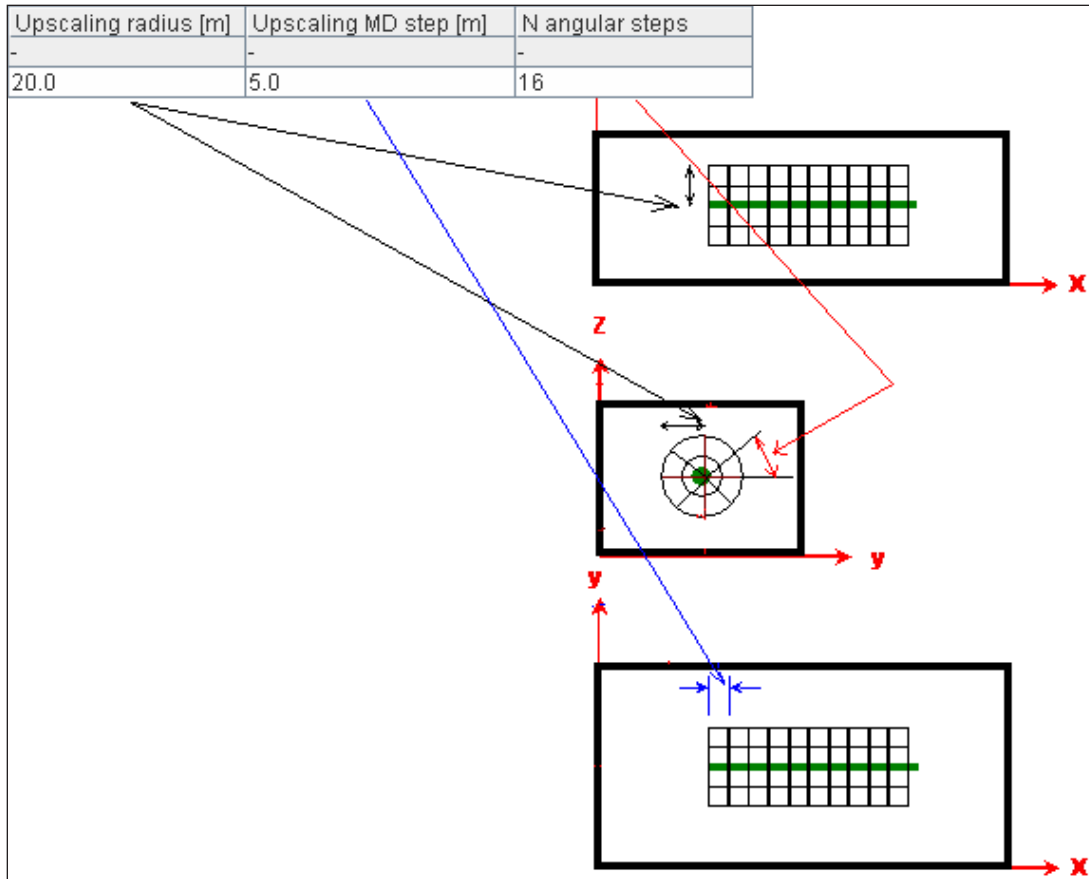
Upscaling radius [m]	Upscaling MD step [m]	N angular steps
-	-	-
20.0	5.0	16

Figure 33. Upscaling of Parameters

Mobility

Phase mobilties control how the fluid properties and fluid-rock interactions affect production or injection.

Mobilities are used in the basic equation for flow of each phase:

Phase influx = phase mobility × transmissibility × (reservoir_pressure – annulus_pressure).

$Q_{phase} = \lambda_{phase} \cdot T \cdot \Delta P$

The phase mobility term (λ) is:

phase_mobility = phase_relative_permeability/phase_viscosity.

$\lambda_{phase} = k_{r\ phase}/\mu_{phase}$

Mobilities are used in the basic equation for flow of each phase:

Phase influx = phase mobility × transmissibility × (reservoir_pressure – annulus_pressure).

$Q_{phase} = \lambda_{phase} \cdot T \cdot \Delta P$.

Injection Mobility

Netool has the following two methods for determining mobility of the injected fluid in the formation:

Long Term.

This model assumes injector has been operational for a long time and formation is filled up with the injected component. Mobility equals to the mobility of the component being injected:

Gas Injector	Water Injector
$\lambda_g \quad = \quad 1/\mu_g$ $Q_g = \lambda_g \cdot T \cdot \Delta P$	$\lambda_w \quad = \quad 1/\mu_w$ $Q_w = \lambda_w \cdot T \cdot \Delta P$

Short Term.

This model assumes injection process is starting and formation has its original fluids in place — the injected phase has not penetrated the near-wellbore area to any great extent. The injected fluid is displacing all movable phases according to their calculated mobility. The effective mobility equals to the sum of mobilities of phase components:

Gas Injector	Water Injector
$\lambda_o = k_{ro}/\mu_o \quad \lambda_g = k_{ro}/\mu_g$	$\lambda_w = k_{ro}/\mu_w$
$Q_g = (\lambda_o + \lambda_g + \lambda_w) \cdot T \cdot \Delta P$	$Q_w = (\lambda_o + \lambda_g + \lambda_w) \cdot T \cdot \Delta P$

Skin

NETool includes skin in the PI model. Skin can i) be entered manually or ii) be automatically determined.

Input parameters for skin calculation:

Table 18: Skin Calculation Parameters

Skin type	Description
Additional skin	This is skin in addition to perforation skin, Non-Darcy skin and near well bore damage skin. Additional skin may be entered per segment or be imported as log.
Perforation skin	If perforation skin is checked under Skin folder the following parameters may be defined in Perforation subfolder Perforation Skin Model – this item is used to choose between the two perforation skin models. This item is not active unless the completion type is Perforated liner or Cased hole gravel pack. Choices include: *Karakas, Tariq* – a model based on **SPE 18247** numerical simulations of vertical well which has been adopted for horizontal and deviated wells.

	Furui, Zhu and Hill – a model developed for horizontal wells; which considers the effect of perforation orientation in anisotropic media. This model was published in **SPE 77363** and is adapted in NETool for use in deviated and vertical wells.
	The two perforation skin models are discussed in more detail in the Perforated Liner Section of the NETool Technical Manual.
	K_c/K – Crushed zone permeability / original formation permeability ratio
	Radius of crushed zone – r_c – radius of the zone around the perforation tunnel where the permeability has been damaged by the perforation process.
	Phasing angle – α – phase angle between successive shots in the perforation gun. Due to the differences in the methods that were used to derive the two skin models, they have different ranges of validity with respect to perforation phase angle Valid values are: Karakas, Tariq model: 0, and 45-360 degrees Furui, Zhu, Hill: 90, 120, 180, 360 (0) degrees
	Perforation Orientation – this input value is only used with Furui, Zhu and Hill Perforation skin Model and is grayed out when the Karakas and Tariq Perforation Skin Model is chosen. This is the angle of the perforations from the horizontal plane when considered for a horizontal well. Commonly high side or low side perforating is used when oriented perforating, both are designated by an input value of 90.
	Shots density – number of perforations per length unit
	Length of perforations – length measured from OD of casing/liner
	Hole diameter – entry diameter of perforations
	The last three parameters may also be edited under Perforated Cemented Liner and Cased Hole Gravel Pack.
Non-Darcy skin	If Non-Darcy skin is checked the following options are available:
	D factor – D is non-Darcy flow coefficient and may be defined as:
	Manual – manually entered D factor when known, e.g. from well tests
	From Forchheimer beta – the non-Darcy term is calculated from rock properties – Forcheimer's β-factors
	Rock Forchheimer beta
	Damaged zone Forchheimer beta
	Additional details are provided in the NETool Technical Manual.
Near wellbore damage	If this parameter is checked the following may be defined:
	K_d/K – ratio of damaged zone permeability to native permeability. Used for all automatic skin calculations.
	Radius of Damaged Zone – Radius of the damaged zone, measured from

