Geothermal Modelling





Ruggero Bertani

Geothermal Innovation & Sustainability
Enel Green Power

Trieste, December 2015

THERMAL FLUX

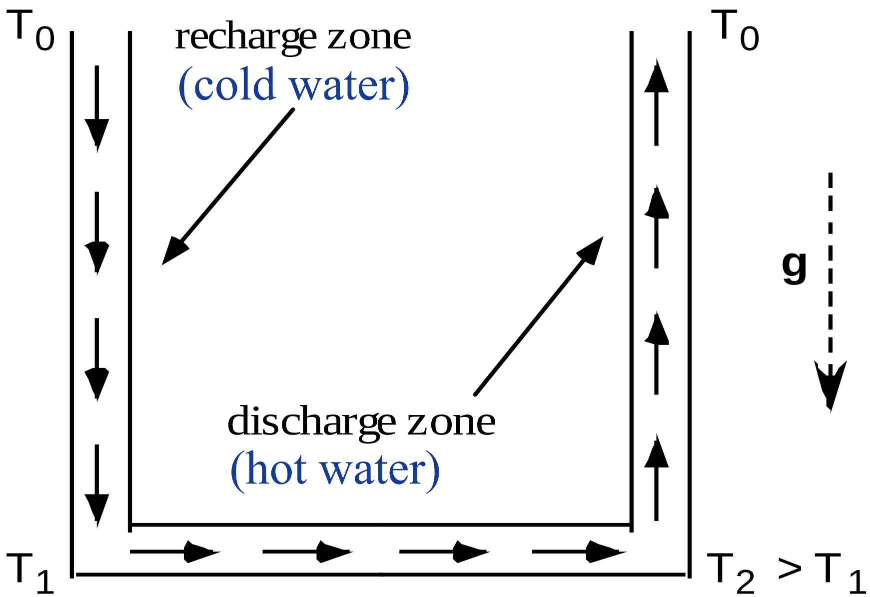
$$\mathbf{q} = -\mathrm{K}\nabla\mathrm{T}$$

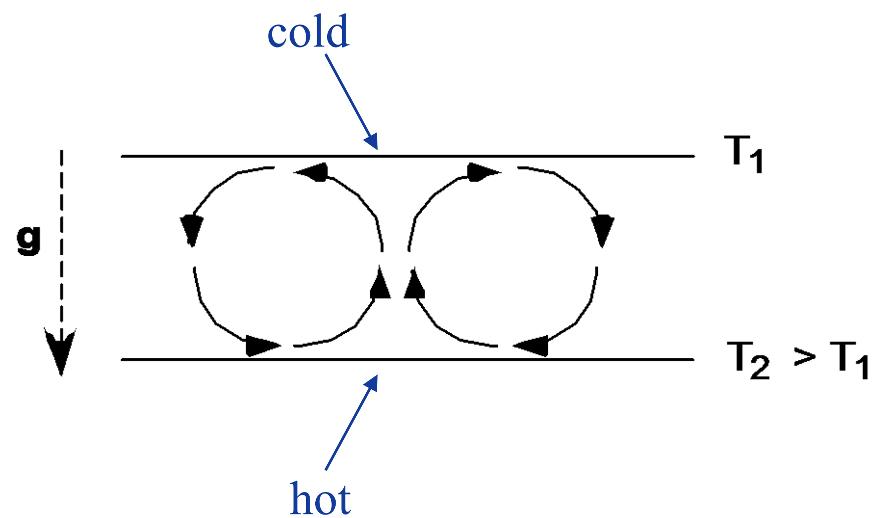
$$q \approx 2\frac{W}{m^{\circ}C} \times 0.03\frac{^{\circ}C}{m} = 0.06\frac{W}{m^2}$$

In 1 km^2, standard thermal flux radiated is **60 kW**

Geothermal Modelling

Natural circulation model: convective cells.



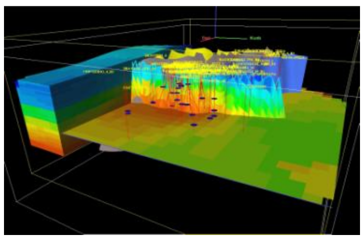


Numerical modeling of geothermal fields

Input

Reservoir characteristics

Geological / conceptual model



Well testing

- Injection test
- Build Up
- Drawdown

Reservoir geometry

- *extension*
- *thickness*
- *system geology*
- *faults*

Reservoir geology and characteristics of the rocks

- *permeability*
- *porosity*
- *comprimibility*
- *thermal conductivity*
- *density*

Physico-chemical data

Monitoring of existing / exploratory wells

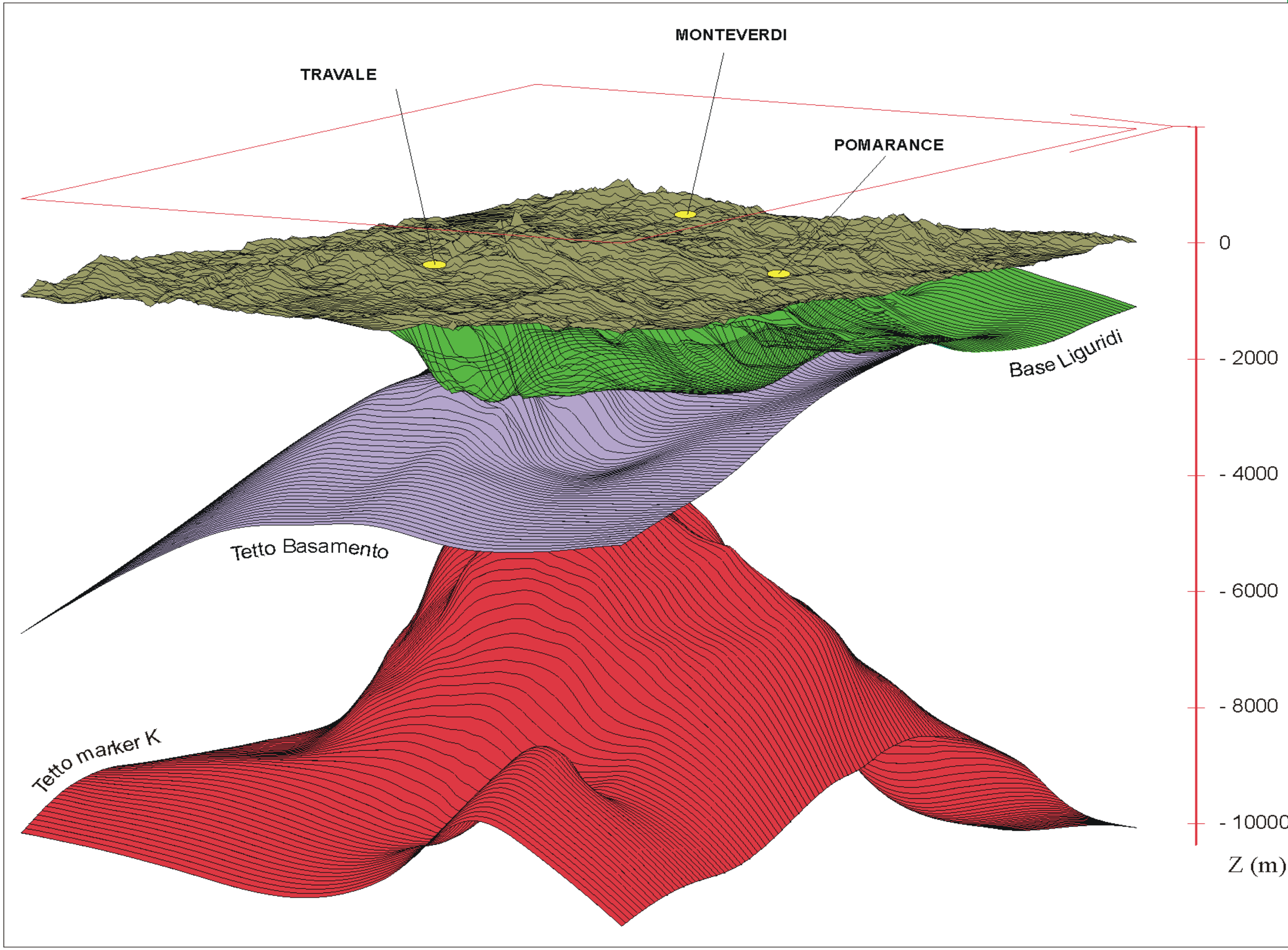
- Fluid monitoring in existing wells
- WHP monitoring in observing wells
- Monitoring of water level in wells
- Build-up temperature and T&P Logs

3D distributions of temperature and pressure in the reservoir

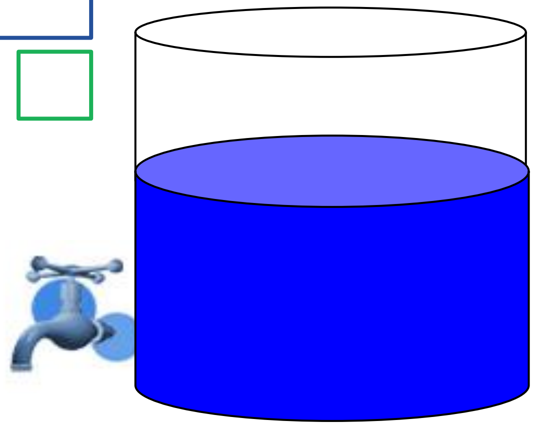
Initial conditions

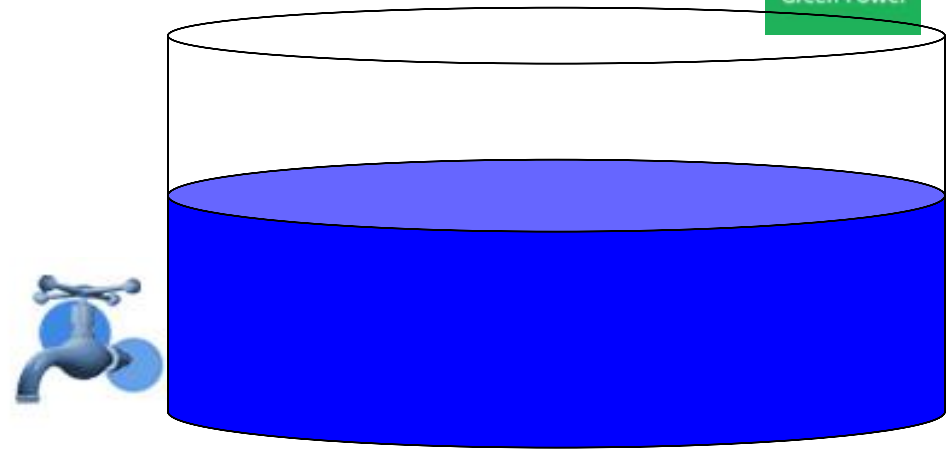
System evolution in the time

Geothermal Modelling

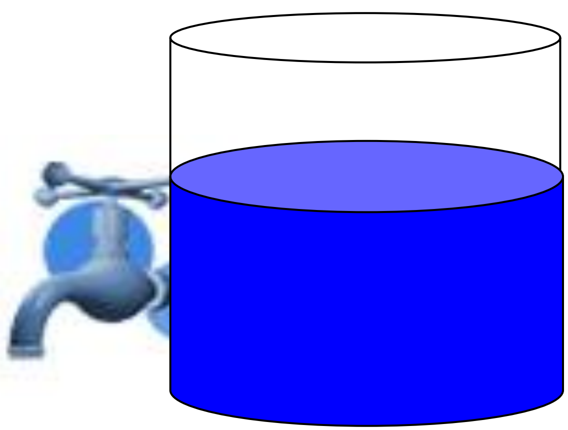


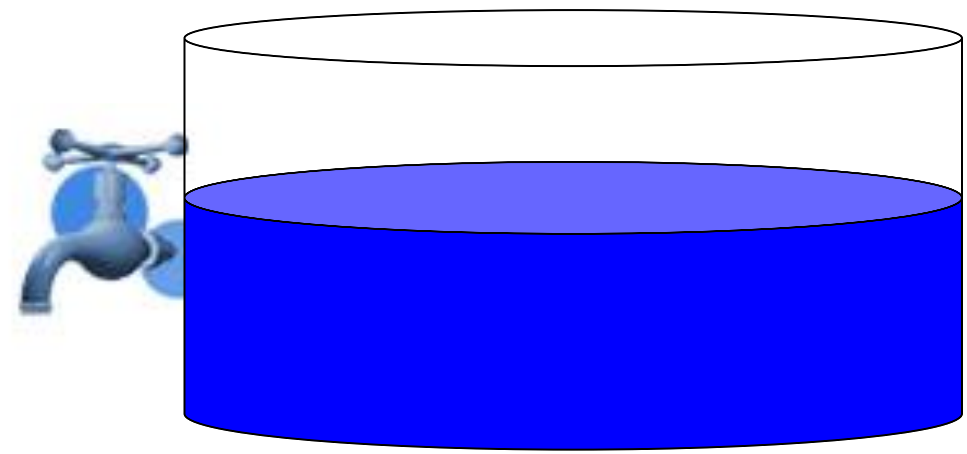
Geothermal Modelling



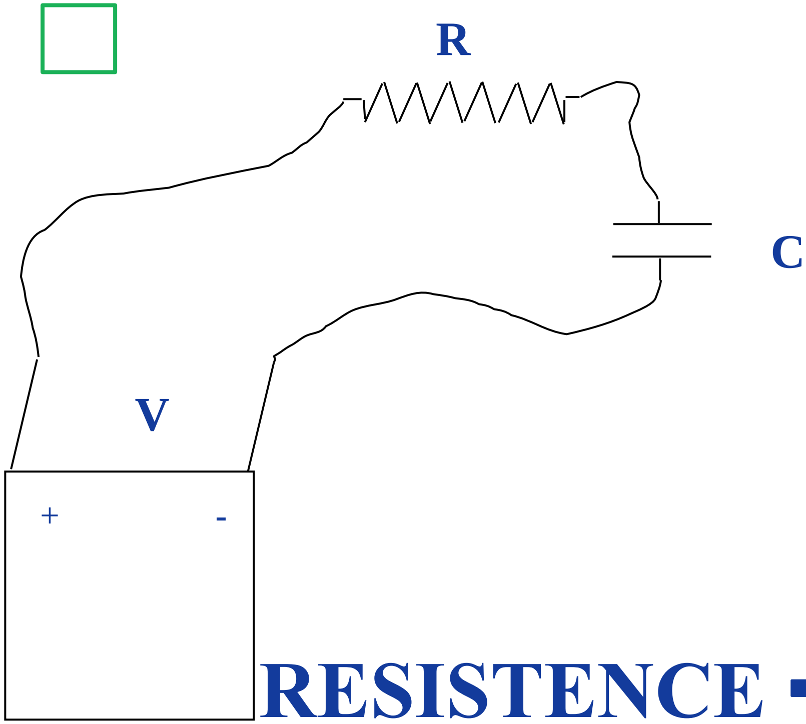


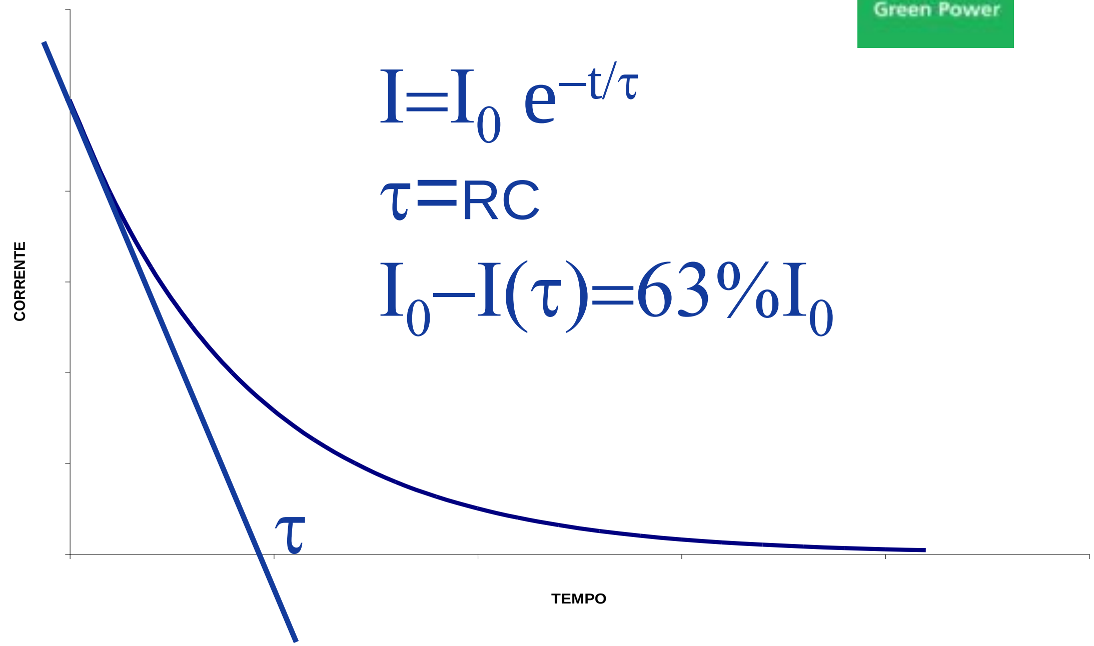
TAP → PERMEABILITY
CAPACITY→VOLUME*POROSITY