	the center of the Wellbore
	A schematic of the damaged zone is included in the section Open Hole in the NETool Technical Manual. The perforation skin calculation considers the damaged zone and the perforations penetration into or through the damaged zone.
	The following other input items also affect the calculation of the perforation skin:
	Under the Perf. cemented liner folder: shot density, hole diameter and perforation length and
	Under this Skin folder: K_d/K and Radius of Damaged Zone.
	As mentioned under the description of the Cased Hole Gravel Pack completion in the NETool Technical Manual, pressure drop through the perforation tunnel is not included in the skin as calculated for the PI equation. Therefore, Gravel filled perf. length under the Cased Hole Gravel Pack folder does not affect the skin as used in the PI equation and displayed in the NETool results. The pressure drop through the gravel filled (or void) perforated tunnel is included in the pressure drop from annulus to tubing.
	Please note that described skin models give approximate estimate for skin factor and are good only for commonly used parameter values (like perforation length, density, permeability reduction, etc.). For a rigorous skin calculation, 3D finite element simulation is required in each case.
	The perforation skin models are discussed in more detail in the NETool Technical Manual.

Discretization

This folder controls spatial discretization of the solver.

No of Intermediate Nodes NETool uses intermediate nodes between the nodes defined by the user. By default, the number of intermediate nodes is set to 10. In some cases where a large number of nodes are defined, the number of intermediate nodes can be reduced to speed up simulation time. In other cases where a very small number of nodes are defined by the user a larger number of intermediate nodes may be used. Sometimes convergence for difficult cases can be improved by changing the number of intermediate nodes. When error messages recommend changing the number of intermediate nodes, the user should also check the input data, especially consistent PVT data, for self consistency because NETool cannot solve physically impossible cases.

Spacing of Intermediate Nodes This is an alternative option to define intermediate nodes. The distance between every two adjacent nodes is specified. The number of intermediate nodes is calculated as Segment length / Spacing of intermediate nodes

rounded upwards to the nearest whole number. The nodes are then evenly distributed in the relevant segment.

Log Entry Capabilities

NETool allows for easy import of log data for some parameters. The log screens also allow for interactive manipulation of the parameter values per segment. The screens also facilitate manipulation of nodes (move, delete, create).

Log data can be pasted into NETool or imported from file as two columns format:

      Measured Depth Log Value

These log values are automatically averaged (arithmetic averaging) by NETool for each individual segment.

The following parameters can utilize log input capabilities:

- Wellbore Diameter – generally taken from the open hole caliper log

- Reservoir Pressure Values – sometimes from logs and other times from another source and entered as a log for convenience

- Transmissibility – this value would normally be calculated by another software tool and the log import in the case is a convenience issue.

- Transmissibility A - this value would normally be calculated by another software tool and the log import in the case is a convenience issue.

- K_h (Horizontal Permeability) – usually derived from log data based on field specific correlations between log data and core tests.

- K_v/K_h (ratio of Vertical to Horizontal Permeability) – occasionally field specific correlations are made, this particular input is more commonly used for convenience

- Skin – usually derived by some external method – included for convenience.

- Mobility – three different options available.

- Oil/Gas/Water Mobility – this would normally be data generated by a reservoir simulator and the log import functionality provides a convenient method of moving this data to NETool.

- Oil/Gas/Water Relative permeability – processed saturations derived from downhole log measurements. In addition relative permeability must be provided in order to calculate the mobility.

- Oil/Water Flowing Fractions – usually taken from production log data.

To access the log entry features of NETool, the particular input item must be set to manual (except in the case of wellbore diameter for which there is not the option to extract from the reservoir grid).
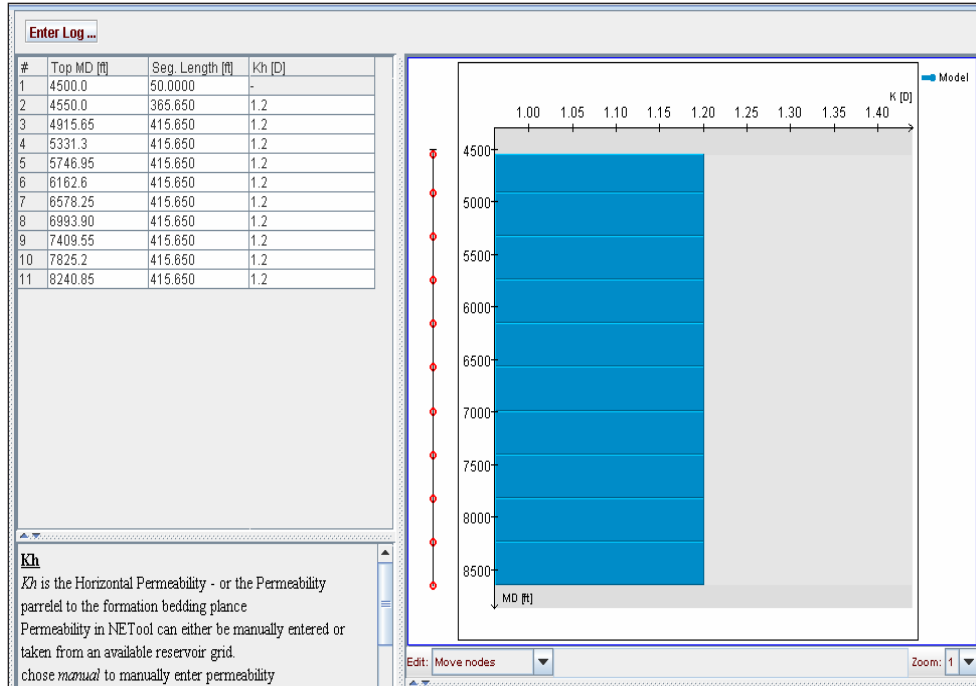
Figure 34. Kh Screen Prior to Entering the Log Values

To enter the log, click on the Enter log box at the top of the view. This will bring up a text entry windows.

When the window is initially opened, it will have three lines of text describing the two data that NETool is expecting and the units for that data.

<u>Log input and units</u>

The units are dependent upon the current units active when the **Enter Log** button is pressed. In this case we have ft for measured depth, if your data is in meters for measured depth, just cancel the window, change the units in NETool to metric and click **Edit Log** again. The units will then appear in metric.

You can also locally change the units on this screen and those will affect the units expected for the log. For example if the units for permeability in your data are mD rather than D, highlight one of the permeability values and change the units to mD. When the log window is re-opened, NETool will display the units of permeability as mD and hence expects input in mD.

The log input window accepts text input. The most common method of generating this text input is to open the data from its source within your organization in a spreadsheet (MS Excel for example). Then use Excel to generate to adjacent columns of values. Then highlight these two columns of values and copy/paste them into the NETool input window.