Geothermal Modelling





$I = I_0 \, e^{-t/\tau}$

$\tau = RC$

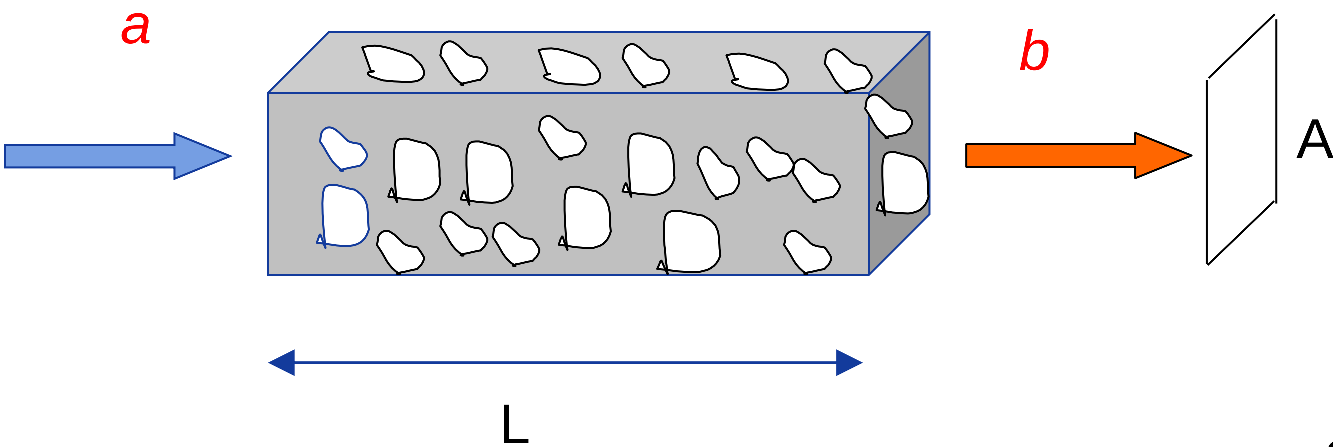
$I_0 - I(\tau) = 63\% I_0$

RESISTENCE ➔ PERMEABILITY

CAPACITY➔VOLUME*POROSITY

VOLTS ➔ PRESSURE

Geothermal Modelling



The driving force of the fluid through the porosity from *a* to *b* is the Pressure Difference

$$q = \frac{\Delta P \, A \, K}{\mu \, L}$$

$q=[m^{3}]\,[s^{-1}]$

$\Delta p = [Pa] = [kg]\,[m^{-1}]\,[s^{-2}]$

$A= [m^{2}]$ **$L = [m]$**

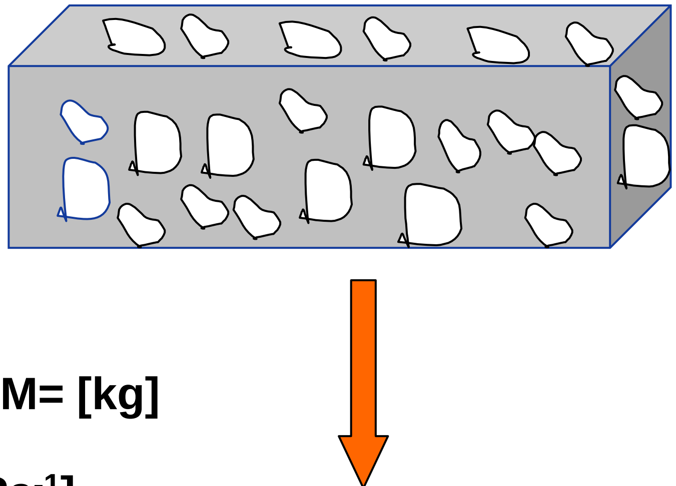
$\mu = [Pa]\,[s] = [kg]\,[m^{-1}]\,[s^{-1}]$

K= Permeability$[m^{2}]$

Geothermal Modelling

V

ϕ



ΔM

Pressure Reduction is proportional to the Extracted Mass

Comprimibility= (DV/V)/DP

V= [m³] **M= [kg]**

c_t c_{fl} c_r = [Pa⁻¹]

ρ= [kg] [m⁻³]

C_M= [kg] [Pa⁻¹]

$$\phi c_t = \phi c_{fl} + (1-\phi) c_r$$

$$\Delta M = \phi c_t \, V \, \rho \, \Delta P$$

$$C_M = \Delta M / \Delta P$$

$$C_M = \phi c_t \, V \, \rho$$

$$\Delta P = \frac{\Delta M}{\phi c_t \, V \, \rho}$$

ϕ= **Porosity**

Geothermal Modelling

Enel Green Power

Darcy Law (Henri Darcy, 1856)

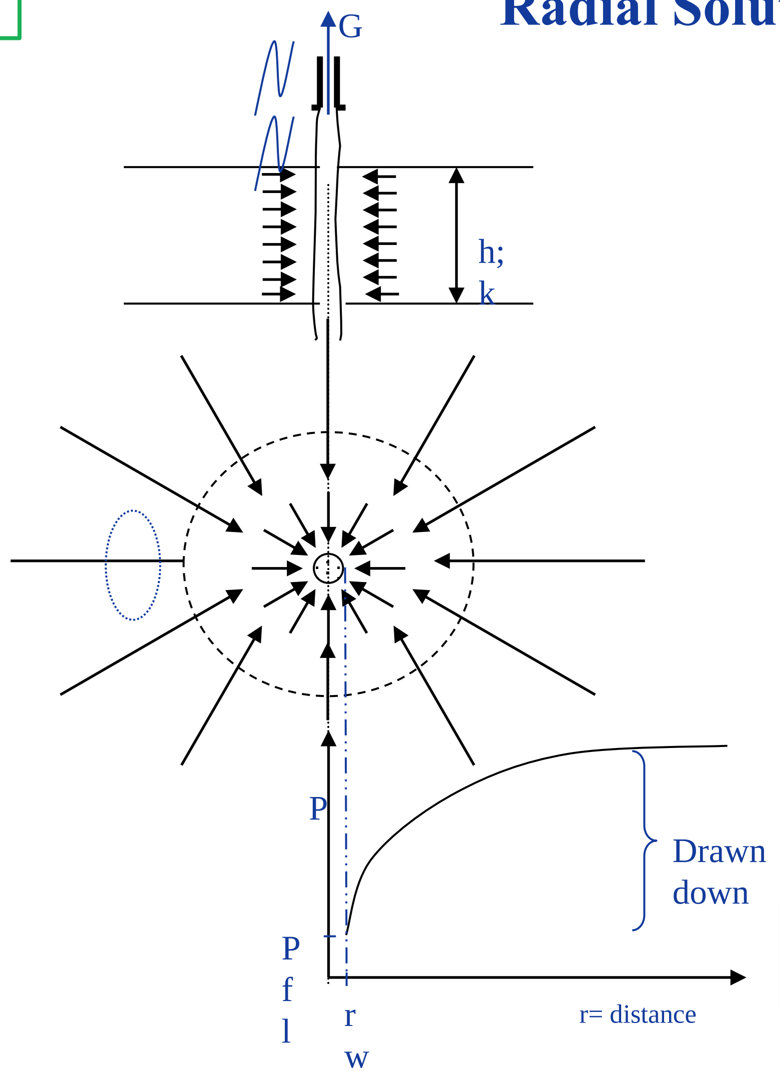
$$\mathbf{F} = -k\frac{\rho}{\mu}(\nabla P - \rho\mathbf{g})$$

$$\eta = \frac{k}{\mu\phi c_t}$$

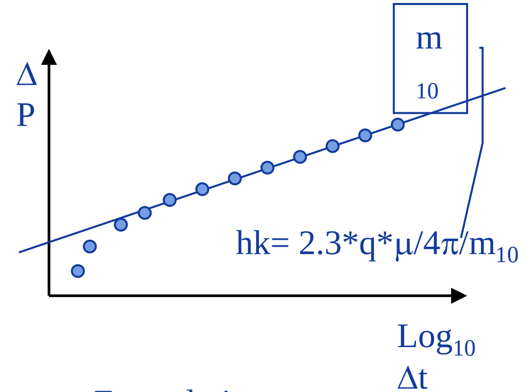
$$\begin{pmatrix} F_x \\ F_y \\ F_z \end{pmatrix} = -k\frac{\rho}{\mu}\begin{pmatrix} \Delta P/\Delta x \\ \Delta P/\Delta y \\ \Delta P/\Delta z - \rho g \end{pmatrix}$$

Geothermal Modelling

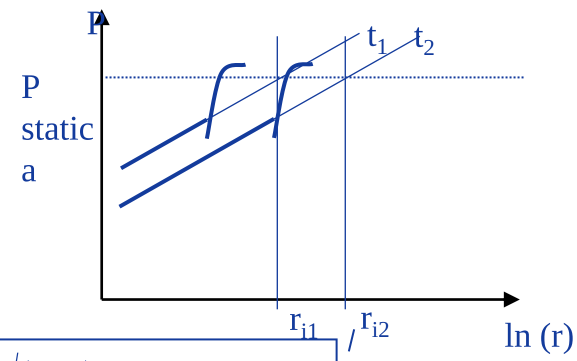
Radial Solution



From draw down test



From drainage cone



$r_i = 2\sqrt{(\eta t/\gamma)}$

$\gamma = 1.781$

Radial Solution

$$\Delta p = \frac{q\mu}{4\pi hk} E_1\left(\frac{r^2}{4\eta t}\right)$$

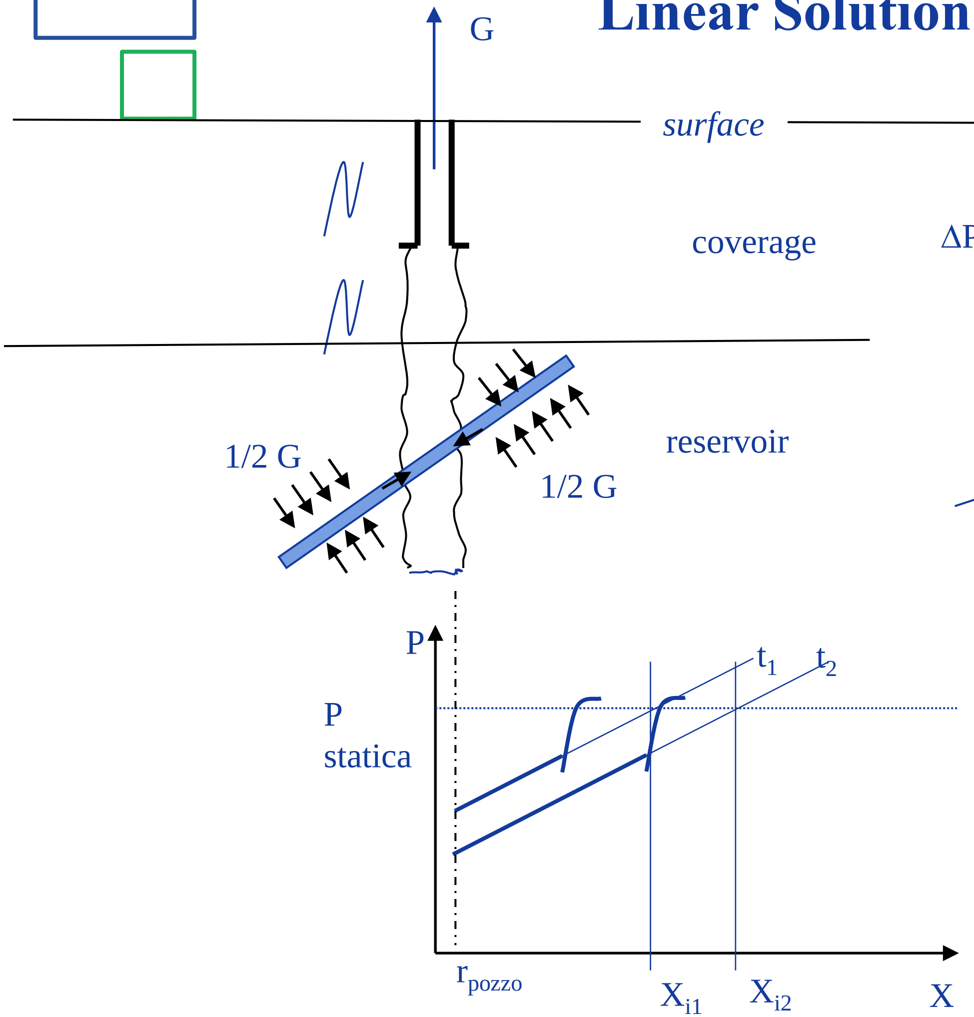
$$E_1(x) = \int_x^{\infty} (1/y) e^{-y} dy$$

$$\Delta p = \frac{q\mu}{4\pi hk}\left(\ln \frac{4\eta t}{\gamma r^2}\right)$$

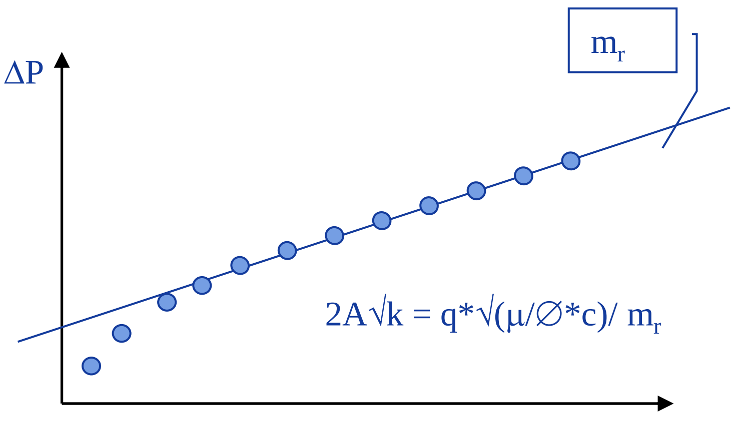
$$\frac{\eta t}{r^2} > 10^2$$

Geothermal Modelling

Linear Solution



From draw-down test



Afracture surface[m^2]

k= permeability [m^2]

Q= flow rate[m^3/s]

m_r= slope [Pa/√s]

μ= viscosity [Pa*s]

∅ = porosity

c= comprimibility [Pa^{-1}]

Linear Solution

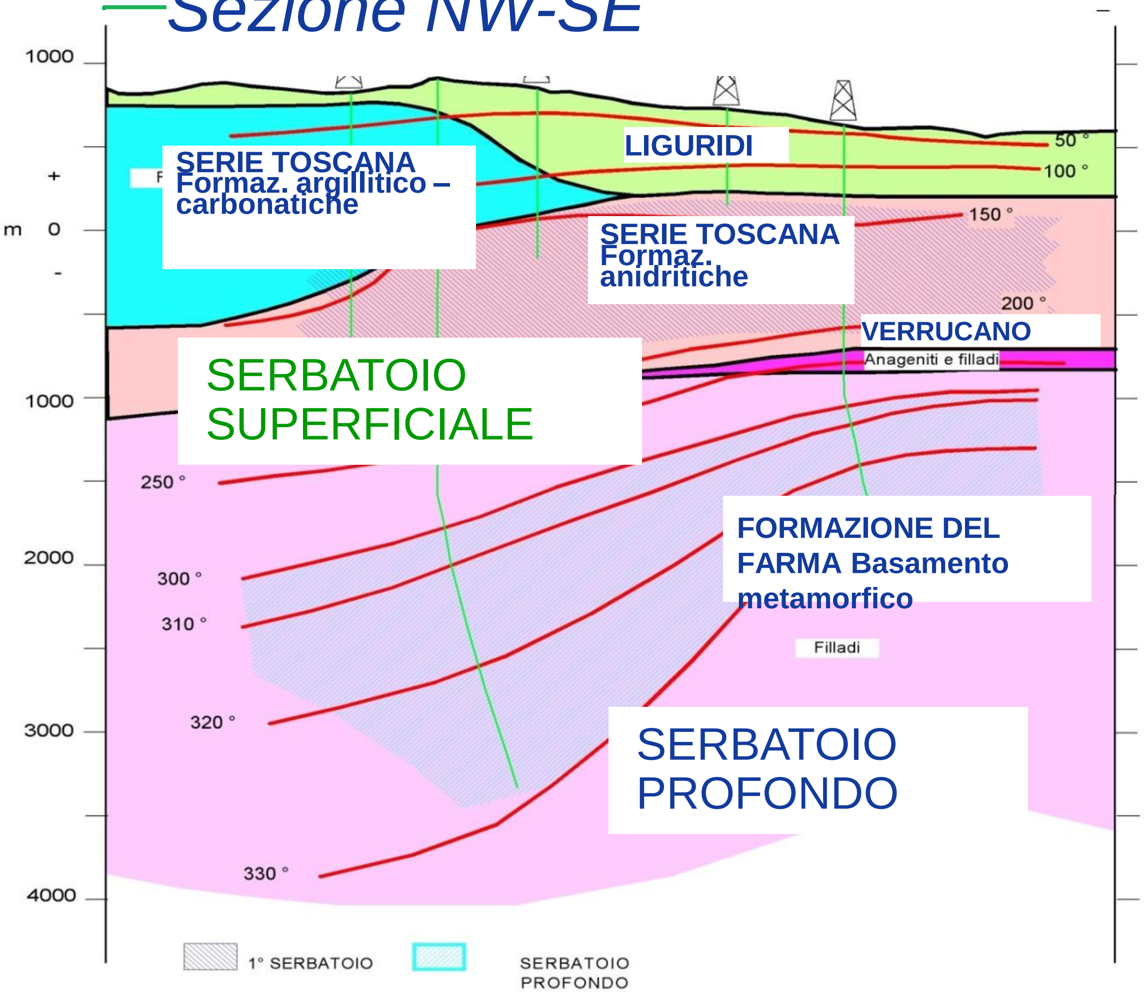
$$\Delta p = \frac{2q\mu}{Ak}\sqrt{\eta t}\,\mathrm{ierfc}\left(\frac{x}{2\sqrt{\eta t}}\right)$$

$$\Delta p = \frac{q\mu}{kA\sqrt{\pi}}\sqrt{\eta t}$$

$$\frac{x}{2\sqrt{\eta t}} \longrightarrow 0$$

Geothermal Modelling

Sezione NW-SE



SHALLOW
- Shallow reservoir in "tuscany series"
- Top: 400 - 1000 m
- Thickness 800 - 1000 m
- Temperature 170 - 200°

DEEP
- Metamorphic Basement depth 2000 - 3000 m
- Top: 300° C isotherm
- There is no particular lithological signature

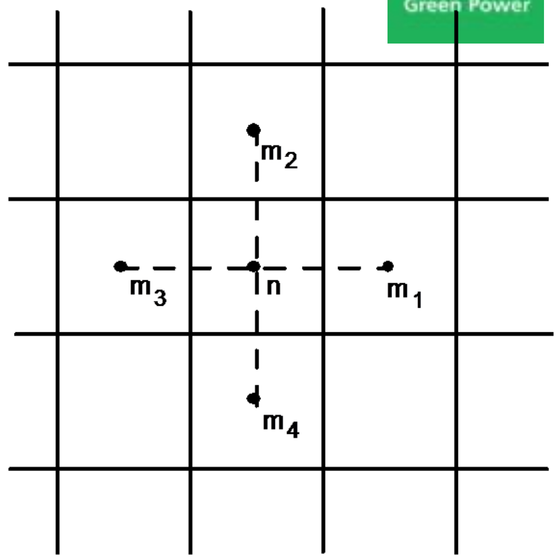
Geothermal Modelling

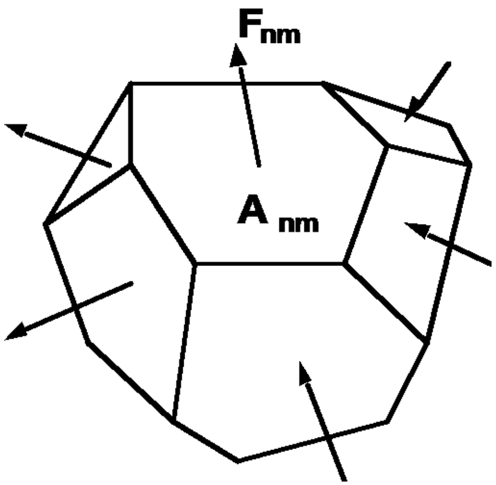
Space and Time discretization

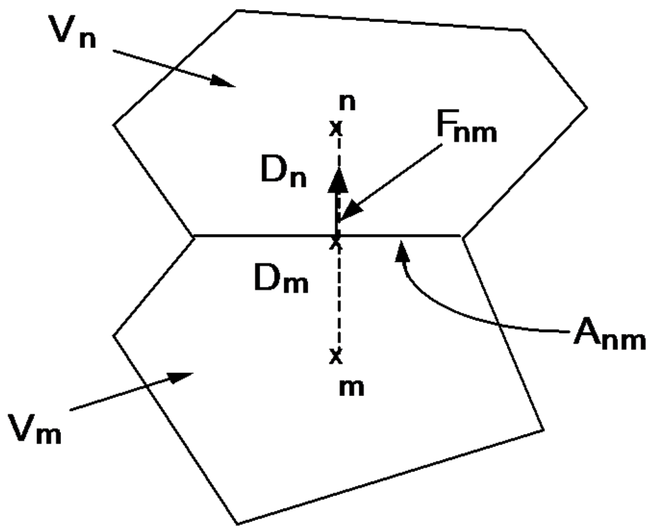
Energy and Mass balance

Darcy Law

Recalculation of Pressure and Temperature

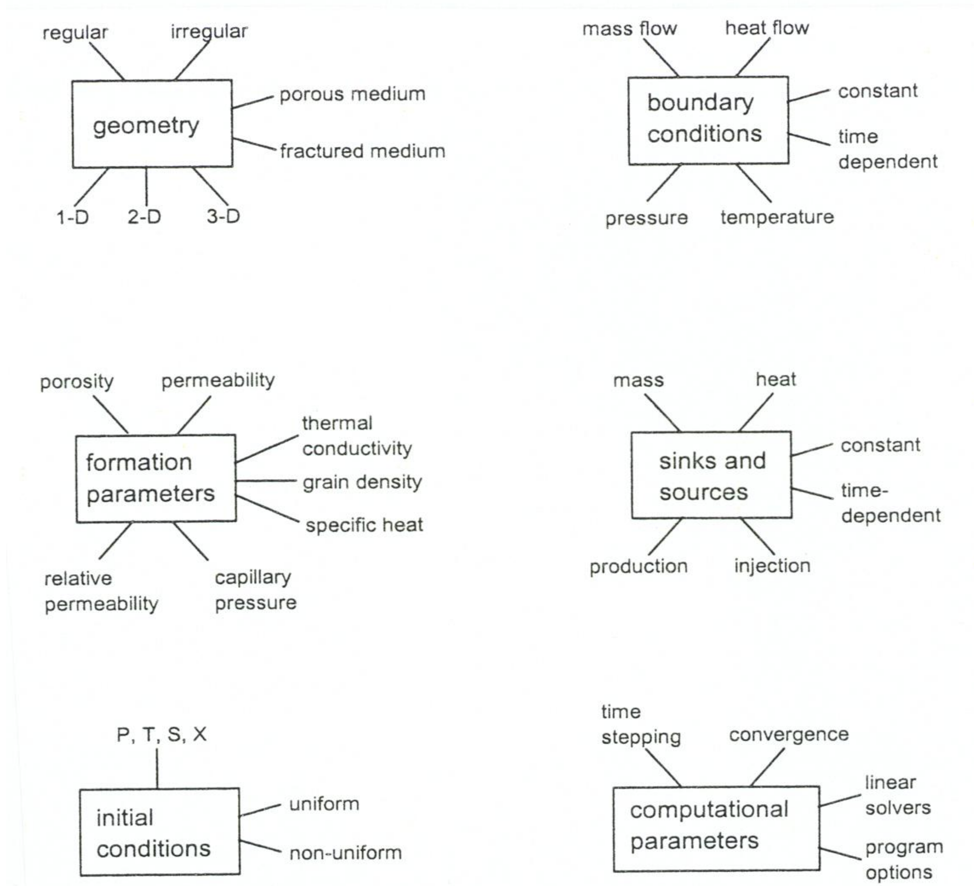






Geothermal Modelling

They address six categories, namely reservoir geometry, formation parameters, boundary/initial conditions, sinks and sources and computational parameters



Simulation input data group (source: Pruess, 2002).

Geothermal Modelling

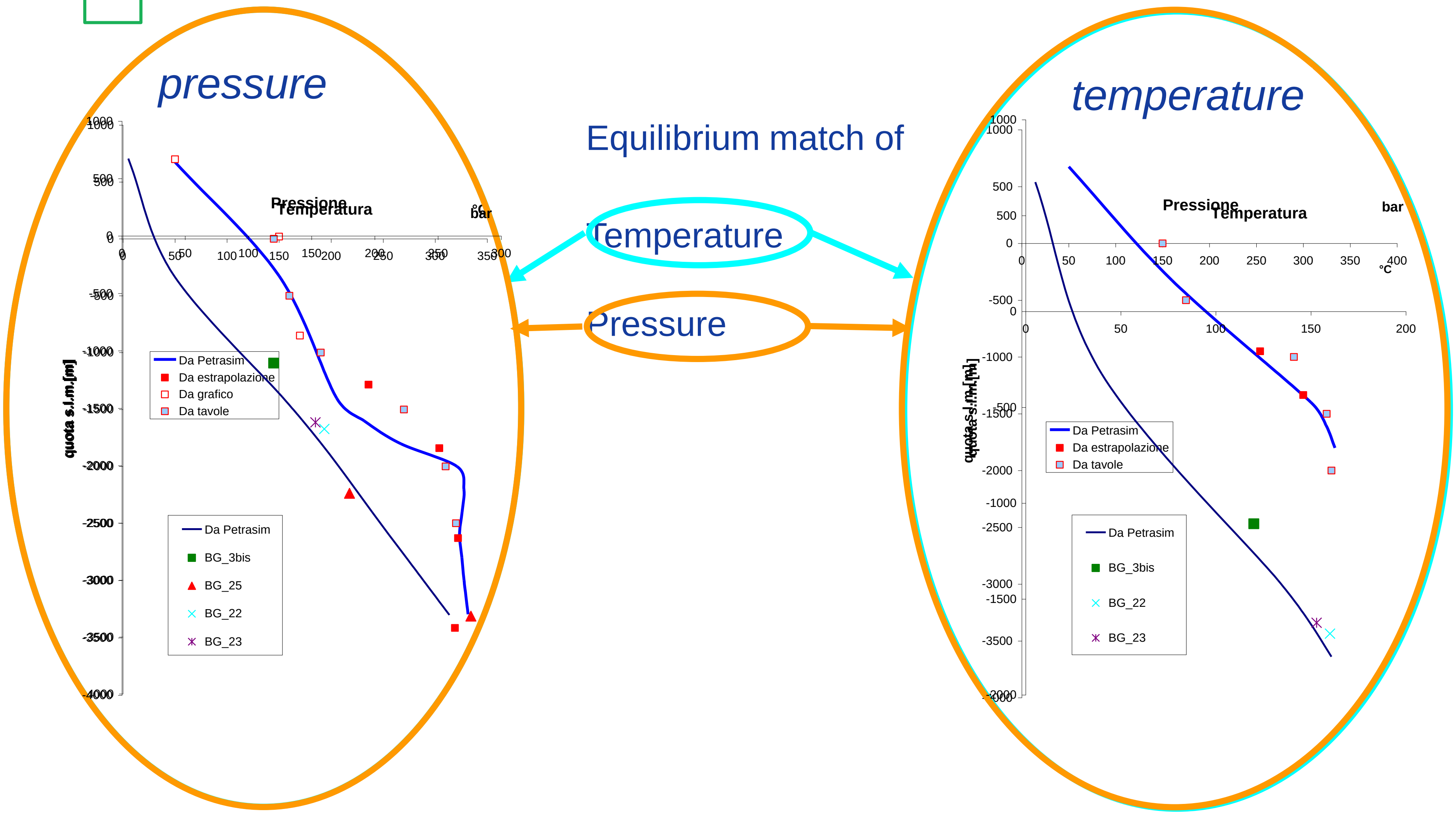
FOR EACH CELL:

- density (2800 kg/m^3)
- porosity (1.3 %)
- permeability (m^2, X,Y,Z)
- conducibility (3.5 W/m°C)
- specific heat(850 J/kg°C)
- comprimibility (3 x 10^{-11} m^2/N)
- expansivity (10^{-5} 1/°C)

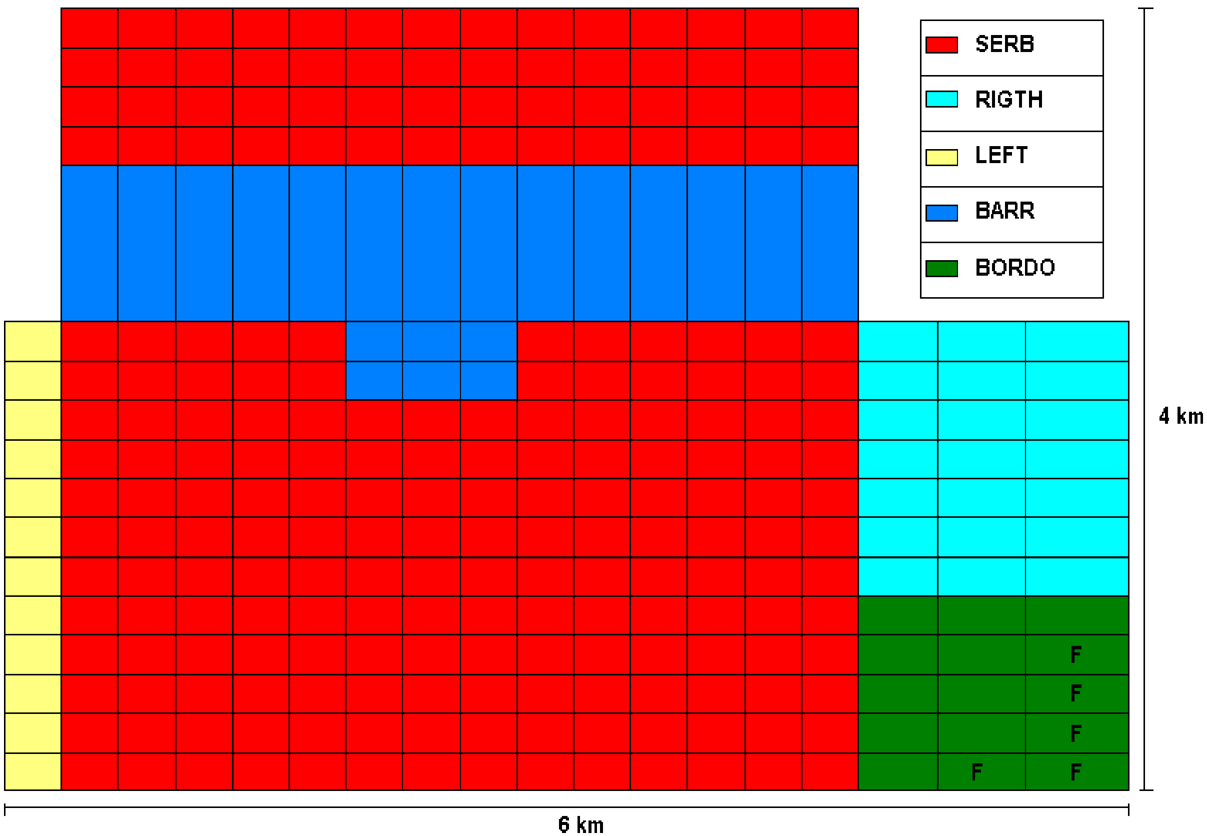
Geothermal Modelling

Natural State Model

log T and P



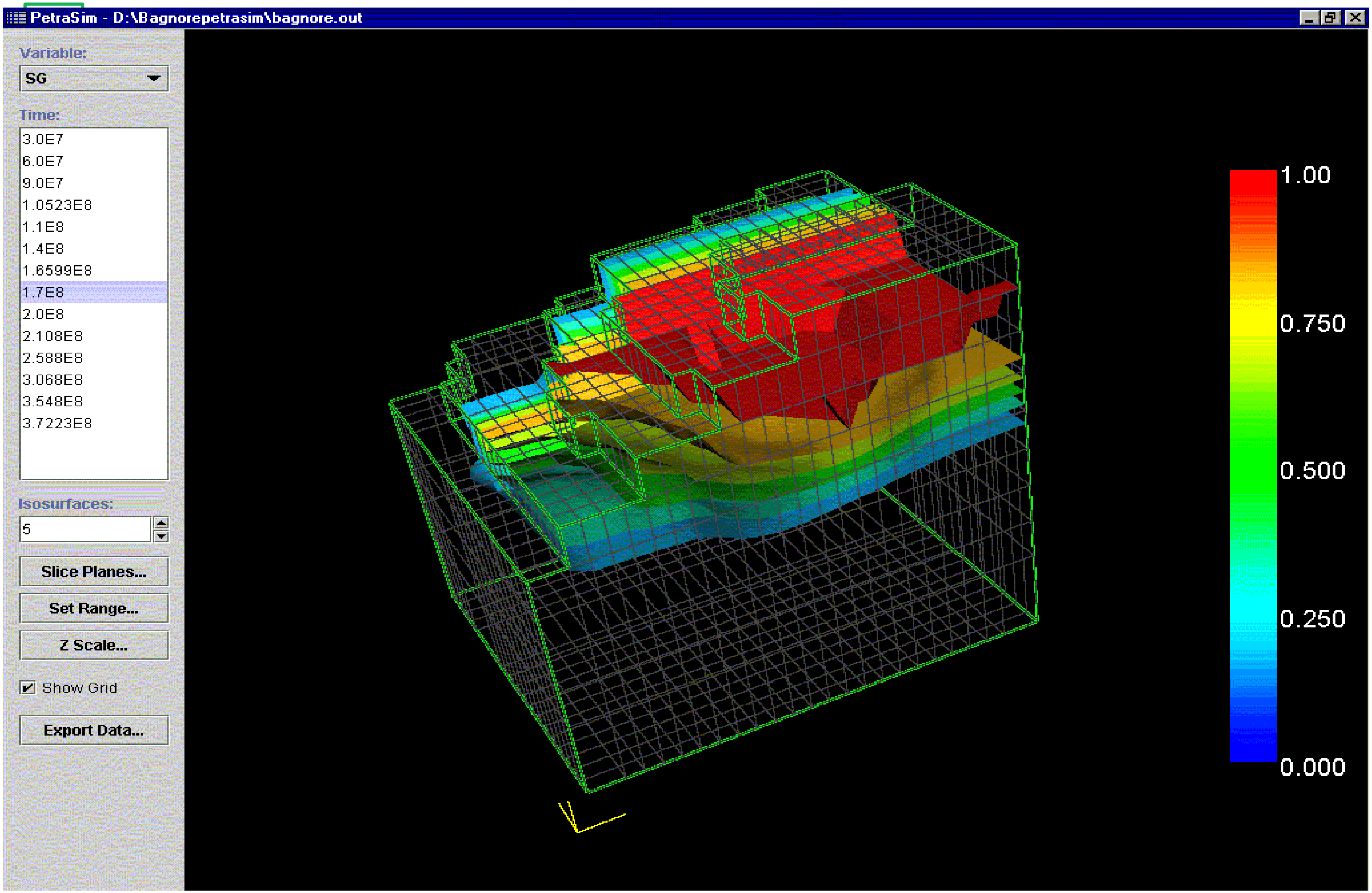
Geothermal Modelling



Modeling grid and
the recharge area (F)