After **OK** is clicked, NETool has "saved the log" in NETool internal units. If the units in NETool are changed and the **Enter Log** button is clicked to view or edit the log data, the log will be presented in the current NETool units, as shown in **Figure 35**.

The log will then be displayed on the graph of value (in this case K_h vs. measured depth). The values assigned to each segment will be changed to the average log value for each segment.
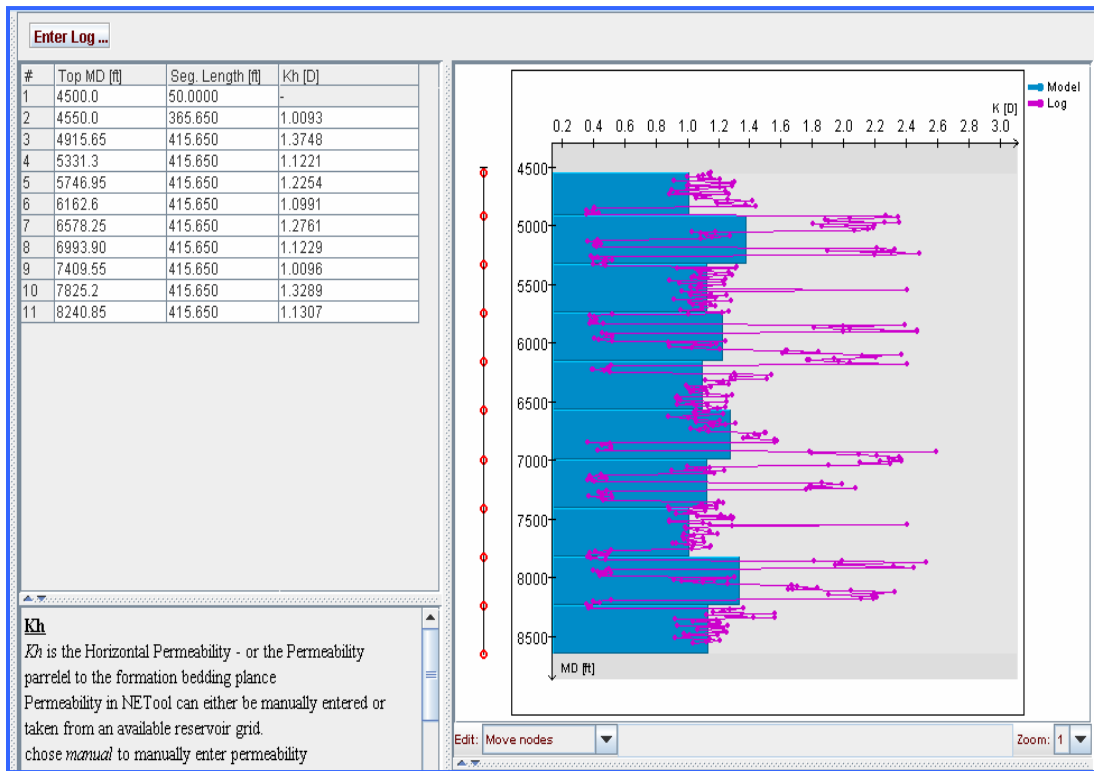

Figure 35. Kh Screen With Log Values

Interactivity with log data

The log graphic screen offers the following interactive options for working with the log data such that the averages taken from the log are transferred to NETool in an accurate manner. An example is illustrated in **Figure 35**.

Change the value of the average value for segment

User may prefer to adjust the average that NETool calculates, there are two ways of doing that:

Graphically – to change the average graphically, simply use the mouse to move the average bar up or down to an appropriate value.

Numerically – The average can also be changed by typing in a new value in the table – in this case under K_h

Move nodes – one can also move the nodes to improve the discretization to better match the log.

Graphically – The nodes can be moved graphically by using the node indicators on the graph, for which the graph must be in **Edit: Move Nodes** mode on the bottom of the screen.

Numerically – the nodes can also be moved numerically by typing in a new top measured depth in the table.

Figure 36 shows the example with the nodes moved to provide discretization that matches the layering apparent in the log better.

To achieve a re-average of the log after moving the segments, click on **enter log** and then **OK** and the log data imported will be re-averaged.

Add/remove nodes – nodes can be added or removed graphically by changing the Edit menu below the plot to **Add/remove** nodes. Nodes are added by clicking on the **node line**. Nodes are deleted by right clicking on a node and selecting delete.

In the example above, the default number of nodes was insufficient for the layering of the reservoir. Additional nodes were added at the layer boundaries to create the final discretization.

It should be noted that moving the nodes for the log moves the nodes in ALL of NETool and any completion components that have been assigned are also moved. It is recommended to do work with the node placement in the log import prior to entering the completion.

Zooming – one can zoom in on the log graphics screen by changing the zoom level on the bottom right of the graph screen.
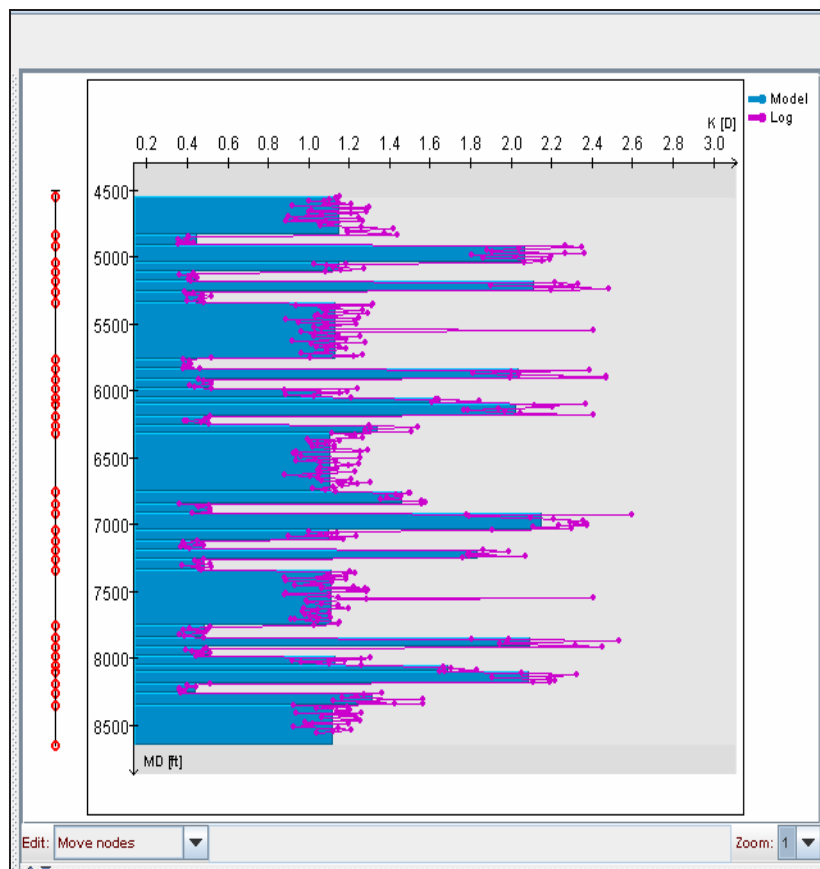

Figure 36. Interactive Work With Log Data

Running NETool Simulations

Before running a simulation, make sure your well case contains consistent and physically correct data.