Permeability as assumed from the drawdown analysis for radial model

Geothermal Modelling



PetraSim: TOUGH2 Basics

Thunderhead Engineering Consultants, Inc.

www.thunderheadeng.com

+1.785.770.8511

Phases and Components

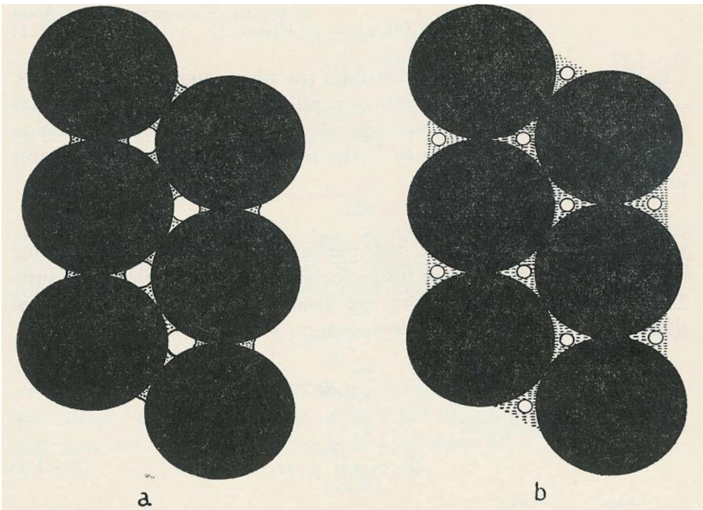
Phases

- Homogeneous continuum
- May consist of one or more chemical components
- Examples: aqueous phase, non-aqueous phase (oil), gas, solid
- In a closed system, amount of different phases may change
- Phase change usually involves substantial heat effects

Components

- Chemical species
- Can be present in several different phases
- Examples: H_2O, NaCl, CO_2
- Distribution of components in phases determined by chemistry
- All components in a phase flow together
- In a closed system, components are conserved.

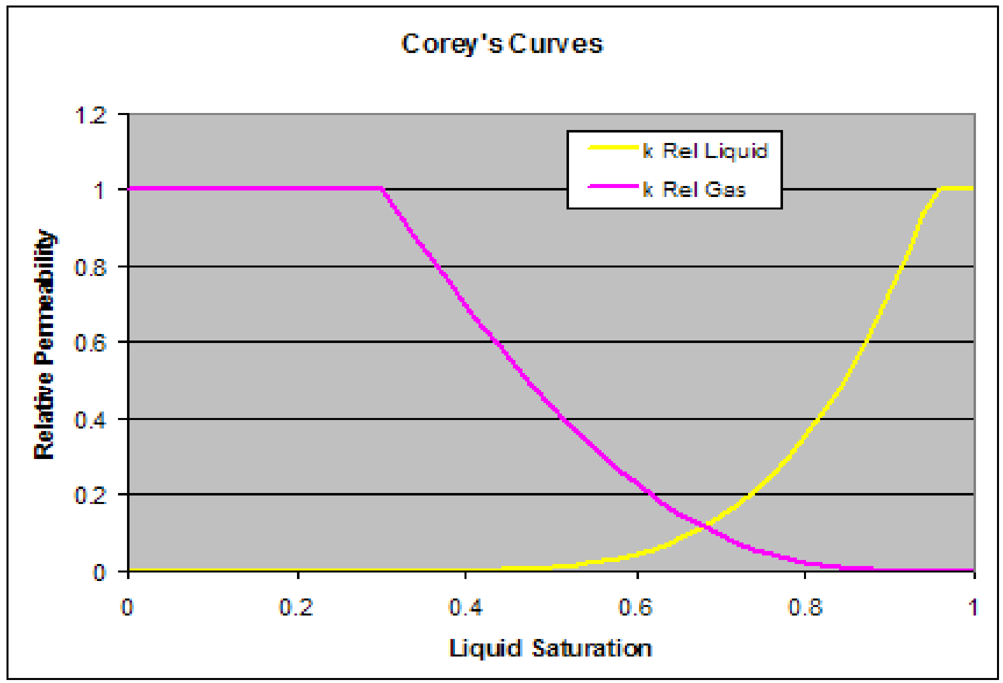
Relative Permeability



Saturation regime: The porous medium is completely saturated with one phase.

Pendular regime (a): One phase occurs in the form of pendular bodies that do not touch each other so that there is no possibility of flow for that phase.

Fenicular regime (b): The porous medium exhibits an intermediate saturation with both phases.



Implications for Solution

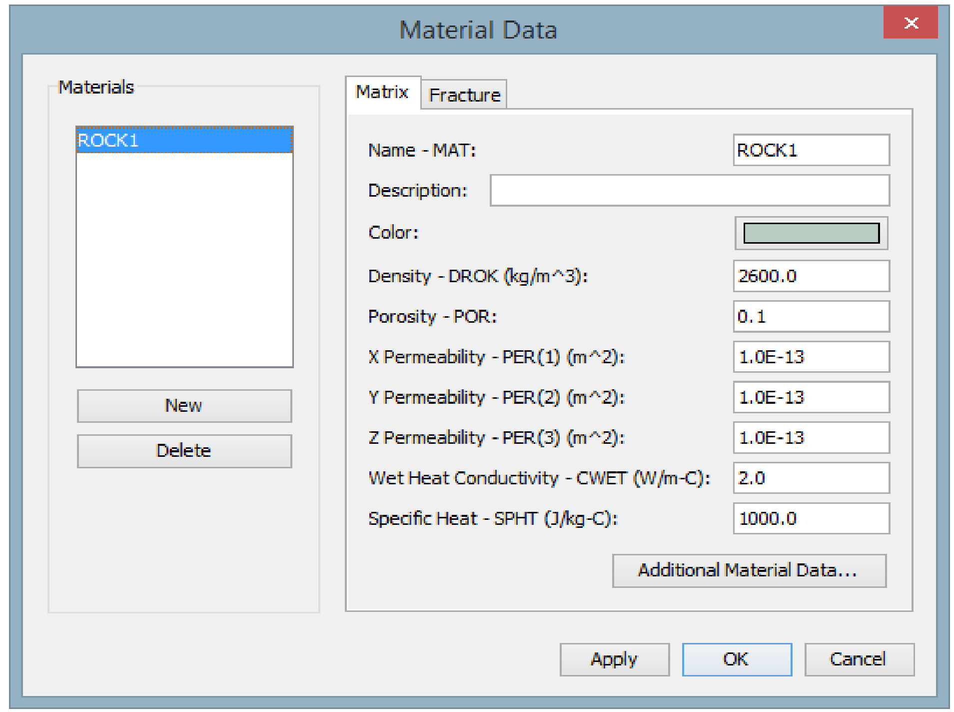
- Must start with reasonable physical assumptions
- Getting correct initial conditions often requires a steady-state solution
- In our experience, it is rare to find an error in TOUGH2, but getting solutions can require several iterations

Equations of State

- EOS1 – Water, water with tracer
- EOS2 – Water, CO_2
- EOS3 – Water, air
- EOS4 – Water, air (vapor pressure lowering)
- EOS5 – Water, hydrogen
- EOS7 – Water, brine, air
- EOS7R – Water, brine, air, radionuclides
- EOS8 – Water, “dead” oil, gas
- EOS9 – Saturated/unsaturated water flow

Materials

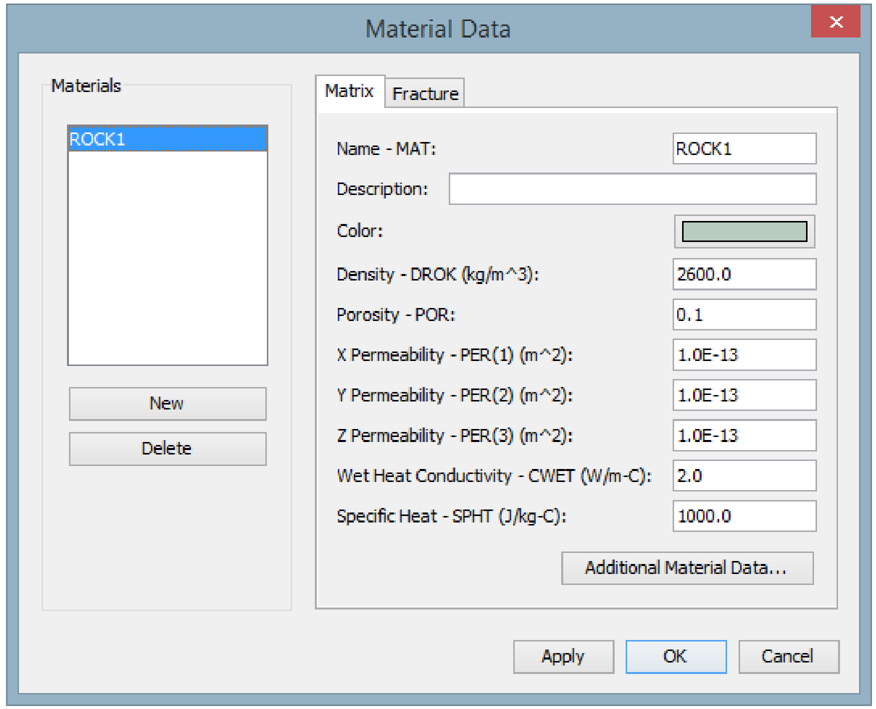
- Materials are used to define the permeability and other properties in an analysis.
- Each cell is associated with a material.
- Information stored in this Material Editor are listed in the ROCKS section of a TOUGH2 input file.



Materials

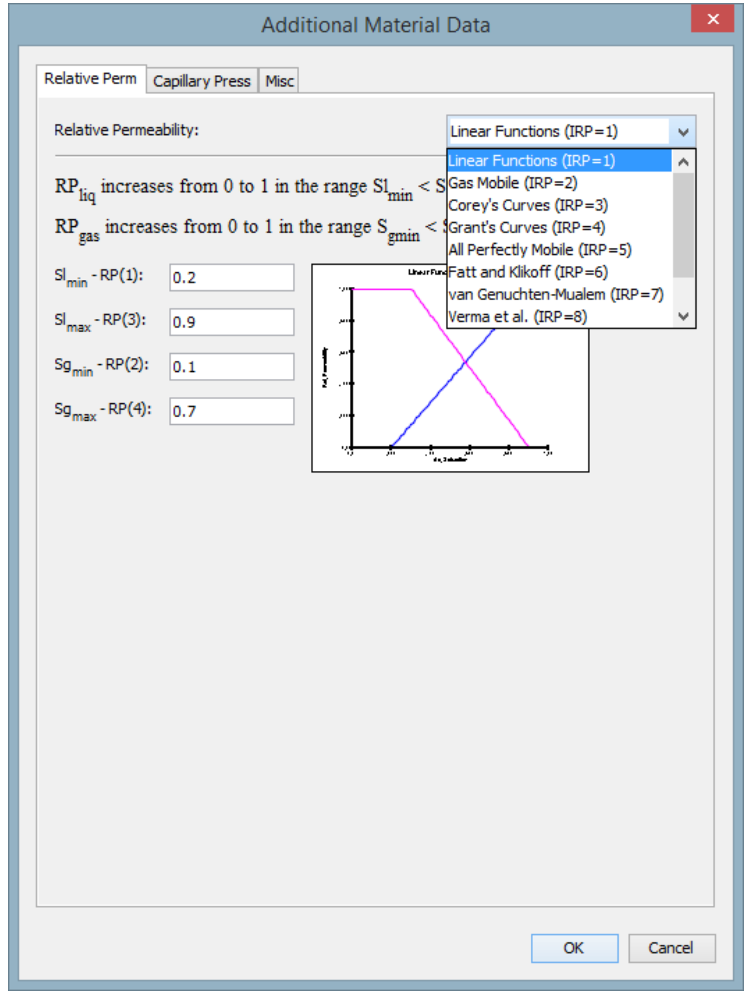
Parameters include:

- **Name** – limited to 5 characters
- **Description** - A longer description for user clarity.
- **Color** – used for display
- **Rock Density (kg/m3)**
- **Porosity**
- **X, Y, and Z Permeability** - only define 1 value for xy direction when working with polygonal mesh
- **Wet Heat Conductivity**
- **Specific Heat**
- **Relative Permeability**
- **Capillary Pressure**
- A few others



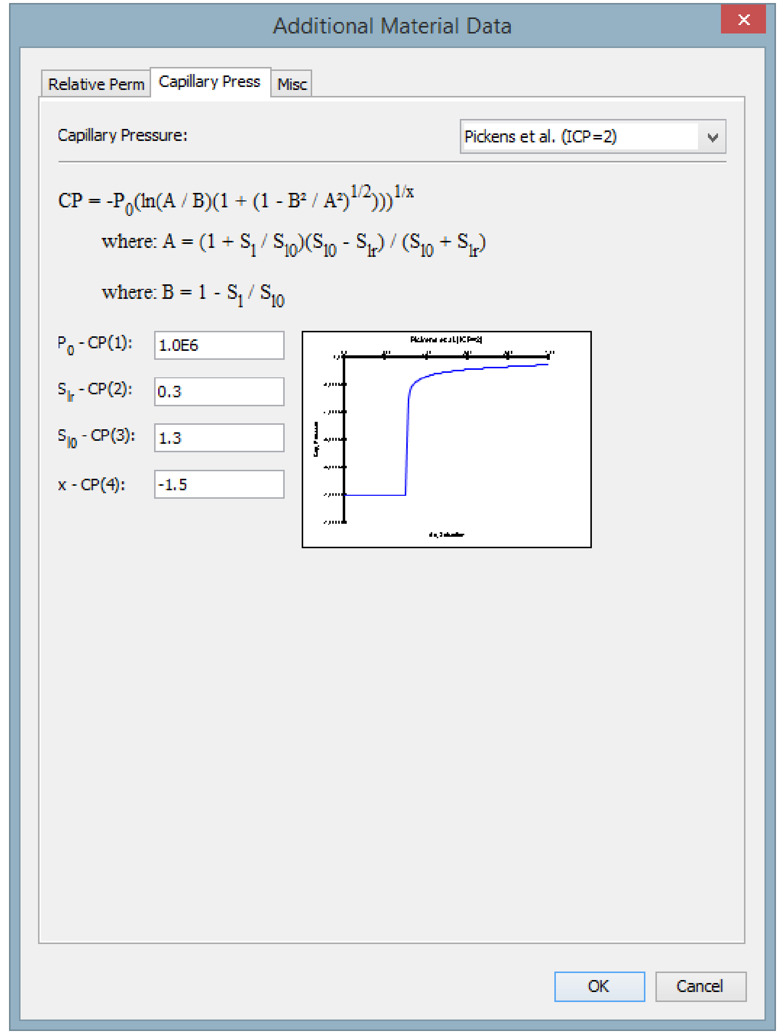
Relative Permeability

- Accessed through the Additional Material Data button.
- You select the preferred RP function and enter desired parameters.
- Plot displays the curves (gas in magenta and blue is liquid)
- Curves can drastically affect model results, so look in literature for accepted parameters