Select the **Run** button to launch NETool simulations. Normally the simulations only take a few seconds unless heavy upscaling is taking place.

Results are reported in the **Plots Viewer**.

Calculations can be interrupted by clicking the **STOP** button. Please note that calculations may not be interrupted immediately, you may need to wait for several seconds before NETool can accept user input again.

While NETool is running, some text will be displayed in the black MS-DOS command window.

If you have any difficulties with NETool simulations, the text reported in this window can also be helpful.

Warnings and errors may also appear in the **Plots Viewer**.

Simulation Results

The NETool simulation results presented in **Plots Viewer** window (as illustrated in **Figure 37**), are both graphically illustrated as plots and as a table of numeric values that can be easily cut and pasted into Excel for additional calculations/plotting. The numeric values are included on the Details screen in NETool.

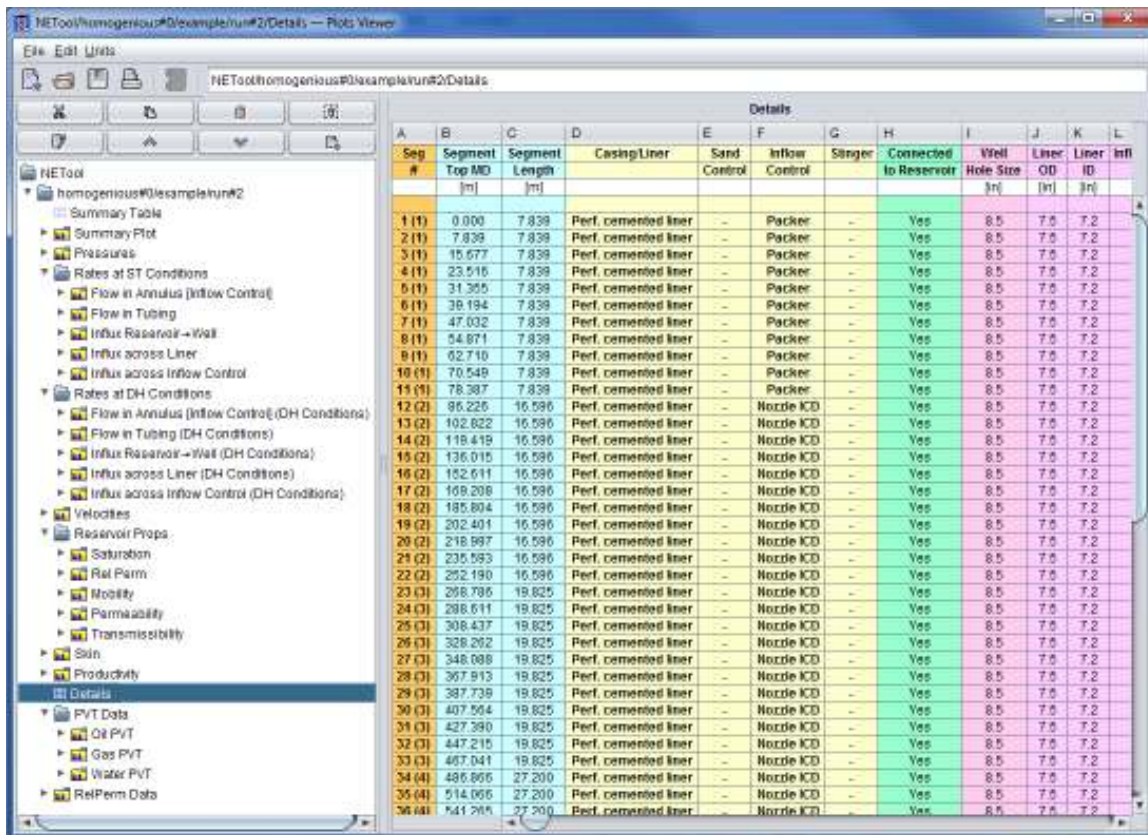The data is organized in a tree view.

Meaning of the folders:

- **Folder**. Results from a single simulation are organized in a **folder**.

- Group of plots in a single screen. If you wish to see plots individually, click on the handle ☛ to expand.

- A single plot

- Tabulated data, for example **Details** screen.

n - Text, for example, **Warnings** or **Nick name** to run.

Figure 37. Simulation Results Display

Output Units

In general, it is best to run NETool with your preferred unit system (FIELD or METRIC). The plots on the output can be viewed in either units system by choosing the units system on the output menus. However, the Outcome and Details pages **are available ONLY IN THE UNIT DEFINED FOR THE SIMULATION**.

Saving, Copying and Printing Plots

To save all results use

> **File➔Save As**

This will save all the data as a ***.ntlplot** file

To save a part of the output, for example to save an output from a particular simulation run or to save an individual plot, click on an appropriate item in the "tree view" on the left hand side of the screen and then choose

> **File➔Save Selection As**

This will save a ***.ntlplot** file with only the data selected. Use **Ctrl** key to combine several items in the selection.

*.ntlplot files can be opened later:

- - with a standalone **NETool Plots Viewer** program included in the package

- - from any running output window by using **File➔Open**.

- Exporting data:

The plots (,) can also be exported as image files by selecting them in the **Tree View** and using

> **File➔Export Plot as Image File** …

Tables ( ) can be exported in HTML format with

> **File➔Export Table as HTML File** …

Copying to the clipboard:

- Use **Edit➔Copy** or **Ctrl+C** to copy plots that you see. Paste them into Excel, Word or other applications. Plots will be pasted as pictures at the size as you see them.

- Any portions or the whole table can be selected with the mouse and copied and pasted into Excel (or other programs where they will normally appear as an html table). To select the entire table type **Ctrl+A**.

- If you wish to have table columns and rows transposed, for convenience (or because the data is too wide to fit in the available number of columns in Excel, use **Ctrl+T** to transpose columns and rows. A repeat of **Ctrl+T** will toggle the transposition back.

Print plots with **File➔ Print**.

Table 20:Overview of NETool Results

• Parameter	• Description	• Fluid Phase
• Summary table/plot	• Main results for the whole well of oil, gas and water rates, GOR (Gas Oil Ratio), Wcut (Water Cut), BHP (Flowing Bottom Hole Pressure	•
• Pressures	• Tubing pressures, annulus pressures, reservoir pressures, pressure drop across completion	•
• Rates – ST conditions	• Rates in Stock Tank conditions: • Annulus flow rate • Tubing flow rate • Flux between well and reservoir • Flux between annulus and tubing (across completion)	• Oil, gas, water, condensate (if present) rate, GOR, LGR and water cut
• Rates – ST conditions	• Rates in Downhole conditions: • Flow rates and volume fractions in tubing and annulus • Fluxes and volume fractions between well and reservoir and between annulus and tubing (across completion).	• Downhole rate, • oil, gas, water fractions.
• Velocities	• Velocities in tubing, annulus and ICD's	•
• Permeability	• Graph of horizontal permeability Kh, vertical permeability Kv, Kv/Kh ratio and isotropic permeability ($\sqrt{Kv*Kh}$) along the well path as were used in the calculations (including upscaling, log averaging, or multiplier that was applied in the input)	•
• Saturations	• Saturations (or Flowing Fractions depending on the basis of the simulation)	• Oil, gas, water
• Relative Permeability	• Relative Permeability of oil, gas and water calculated based in input saturations and rel. perm table.	• Oil, gas, water
• Mobility	• Mobility of oil, gas and water	• Oil, gas, water
• Transmissibility	• Transmissibility; A and B transmissibility factors.	•
• Skin	•	•
• Productivity	• Productivity for oil, gas, water in stock tank conditions along the well	• Oil, gas, water
• IPR/Target	• IPR/Target and IPR/Target Table – Inflow Performance Relationship – Flow rate vs. bottom hole pressure as typical of IPR curve. IPR is shown	• Oil, gas, water

	when Global Settings→Target is other than Flowing BH Pressure.	
• Details	• A table with numeric values used in or resulting from the calculations	•
• PVT data	• Plots for all parameters present in PVT table, including multiple bubble/dew point tables.	• Oil, gas, water
• RelPerm data	• Plots for all variables in relative permeability table. If multiple tables are used (SATNUM keyword is present in Eclipse model) separate plot for each table used by NETool will be displayed	• Oil, gas, water
• External data	•	•

Comparing Results from Different Runs

NETool has capabilities to make comparison plots where results are combined from different runs. This could be done by pressing "Compare Runs" button in the results window. The button becomes enabled if more than one simulation has been made.
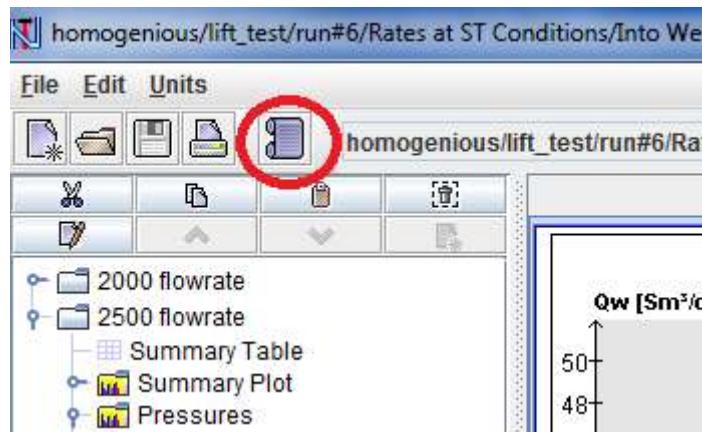
Figure 38. "Compare Runs" button

This button activates "Compare plots dialogue" shown on **Figure 39** below.
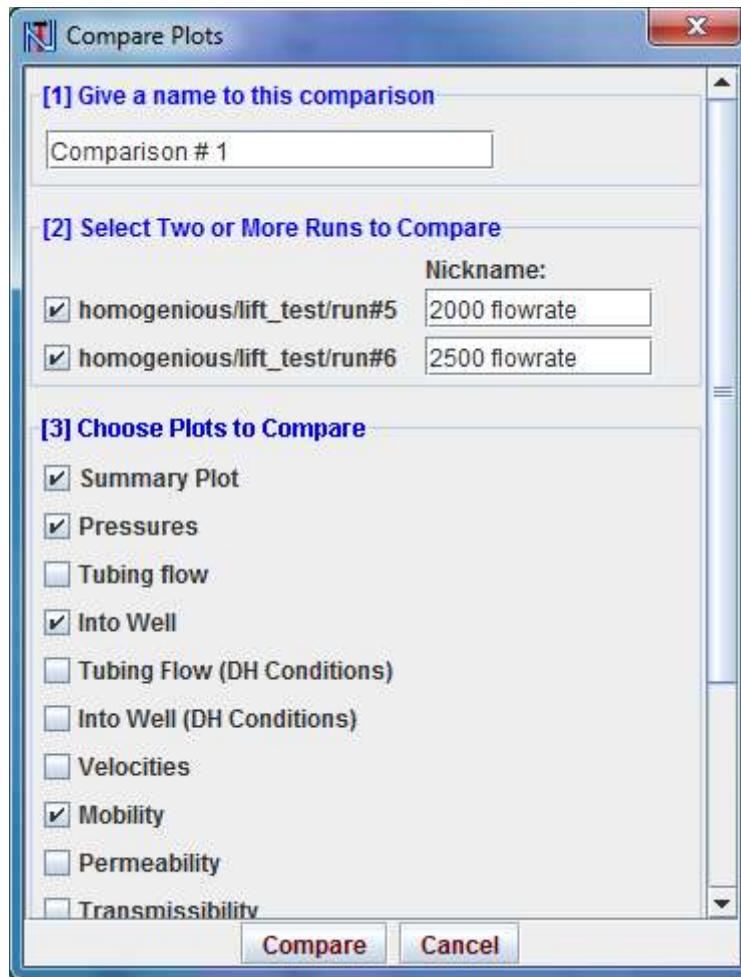
Figure 39. Compare plots dialogue.

User is supposed to:

1. Chose a name for the comparison results. This name will help to keep track of results if multiple compares has been made.
2. Select the runs he wants to compare and give them short nicknames. These nicknames will be used as curve labels (legend) on the resulting plot.
3. Choose data to compare. Only plots present in all runs will show up. (f.ex. if there were no "Pressures" plot in run#6 on **Figure 39** checkbox wouldn't show up)
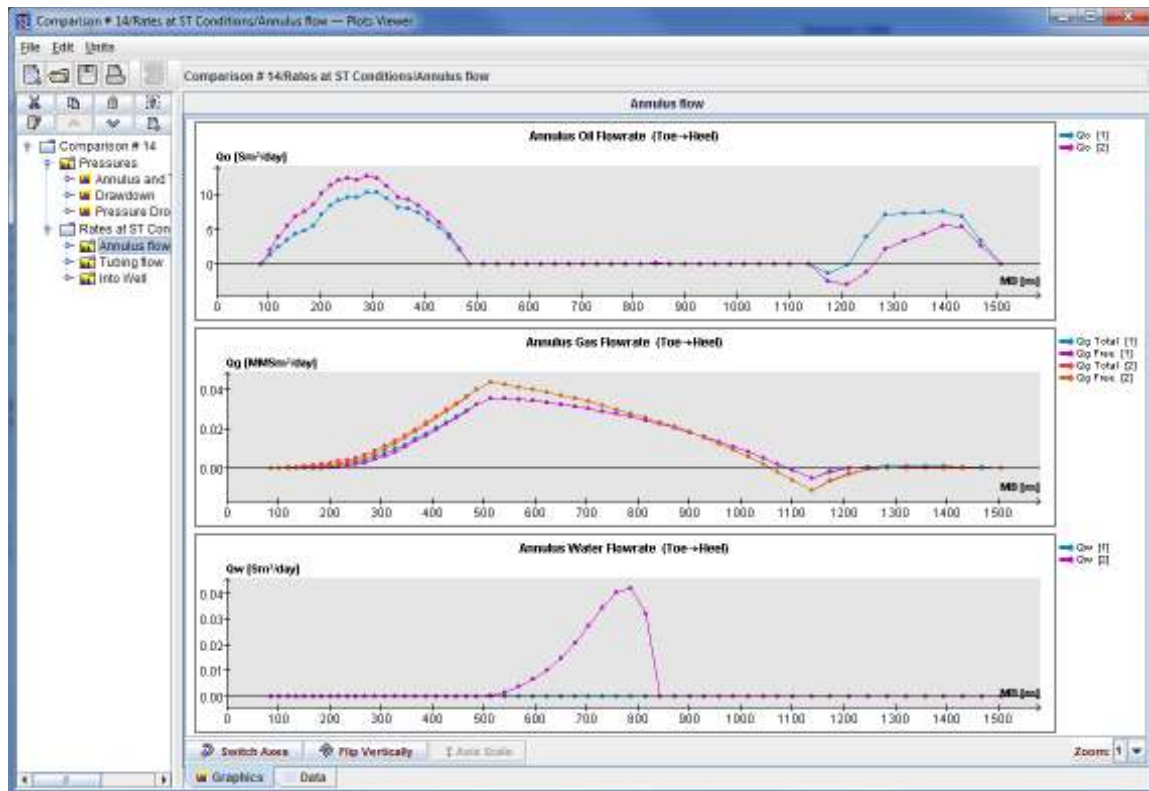
Figure 40. Comparison Plot

Working with Plots

Viewing plots and curves

Viewing a selected subset of plots and curves is possible by only highlighting the chosen folders. Use the Shift and/or Ctrl buttons in combination with mouse clicks to make the selection.