Capillary Pressure

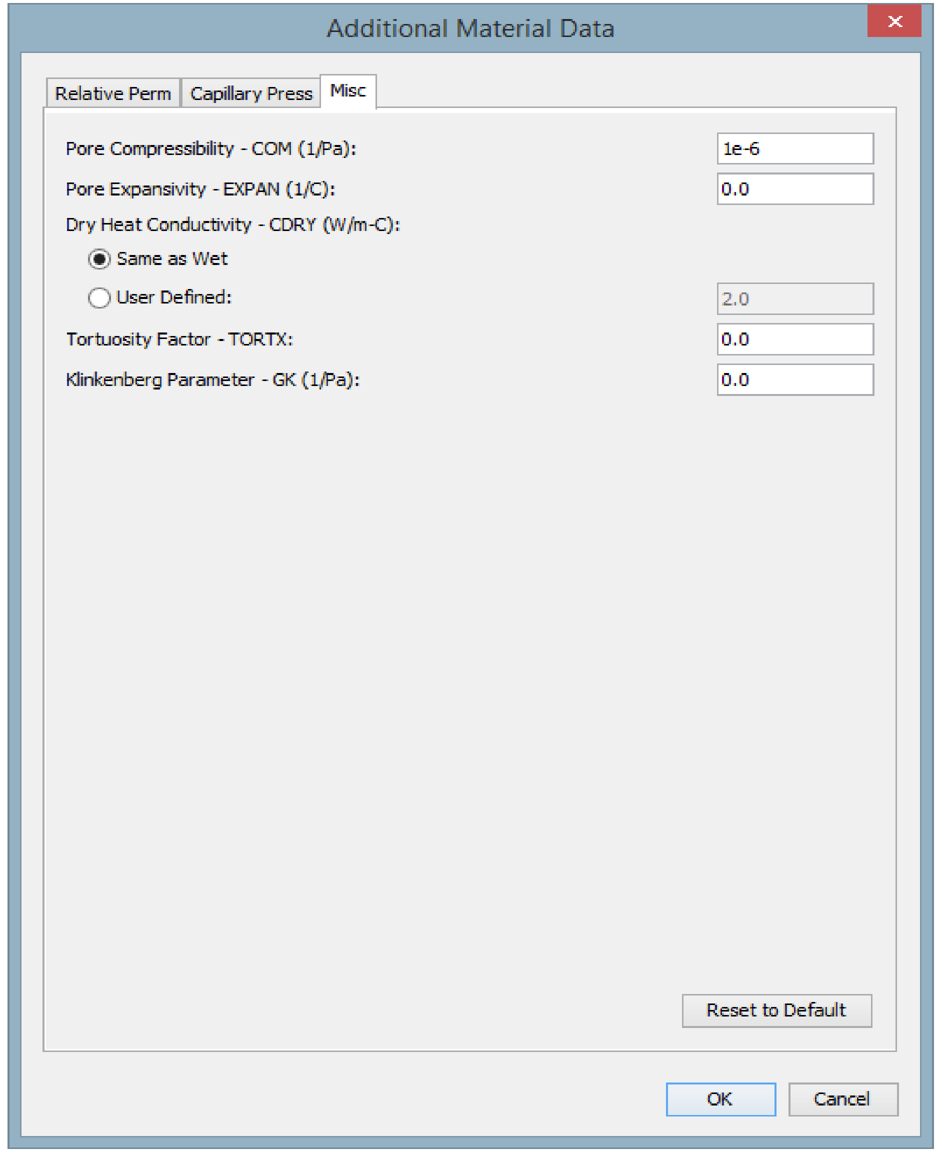
- Similar process for Capillary Pressure



Materials

Parameters include:

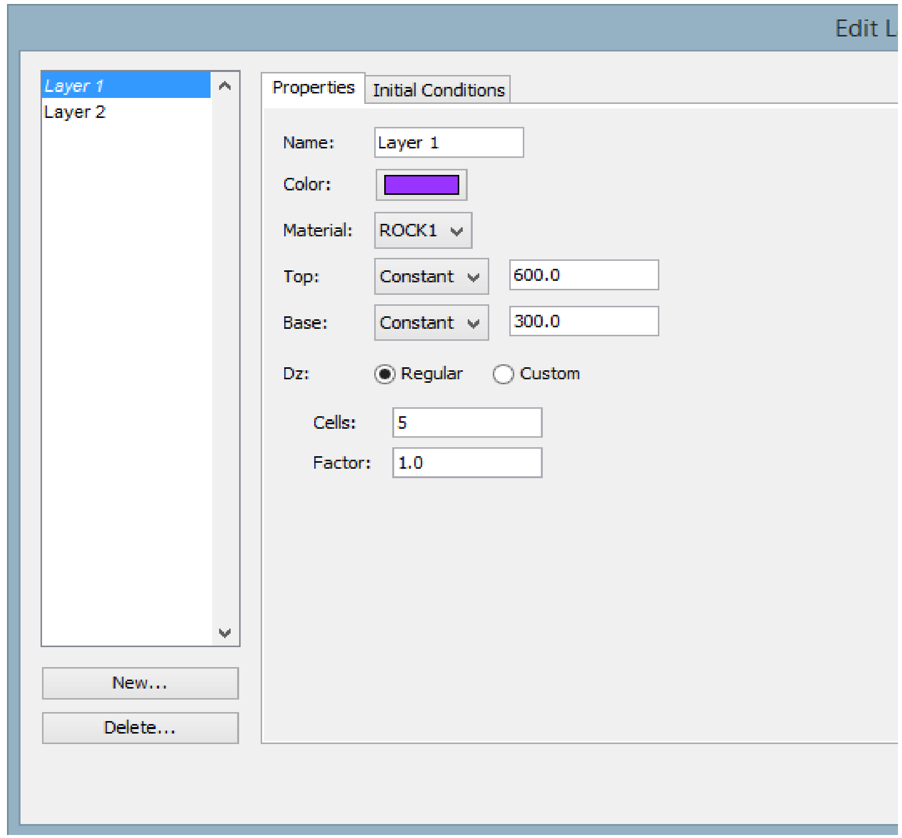
- **Pore Compressibility** – defines how the pore volume changes as a function of pressure. This can be important during injection.
- **Pore Expansivity -** defines how the pore volume changes with temperature.
- **Dry Heat Conductivity -** used with the wet heat conductivity to change the thermal conductivity of the rock.
- **Tortuosity Factor –** related to diffusion, details in the TOUGH2 manual
- **Klinkenberg Parameter** – related to gas phase permeability, details in the TOUGH2 manual



Materials

Materials can be assigned to:

- Layers (through the Layer Manager)
- Regions (right click on the Region in the data tree)
- Cells or groups of Cells (selected through the 3D View)
- Or…

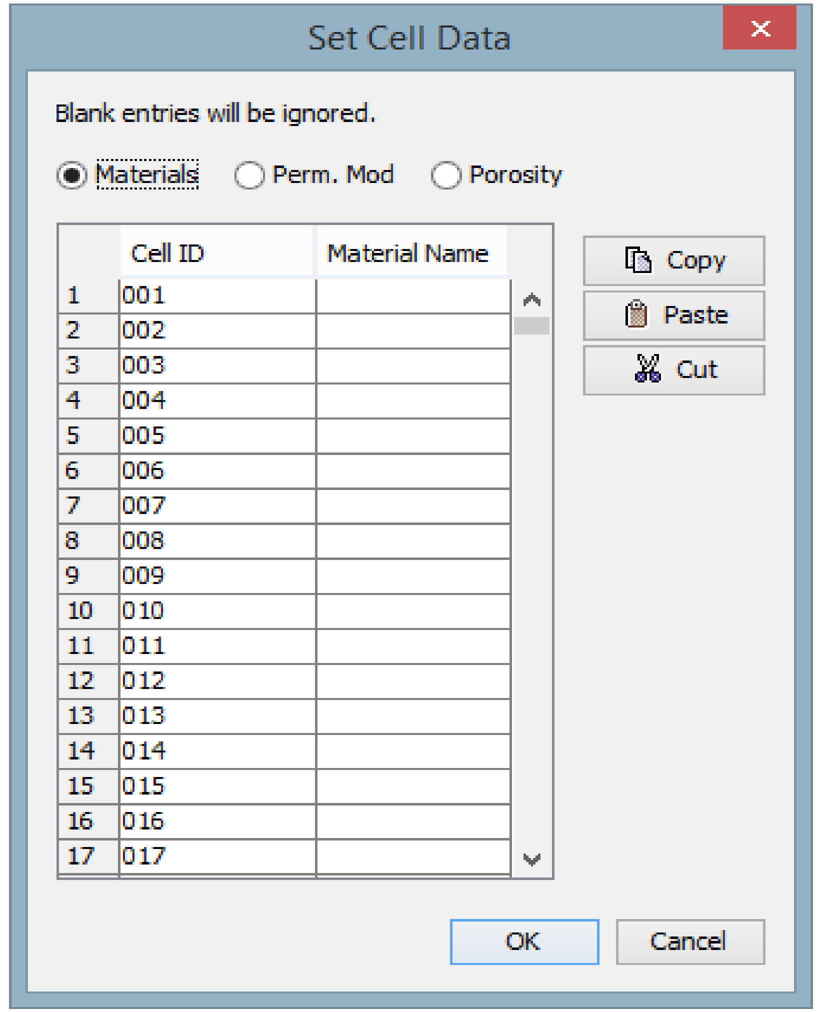


Materials

Set cell data option available through the Model menu, and used to import materials based on an external geological or geostatistical model.

The best way to approach this would be to:

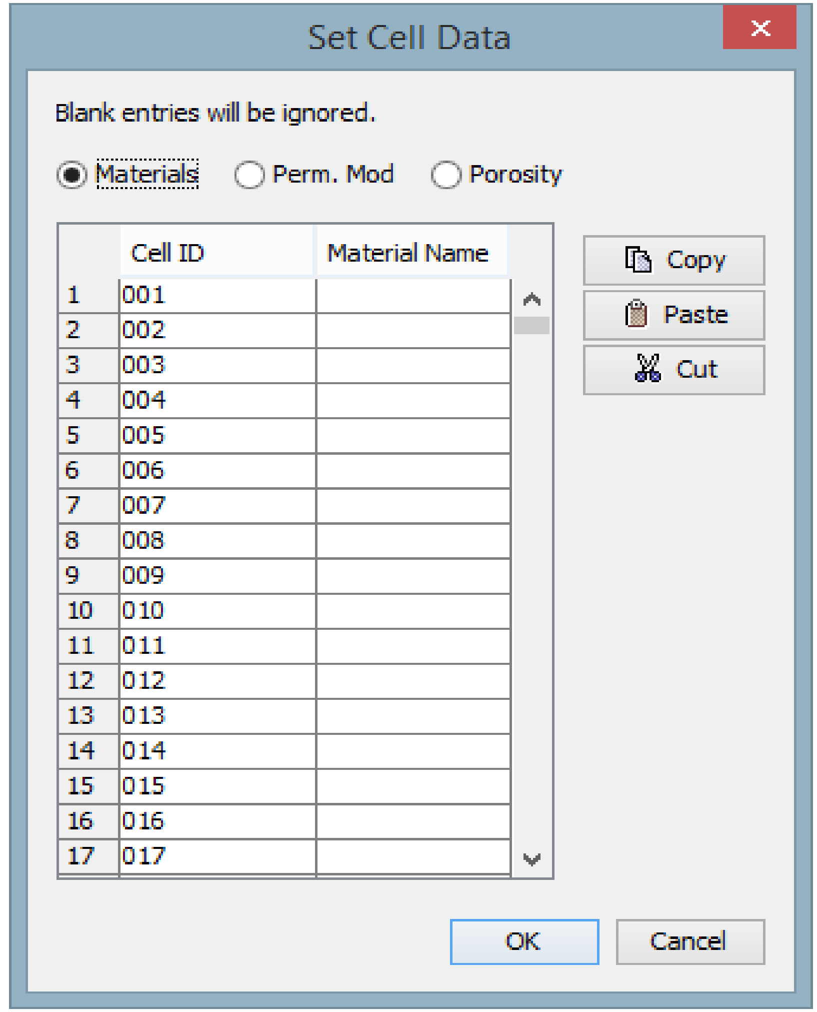
- Export a list of XYZ values for each grid cell through the File menu.
- Use these xyz values to determine the material at each cell, and copy and paste the list into the Set Cell Data window.



Materials

PetraSim also supports the following parameters:

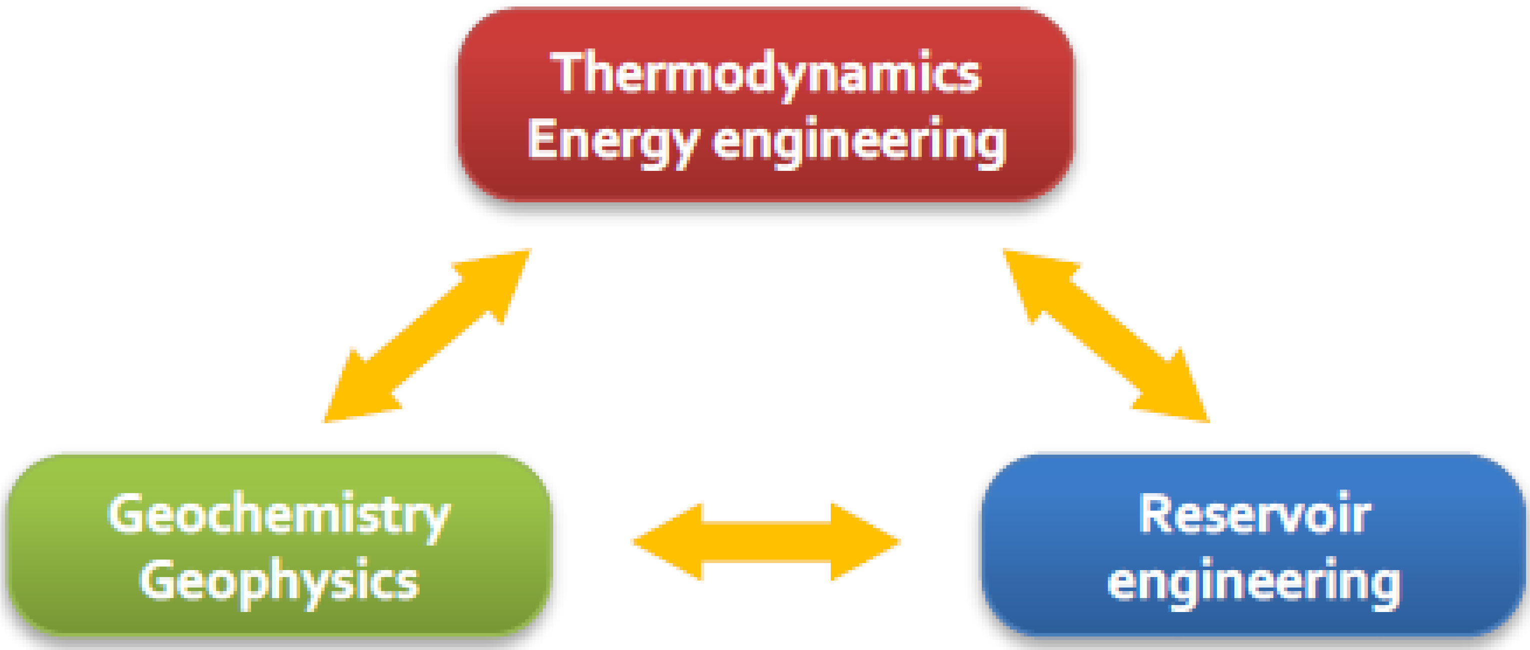
- PMX – Permeability Modifier (multiplier) in the ELEME block of the TOUGH2 input file
- PORX – Porosity in the INCON block of the TOUGH2 input file
- These can be assigned by selecting a cell or group of cells, or through the Set Cell Data window.
- Use of these parameters allow you to spatially vary porosity and permeability values without creating a huge number of material types. You will still be limited to assigning other material properties, RP and CP using the materials defined under ROCKS.



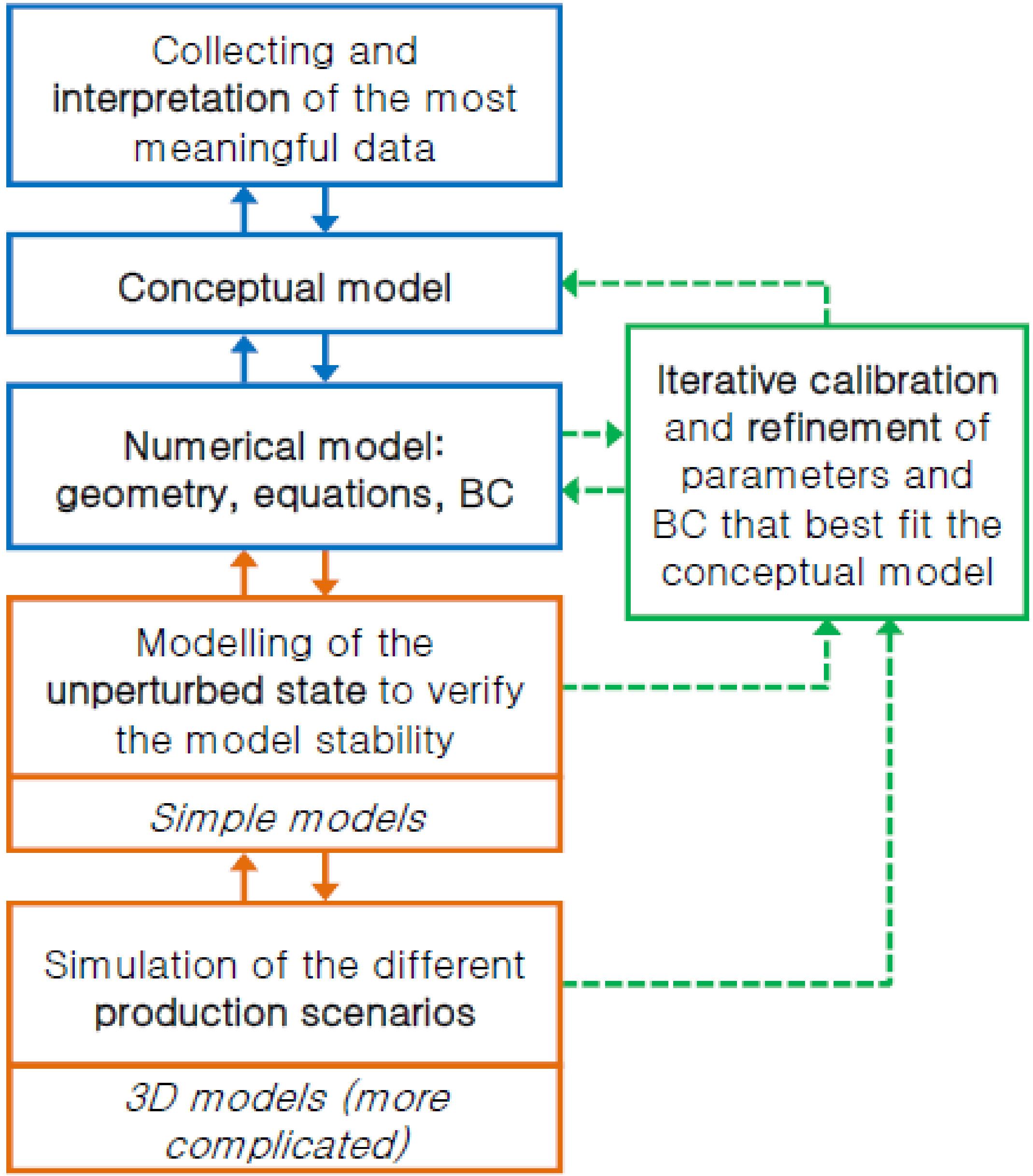
Philosophy

- Most geologic models have a *Natural State* that represents flow and heat transfer before being disturbed.
- Except in the simplest cases, do not expect to define the natural state of your analysis by specifying initial conditions.
- Any realistic model requires that you solve one or more analyses that bring to you to natural state. Then you load the natural state to start your simulation.

Philosophy


Thermodynamics
Energy engineering
Geochemistry
Geophysics
Reservoir
engineering

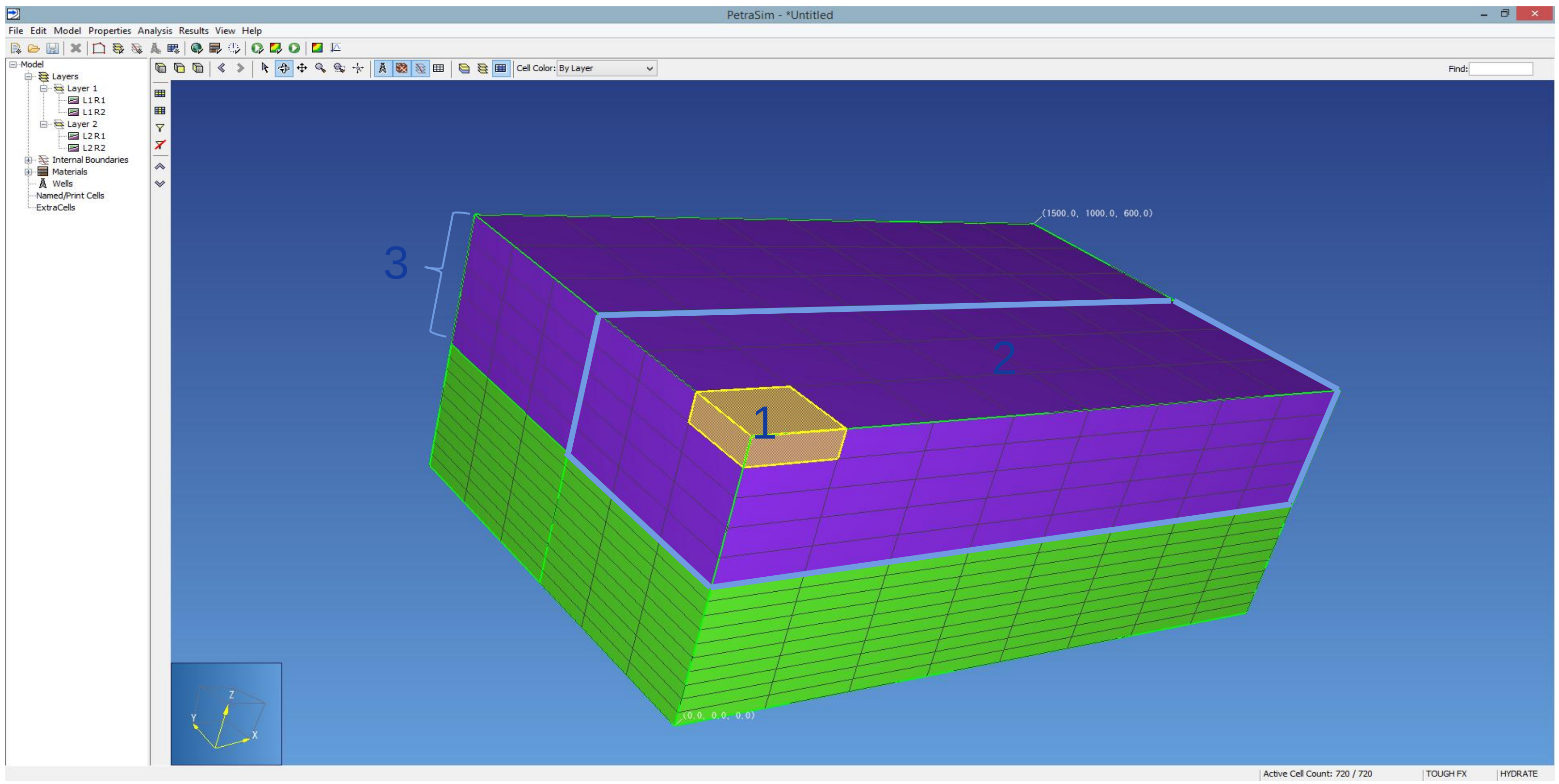
Philosophy


Collecting and interpretation of the most meaningful data
Conceptual model
Numerical model: geometry, equations, BC
Modelling of the unperturbed state to verify the model stability
Simple models
Simulation of the different production scenarios
3D models (more complicated)
Iterative calibration and refinement of parameters and BC that best fit the conceptual model

Initial Conditions

Used to define the initial state of each cell, based on the following hierarchy:

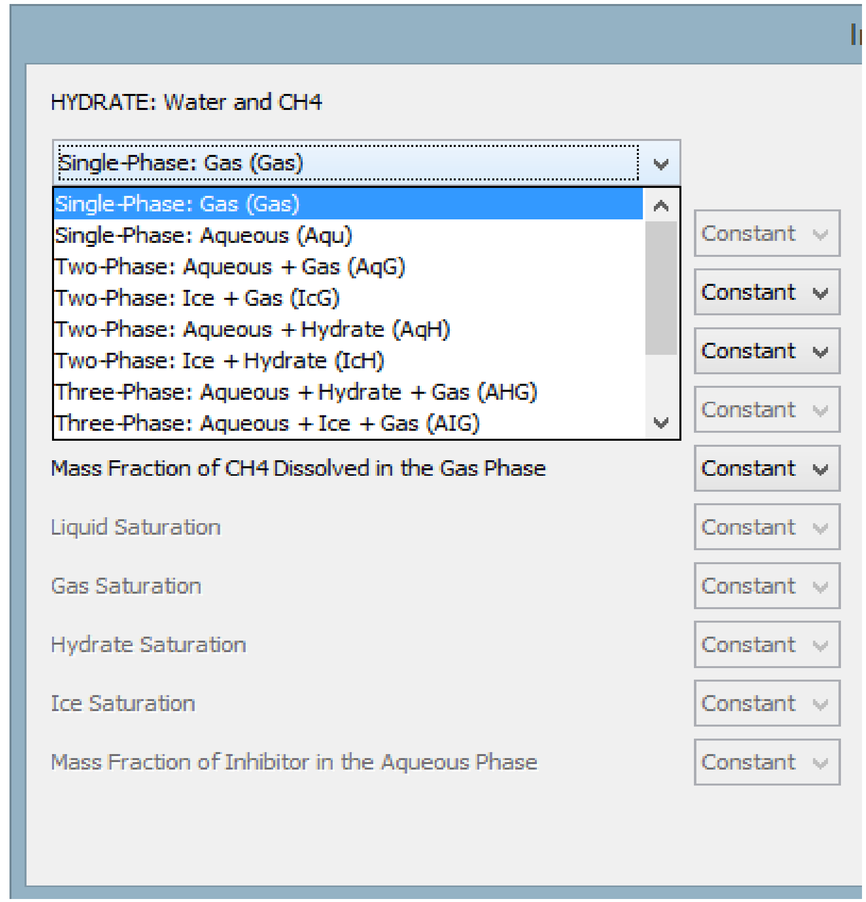
1. Cell
2. Region
3. Layer
4. Default



Initial Conditions

The specific initial conditions are different for each EOS

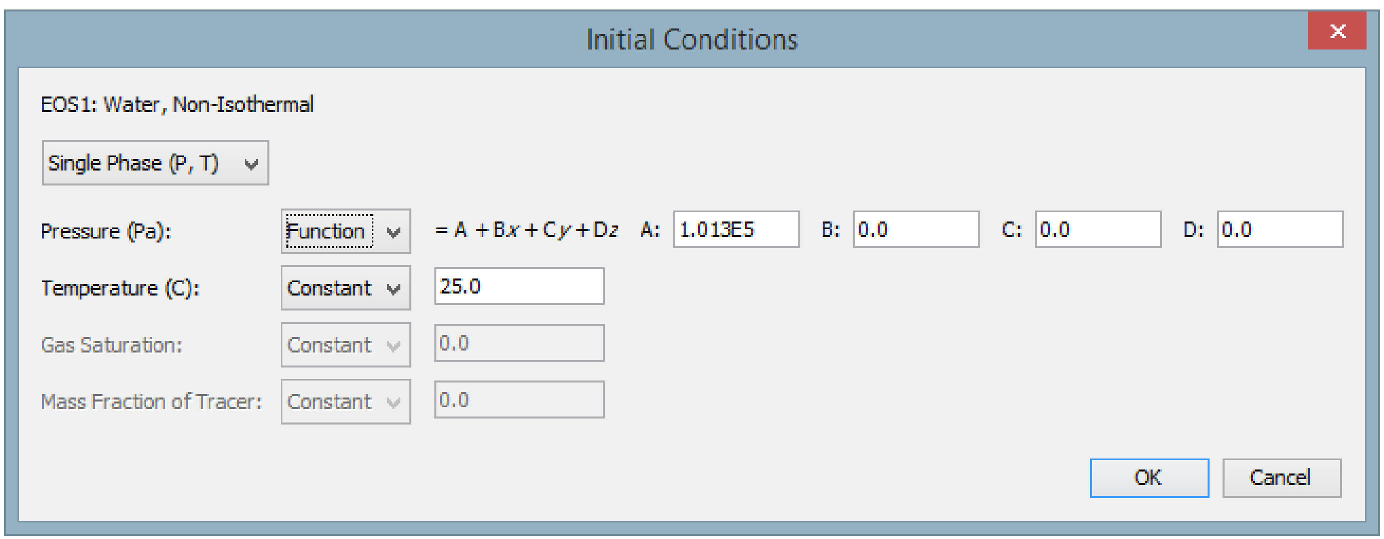
- Single, two, or multiple-phases
- Sometimes option for multiple components (CO2, NaCl, Brine, etc.)
- Only for the simplest models will the initial conditions be uniform over the model



Initial Conditions

Accessed through the Properties / Initial Conditions menu item

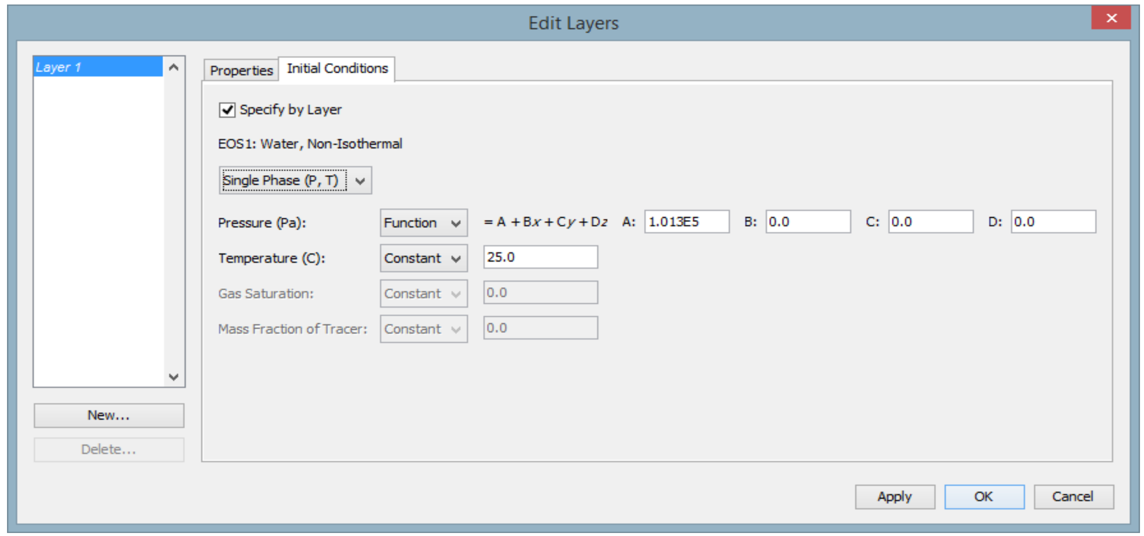
- Options are EOS specific
- Conditions can be defined as:
  - Constant
  - Function (pressure, temperature)
  - File (2D models only, not recommended for 3D)



Initial Conditions

Accessed though the Layer Manager or by right clicking on the Region

- Same definition options
  - Constant
  - Function
  - File
- Layer initial conditions over-ride Default initial conditions

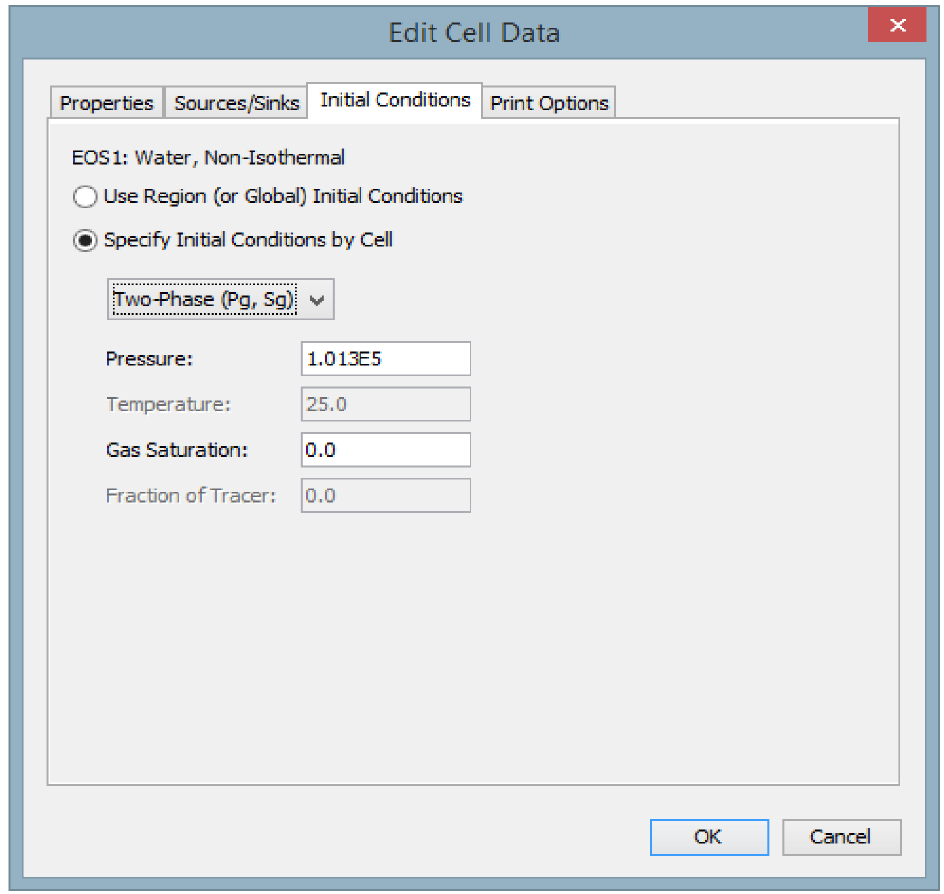


- Region initial conditions over-ride Layer (and Default) Initial Conditions

Initial Conditions

- One or more cells can be selected and assigned unique initial conditions

- When SAVE file is loaded as initial conditions, each cell is assigned unique initial conditions based on the results of the steady state run.