Moving, Copying, Renaming and Deleting Plots

In **Plots** Viewer, right click on the icon of the subject plot ▦ and options for **Cut**, **Copy**, **Paste**, **Delete**, **Rename**, **Move Up**, **Move Down** and **New Folder** are displayed.

Accordingly the right click on the individual curves icon ⁓ gives the same options as for plots except for New Folder.

Alternatively the above operations may be performed through predefined icons or **Edit** command.

The following action, among other things, may be performed:

One curve may be copied from one plot and pasted in another; **Copy→Paste**

A new plot may be made by selecting; New Folder – this way you make your own customized plots

The various curves may be given a new name; **Rename**

Saving Plots

Plots may be saved from **Plots Viewer** using **File➔Save As** or **Save Selection As**.

The **Save As** option saves all plots in the **Plot Viewer**, while the **Save Selection As** option saves only the selected plots.

Loading Plots

The saved plots may be loaded in the **Plots Viewer** through **File➔Open**. Please note that the plots may only be loaded from a **Plots Viewer**.

Exporting Plots as an Image File

A plot may be exported as an image file; **Plots Viewer➔File➔Export Plot as Image file**.

Copy Plots to Another Application

Plot may be copied to text editors or presentation-ware, e.g. Word or PowerPoint. The procedure is to click inside the plot window, choose **Edit➔Copy** and then **Paste** it into the destination application.

External Data – Import and Plotting

External data such as PLT data may be imported and plotted separately or in combination with simulated results from NETool. See External Plots sheet in the **NETool** window. An example is illustrated in **Figure 41**.
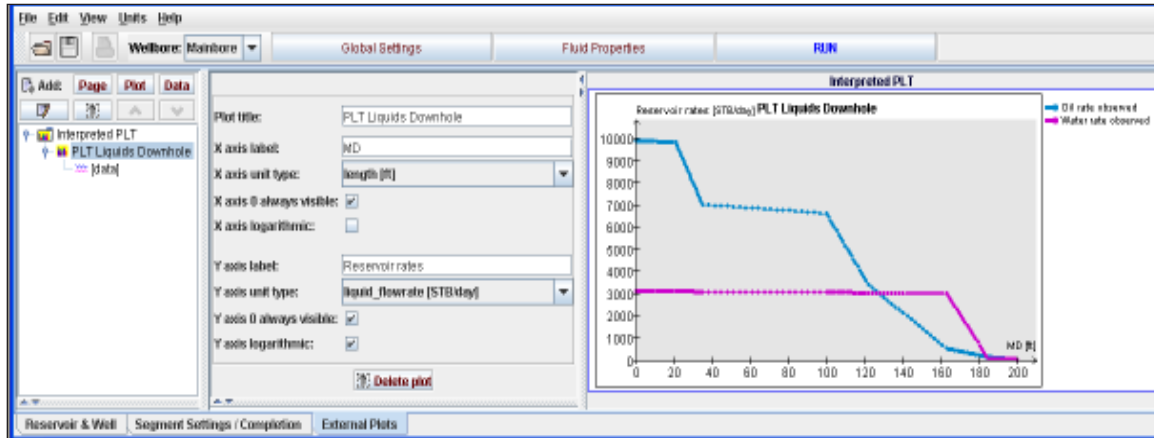
Figure 41. External Plots Sheet

The following buttons are available on the very left side:

Page – adds a new plot page to the input and results. Several plot pages may be added. The Plot page title may be edited and the Transposed box may be checked/unchecked.

Plot – adds an additional plot to the page in both input and results. Several plots are allowed per page. The Plot title may be edited and the Transposed box may be checked/unchecked. Further, the labels type and units of the axes may be selected. The value of zero on the axes may be forced be visible. The axis scale may be linear or logarithmic.

Data – adds a new curve to the active plot on the page. Several curves per plot are allowed.

Plot data can be cut and pasted to and from Excel and other software with similar data format.

Drawing type – selects how the data is plotted <u>line</u> – the data is taken as data points and connected with strait lines <u>segment</u> – the data is taken as an average for that segment of the data and is constant for the segment. Enter data top and data. <u>points</u> – the data is plotted as points

Drawing fill – changes the plots appearance <u>none</u> – data is plotted as lines. <u>zero</u> – fills the plot are from the line to the zero with color. This option is useful for data containing both positive and negative values. <u>bottom</u> – fills the plot area below the line to the bottom of the x-axis with color.

Number of Curves – use this to increase/decrease the number of curves plotted.

Note: decreasing the number of curves will cause loss of y-value data for those curves no longer present.

Legends – Enter the label for that set of y-values as it should appear on the plot. This entry appears only when number of curves is greater than 1.

Data Points – enter the data to be plotted new rows will appear automatically as needed copy/paste works from several formats including formatted text and Excel format.

Rename – Page, Plot and Data names may be edited (by right-clicking **rename**, rename-icon at top left or editing titles/legend in middle part of **External Plots** sheet)

Trash can button – deletes active page, plot or data

Up and down arrows order the page, plot and data

A simulation has to be run in order to generate external plots within the **Plots Viewer** together with the simulated data. The manipulation of the external is similar to that of the simulated results.

Configuration

Memory Size

The memory size of NETool can be increased by adding a key to the NETool's launching shortcut.