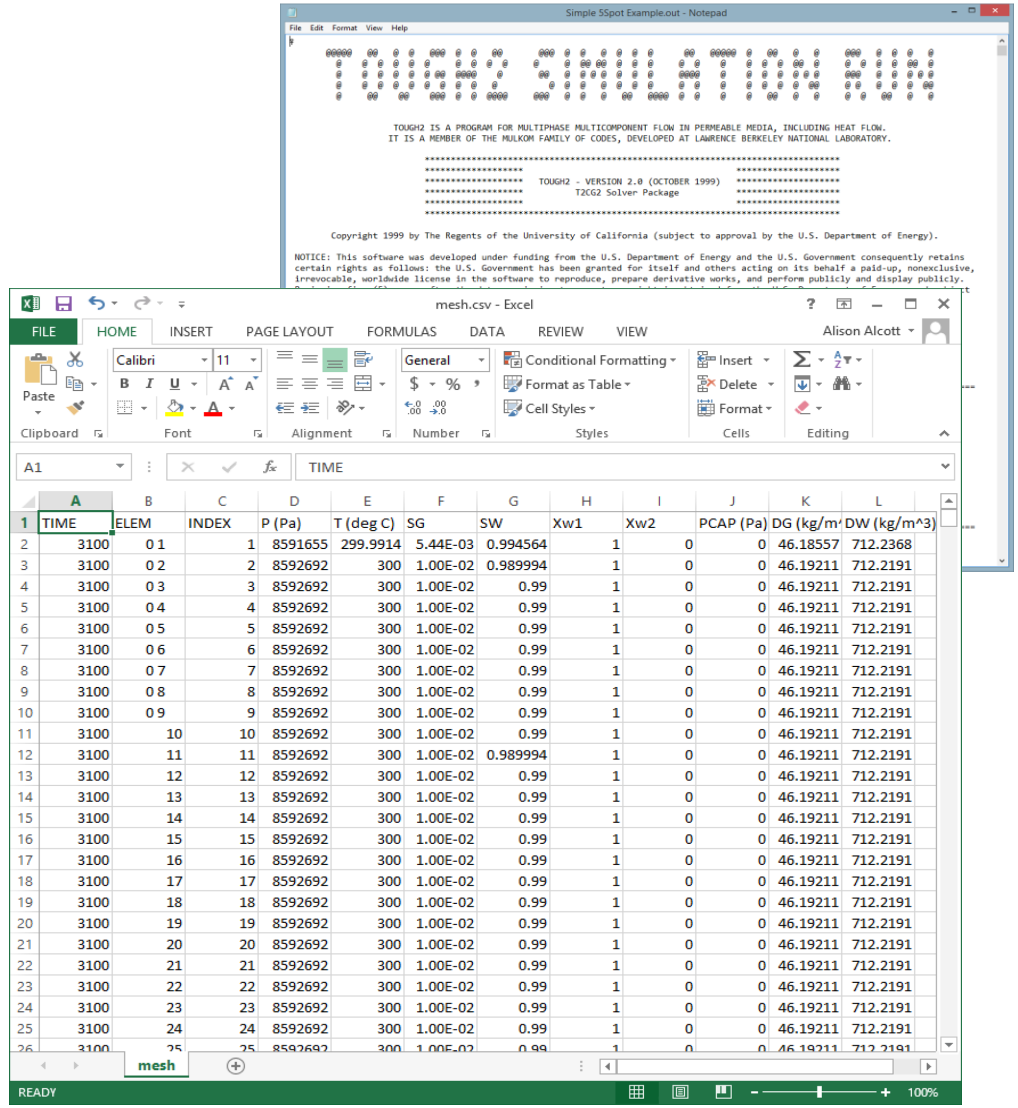
Input files

- .SIM – Binary file that includes your PetraSim model. You should only store one file in a folder.
- .DAT – TOUGH2/T2VOC/TMVOC input ASCII file
- If you are using TOUGHREACT, there will be three input files: Flow.INP, Chem.INP, Solute.INP
- There may be other input files that are EOS specific (CO2TAB, thermodynamic database, etc.)

Output files

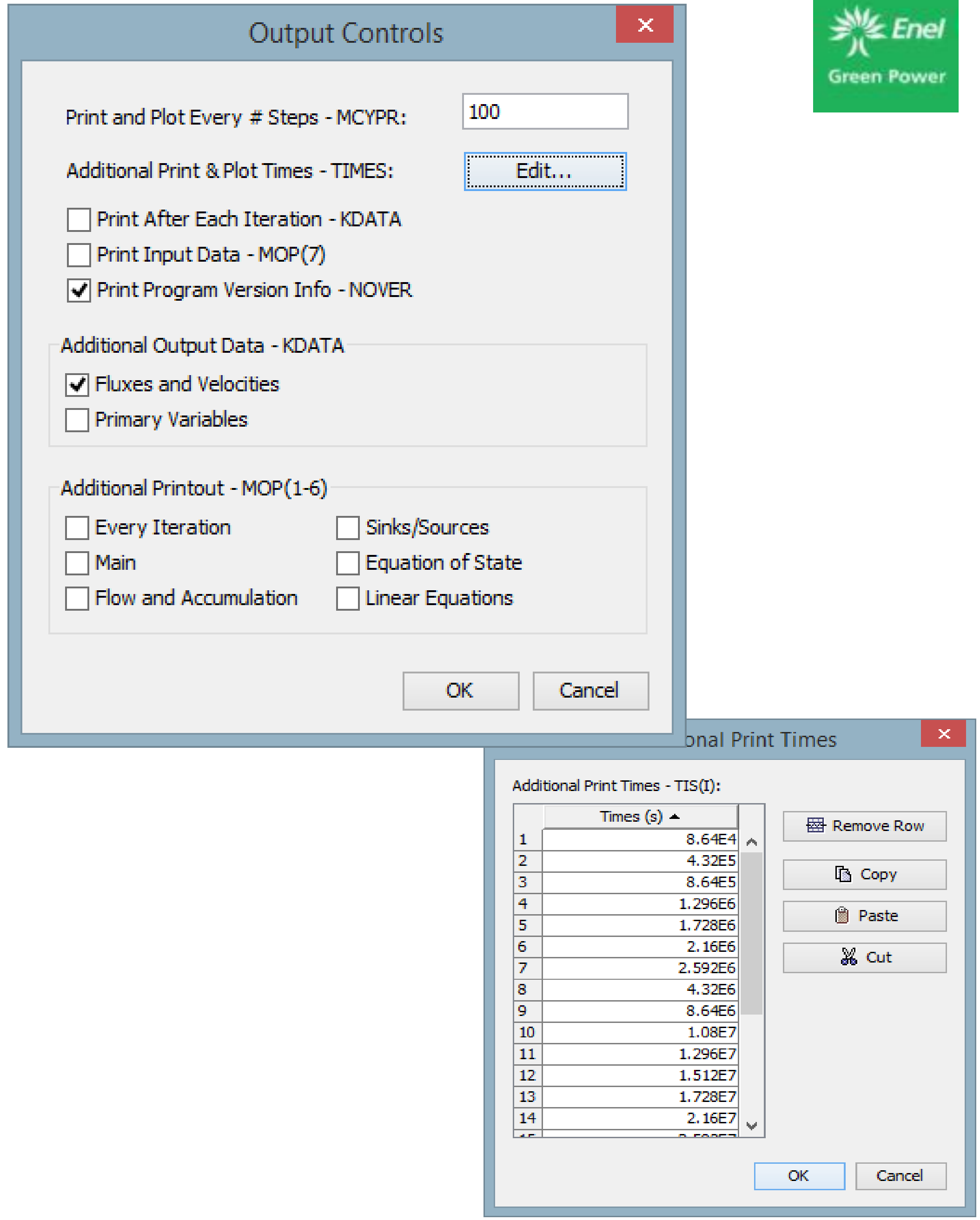
The TOUGH2 executables included with PetraSim create 2 types of files:

- TOUGH2 .OUT files – contain model results and helpful error messages for non-converging models
- .CSV files – Used for result visualization and can easily be loaded into Excel or other programs.



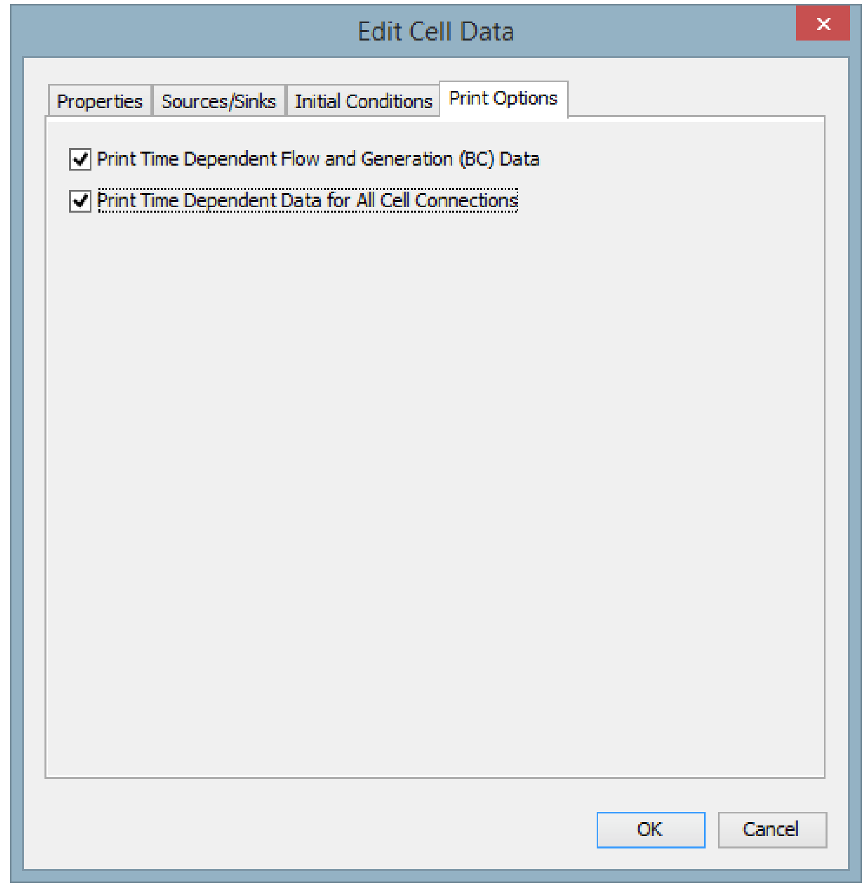
Output files

- Mesh.csv – includes output for all cells in model
- Conn.csv – includes connection information for all cells in model
- The time steps included in these files are for the solution output times only. These are established through the Analysis / Output Controls menu



Output files

- Foft.csv – output data for individual print cells
- Coft.csv – data for print connections
- Goft.csv – data for sources/sinks in the models
- This output is enabled through the Cell or Well Edit windows in PetraSim
- Includes ALL time steps