NETool is launched by run_netool.exe. Modifying the command line argument -Xmx will increase the size of maximum Java memory allowed. The default maximum memory size is 512 MB. If one needs to increase it to 1024 MB the command line in the Netool shortcut should look like:

"C:\Program Files\Landmark\NEToolxxx\bin\win64\run_netool.exe" –Xmx1024m
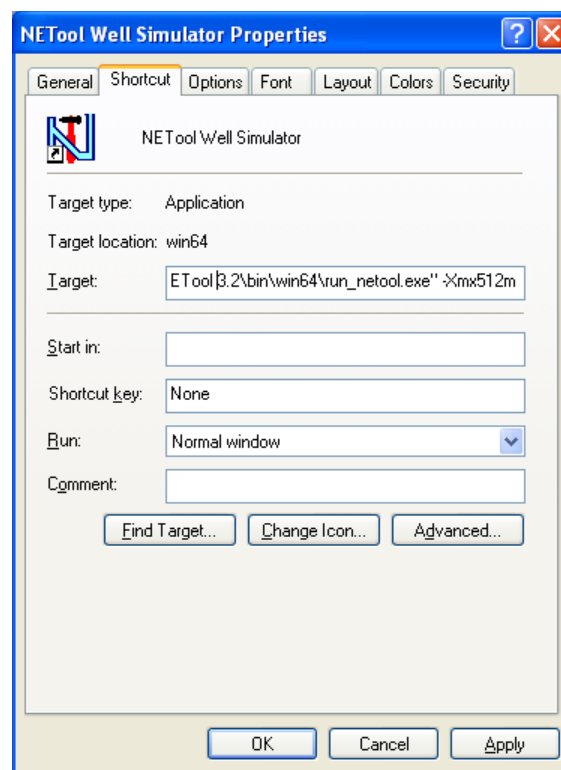
Figure 42. Configuring Java Memory Size

Configuration File

The NETool configuration file ntld.conf is located under **<<NETool_Installation Folder>>\etc\ntld.conf**. It can be edited by using a text editor.

The default location of the NETool repository/database can for example be altered by the parameter **personal_data_folder**:

Add this content on a separate line in the file:
personal_data_folder = <<Your preferred path to the repository>>

- If the value starts with a drive letter (or a slash), the value will represent an absolute path, and hence the value defines exactly where NETool repository will be located. For example: personal_data_folder = C:\some place\netool files On a multi-user systems that will cause all users using the same repository.

- If it's a relative path, i.e. it doesn't start with a drive letter and/or slash, then the path will be defined to be relative to My Documents folder of the user running NETool. For example: **personal_data_folder =My NETool Files** refers to "My Documents\My NETool Files\"**.**

- $USER (case sensitive!) can be used as parameter and will be replaced with the system user name. For example, if the Windows user name is "John Smith": personal_data_folder = C:\some place\$USER\NETool\$USER means "C:\some place\John Smith\NETool\John Smith\"

UNC network paths such as **\\server1\abc** can also be used on Windows

Personal Data Folder location can also be changed through **Edit->Preferences->Folder for User Files.**

Units

Data can be supplied and reported in a variety of units. NETool automatically converts between different units.

Four main unit sets are available: **SI**, **Metric, Metric + Diameter in inches** and **Field**. However, NETool has more units available than the Metric and Field unit collections offer together.

Use **File→Units** to select Metric or Field as preferred unit set. This controls the preferred units when displaying data values and default units for entering data values.

If you create a new well case the default units system is "Metric + Diameter in inches".

The selected unit set for a well case is stored with the well case and used when the case is reopened in NETool.

Please note:

NETool's Metric and Field unit collections differ slightly from the Eclipse definitions.

PVT tables using Eclipse format must be defined in Eclipse's own Metric or Field unit definitions.

Convergence Problems and Simulation Time

Simulation time

Simulation time is generally a function of numerical resolution and accuracy requirements.

Numerical resolution is controlled by number of nodes and number of intermediate nodes

In addition can grid upscaling use considerable time if the number of nodes or number of grid blocks is high. The accuracy of upscaling can be modified in order to reduce the time used on upscaling.

Convergence Problems

NETool is a numerical simulator and may experience convergence problems. It cannot be guaranteed that any case can be solved by NETool.

Typical causes of convergence problems are (in order of occurrence):

- Too wide pressure limits (Global settings->Advanced->well pressure limits)
- Inconsistent or unphysical input data, especially PVT data
- Areas with small pressure drops which causes directional oscillations
- High saturation, reservoir pressure and/or permeability contrasts along the well
- High oil viscosity

Recommended remedial actions are to consider i) input data, ii) numerical parameters in **Global Settings→Advanced** and iii) adjust nodes/intermediate nodes:

- Verify PVT data and other input
- Adjust Well Pressure Range
- Increase or reduce Number of Intermediate Nodes (consider node spacing)
- Increase or reduce Number of Nodes
- Enable "Smooth Pres over" or "Iteration step damping" in Global settings->Advanced
- Adjust Momentum Balance control
- Use correct Hydrostatic options
- Increase "Precision of calculation"
- Reduce "Stability"
- Adjust PVT Derivatives settings

Recent Developments

Developments from 5000.0.1.0 to NETool 5000.0.0.1:

- **Multi-annulus feature is implemented.** Netool can simulate four annuli instead of one. "Segment Settings/Completion->Hole&Completion->Segment type" is now replaced with five columns: "Casing/liner", "Screen", "Inflow control", "Stinger", "Tubing". Each of these layers has an individual completion.
- **Netool UI uses new "Nimbus" look & feel and Java version 7.**
- **Rs and Rv are now independent unknowns at every node instead of one average value for the whole well.** Old model can be enabled by checkbox in "Global settings->Advanced->Simplified Rs/Rv handling = true"
- Added the following functionality to the Completion Plot: resizable segments, autozoom on selected segments, zoom slider (results window only), add/delete segments.
- ICD's are grouped under "ICD" completion type.
- Option added for ICDs and valves to use Discharge Coefficient vs. Reynolds Number table.
- Option added to define reference reservoir pressure for hydrostatic correction at a certain MD rather than heel.
- Reservoir pressures can now be "upscaled" (like permeabilities) and low pass filtered (Global settings->Advanced->smooth Pres over).
- Re-arranged panels in the main UI.
- New models for Halliburton Autonomous ICD.
- Added "momentum" (Global settings->advanced) option.
- Added "Cables in annulus".
- Moved "Junction Type" into "Segment Settings->Hole&Completion"
- Restored and improved OLGAS coupling.
- Added API to netool.dll to run Netool UI/Plots from reservoir simulator.
- Greatly improved Netool-Quiklook coupling.
- Multiple bug fixes.

Developments from 3.2.2 to 5000.0.0.1

Coupling:

- Dynamic coupling with "Quiklook" reservoir simulator has been implemented.
- Netool SDK designed for coupling Netool with third party software is a product available under "NETool DLL" brand. It comes with Netool setup and can be selected during installation process.

Simulation features:

- New vertical pressure drop correlations at Global settings->General->Pressure drop method: "Aziz, Govier, Fogarasi" and "Duns & Ros".
- New completions: Generic Screen, Perforated Pipe Screen.

- New ICD completions: Statoil Autonomous ICD and Halliburton Adjustable ICD
- Packers can now be sealing or non-sealing.
- Netool reads RelPerms from Eclipse-format restart files if available.
- Supported Grid coordinates in addition to Geographic coordinates with dynamic switching between the two coordinate systems.

GUI/usability improvements:

- Netool GUI adapted to Landmark 5000 series software release.
- Added simplified training mode designed to be used in training classes.
- New interface for comparing plots.
- Added Completion Schematics the input screen and "Plots Viewer".
- Added "Inspect Gridblock" button on Reservoir view screen: an easy way to get information about any gridblock in the model (all data + geometry).
- Added window for automatically adding and removing packers between every segment.
- Modified File->Open/Save dialogue window.
- Added "Data" tab for every plot which shows the actual data used.
- Added more options for vertical plots scaling ("Axis Scale" button under plots)
- Average values for some plots at the bottom of "Data" tab in plots viewer are printed.
- Screen completion regrouped: instead of WWS user has to chose "Screen". Under it the following types are available: WWS and new completions Generic Screen, Perforated Pipe Screen.
- Bernoulli pseudo-segments are hidden in the output.
- "Nodes" terminology replaced with "segments" in Reservoir input UI
- Rearranged Cased Hole Gravel Pack inputs.
- Moved "PLT overlay" to Global Settings->General.
- Single "External plots" in a well case, not per lateral as before.
- Added %-of-total scale on the right hand side of stock-tank flow rates plots.
- Added SI Units mode.
- New licensing GUI
- Enter License does NOT use FLEXlm's lmhostid or lc_hostid() any longer. Instead it gets the list of network interfaces, removes all that look like non-hardware ones and print MACs together with interface descriptions.
- Created C#/dotnet bindings for netool.dll

Bug fixes (also improved in 3.2.2):

- Plots->Save All As: Fixed wrong encoding in the XML header.
- Enter log sometimes fail.
- Fixed problems with some Eclipse-format files originating from CHEARS.
- If the segment type in Reservoir tab is changed, the tree-view in Segment Settings might not update
- Changed default values for minor loss coefficient for Tubular and Channel ICDs (0->1.6) and "Max Well Pressure" in "Well Pressure Range"

Developments from 3.0.6 to 3.2.2

Modeling:

- Fixed wrong usage of hidden pressure limits
- On Windows increased stack size limit to prevent crashes in rare cases
- Significantly improved algorithm for grid-block intersections.
- Improved cases with exceptionally high number of gridblocks

GUI fixes and improvements:

- Fixed copy-paste to MS Office-2007
- fixed some plots failing to "compare". (For example "Oil PVT")
- fixed dialogs sometimes showing with wrong (almost zero) size.
- "Enter Log" now uses table instead of text area.
- fixed reservoir_view->top_panel->"Depth" for reservoirs with LGRs
- Fixed velocity plots offset by 1 segment.
- fixed empty error message if plots viewer started with no license
- proper error message in some cases when output is too big
- improved layout of "External Plots->[data]"
- disabled File->Export Netool binary grid for gridless cases
- Improved cases with exceptionally big tables (e.g. "Details" table) by parsing it only if clicked on.
- Increased Java max stack to 512MB to avoid OutOfMemory errors
- Added online help for Tubular and Channel ICDs, updated Nozzle ICDs help

Installation/startup defects:

- Fixed InstallShield installing wrong flavour of Java on Windows64.
- Print more info for support and do more checks at startup

General:

- Enabled Sentinel dongles code to get old dongles working.
- The GUI for entering Sentinel dongles is now removed.
- Linux version: fixed compatibility with RHEL-5.

Developments from 3.0 to 3.0.6

- Re-branding: NETool has been acquired by Halliburton from AGR Petroleum. All trademarks, contact information, support, etc has been adapted to be used by Landmark Graphics Corporation (LGC) - Halliburton software division.
- Licensing: A single-user hardware locked version of FlexLM license is now supported. NETool proprietary license type is now discontinued.
- Added input option for PLT data: PLT data could now be entered in the output folder under Global Settings.
- New Injector PI model: A new model that simulates the start up of the injection process is added.

- Added support for Eclipse PVCO, PVCDO keywords in PVT table
- Added support for 64-bit platforms: NETool runs on the 64 bit versions of the following operating systems: Windows XP Vista; Linux.
- Added completion types: Three general ICD types are now available as completion parts:
  - Nozzle ICD (formerly named Generic ICD)
  - Tubular ICD
  - Channel ICD
- Added plots in result viewer
- Fluid velocity within an ICD unit can be plotted and Reynolds number within an ICD unit is now available in the table details.
- Added a new feature to define intermediate nodes: A new option to define intermediate nodes has been added as an alternative way to the original for intermediate node definition. The new option defines the location of intermediate nodes based on a user-defined-spacing. The spacing can be defined independently for each segment. The new method calculates the number of intermediate nodes required for a segment based on the defined spacing, and then distributes those nodes evenly within the segment. The new option makes it easier to control the resolution of the model when the individual segment sizes vary.
- Enhanced numerical performance: General convergence improvements, for example for multi-lateral wells, wells with variable fluid phase fractions along the well path and for cases with lift curves. Improved calculations speed of the numerical solver.
- Non-Darcy flow behavior: Improvements and new features to general non-Darcy skin and Forchheimer skin specifically have been introduced. An option to model non-Darcy pressure drop in gravel packs and collapsed annulus has been added.
- Enhanced momentum balance modeling: Improved flow modeling by including compressibility effects.
- Improved flow modeling by adding the Bernoulli effect resulting from velocity changes caused by flow area variations.
- Added a new feature allowing lift curves for injectors: Injector lift curves can be used to control injection by a tubing head pressure or to calculate tubing head pressure when injection is controlled by other simulation targets. Injector lift curve is defined by the **Eclipse** format **VFPINJ** keyword.
- New features for more advanced inflow modeling: Added an option for non-linear productivity models. For oil and condensate producers the option models the effect of condensate banking and gas-break out in near-wellbore area. For dry gas producers the option models the inflow similar to the pressure-squared method.
- Enhanced modeling of flow between completions sections adjacent to Open Hole Completions: Tubing nodes in Open Hole completions can now be connected with both annulus and tubing nodes in completions types that have flow within both the annulus and the tubing. Previously the Open Hole completion was only connected to the tubing nodes in the adjacent segment.
- Enhanced modeling of multiphase hydrostatic calculations: The option to include hydrostatics in the well hydraulics calculations has been enhanced. The

hydrostatic calculations are now based on local fluid densities rather than an average fluid density for the entire well. The new modeling handles multiphase fluids and acts separately for annulus and tubing.

- New feature for relative permeability modeling: An additional option has been added where relative permeabilities are defined based on Wyllie's correlations. Both Wyllie's correlations for i) cemented sandstone, or oolitic limestone and ii) well sorted unconsolidated sandstone is supported.

- Enhanced modeling of cross-flow: Two-phase modeling of crossflow is supported.

- Enhanced definition of skin: Input of different skin has been reorganized. Output of skin calculations has been improved.

- New PVT feature for definition of formation volume factor (Bg) for wet gas: The gas formation volume factor, B_g, at a particular Gas Condensate Ratio (R_v) as input into NETool (or taken from the reservoir grid) can now be calculated by linear interpolation between the saturated and dry gas values.

- Added a new feature that allows for Power Law fluids: Single phase Power Law fluids can now be modeled.

- Extended Phase modes: More phase modes are support, e.g. two-phase Oil & Water and Gas & Water…

- Extended PVT modeling: Simultaneous modeling with vaporized oil (R_v) and solution gas (R_s) is now supported.

- Enhancements to Model Builder (pre-processor): Bird-eye view of the reservoir grid has been altered. It has been rotated 90° such that North is up and East is right.

- Enhancements to Results Viewer (post-processor): Added more information to the table Details. Added new plots: Velocities in annulus and tubing Inflow volume fractions and tubing volume fractions

- Graphical Presentation of External Data: External data can now be added and presented together with results from NETool simulations. This capability facilitates a bit easier history matching of PLT results.

- Grid Coarsening allowed: Eclipse grids with Grid Coarsening are now accepted as input.

- Phase slip for collapsed annulus and gravel pack: Phase slip for Darcy flow is introduced for collapsed annulus and gravel packs.

- Support for Microsoft Vista: NETool can now be run on the operating system 32-bit Microsoft® Vista.