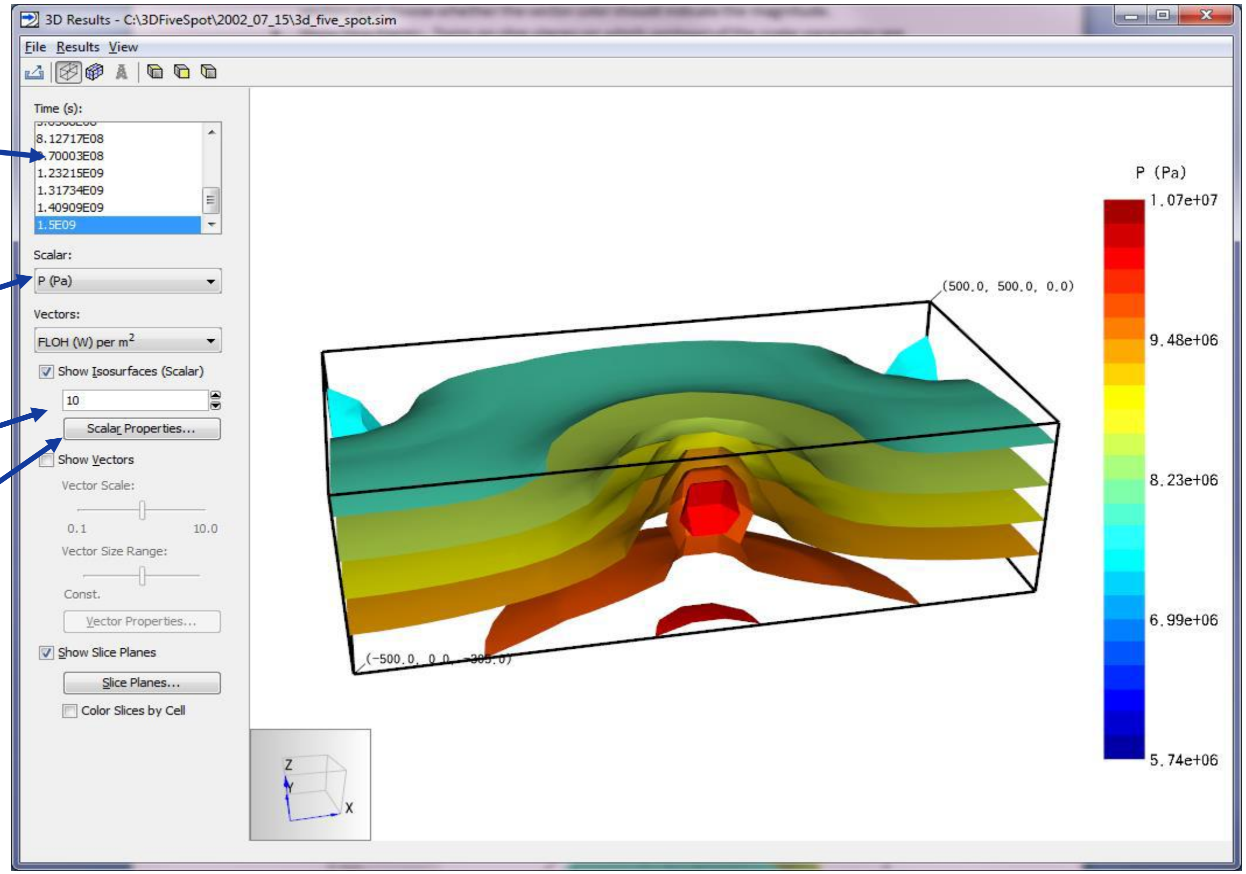


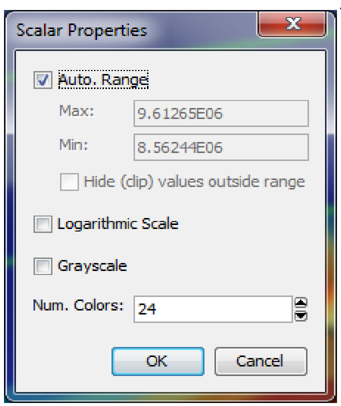
Output 3D Surfaces

Simulation Output Times

Scalar to plot in Isosurface

Isosurface Settings

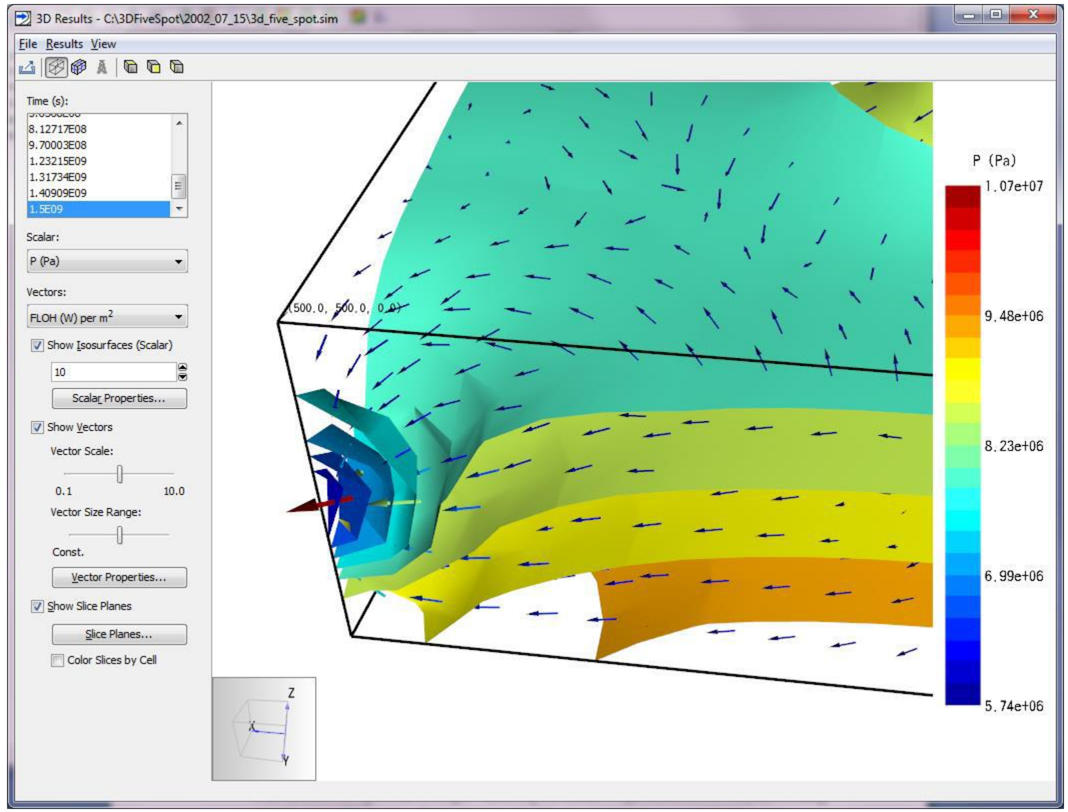


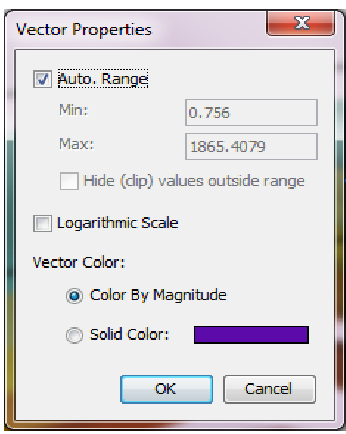


Output 3D Vectors

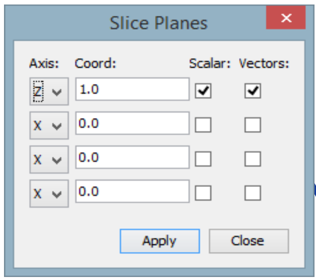
Vector plot parameters

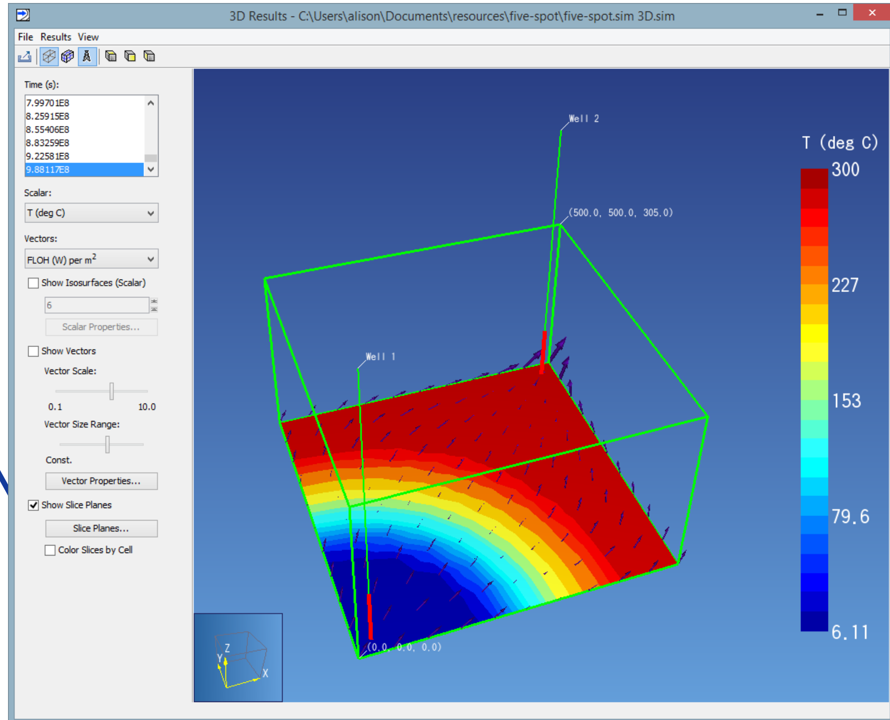
Vectors Settings

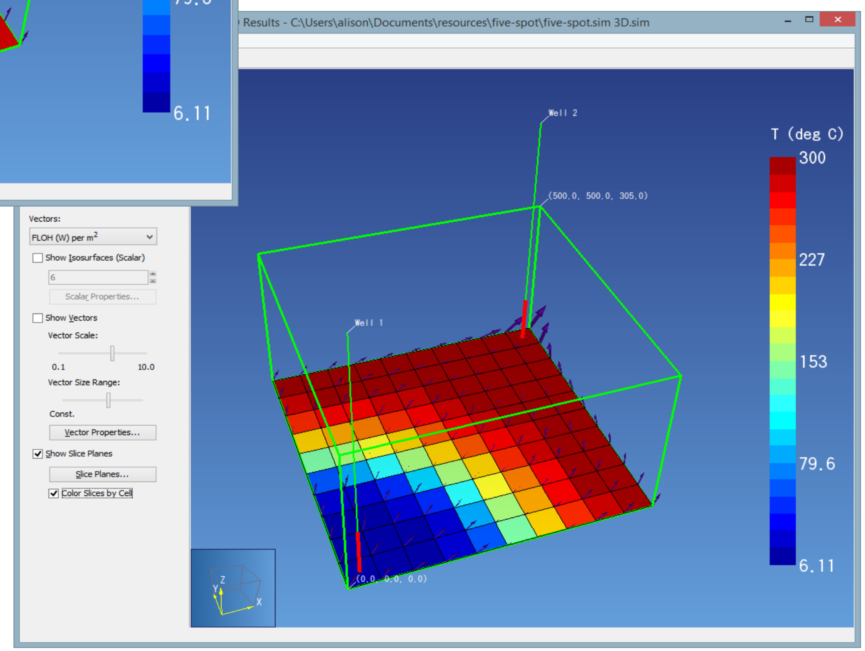




Output 2D Planes

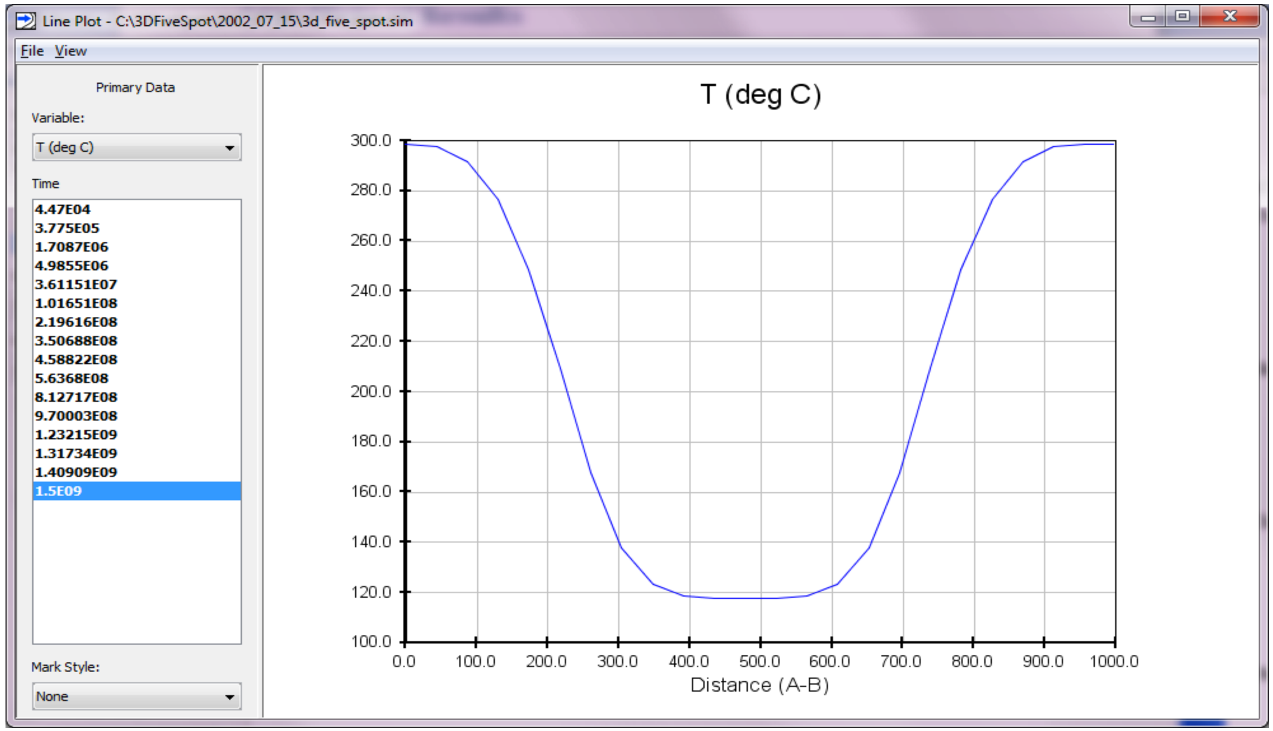






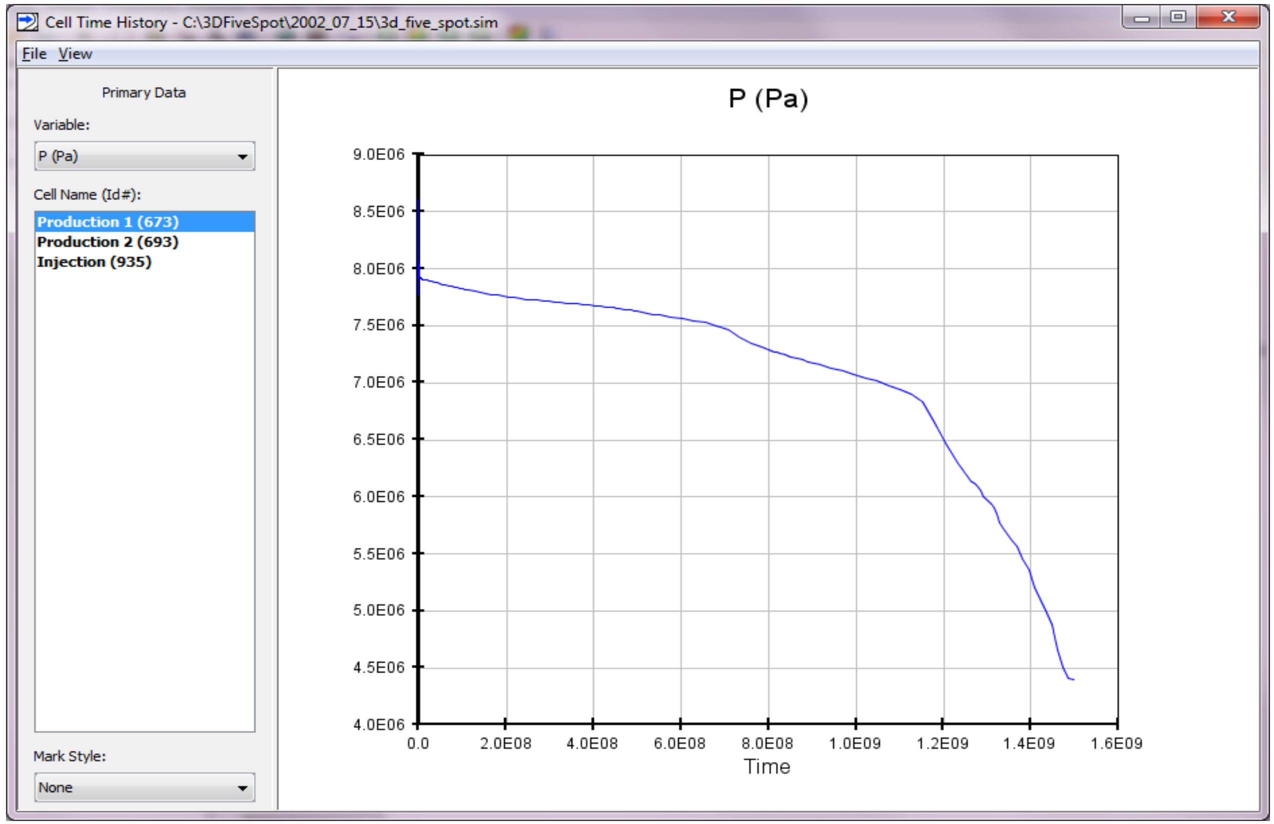
Output Line Plots

- Created through the 3D Results Window
- User enters xyz values for endpoint locations
- User chooses output time and plot variable
- Data can be exported to CSV file



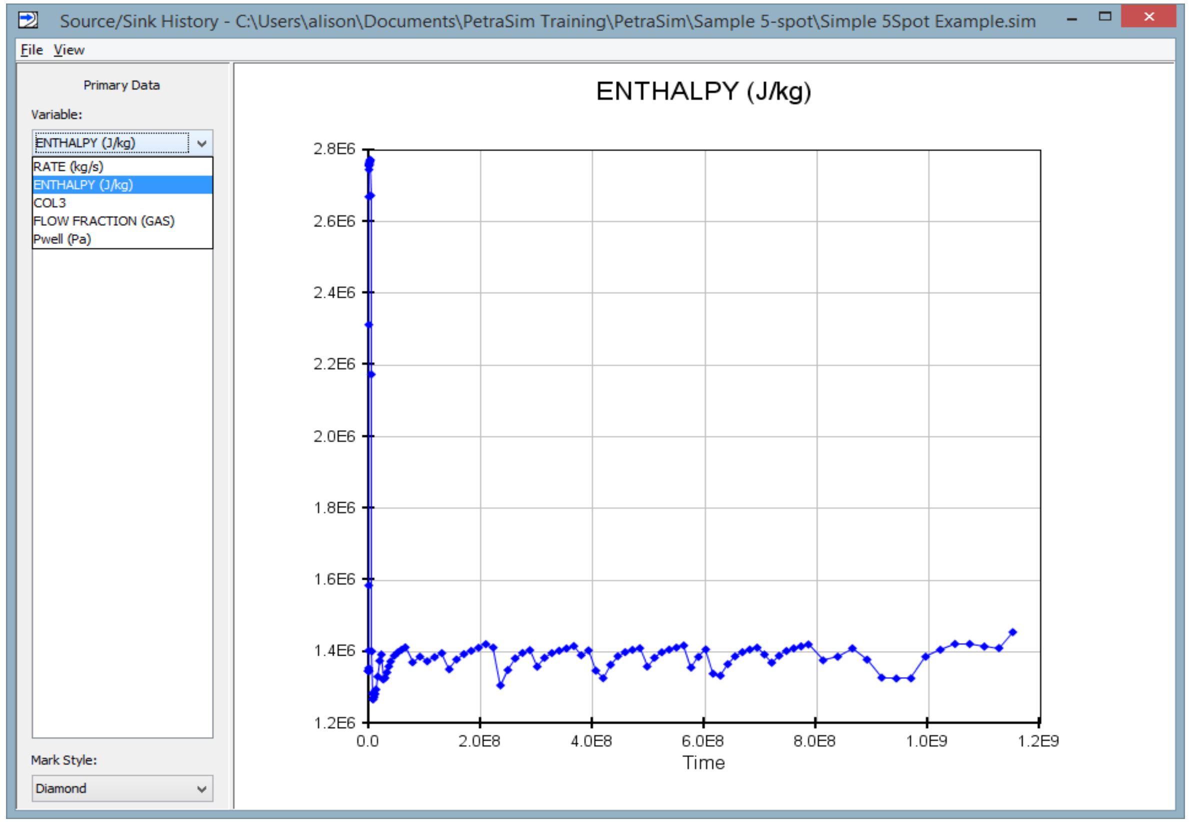
Output Time Plots

- Created through the Results menu
- Uses chooses variable for plotting and cell
- Axes are adjustable
- Data pulled from foft.csv file when available, or mesh.csv file
- Data can be exported to CSV file



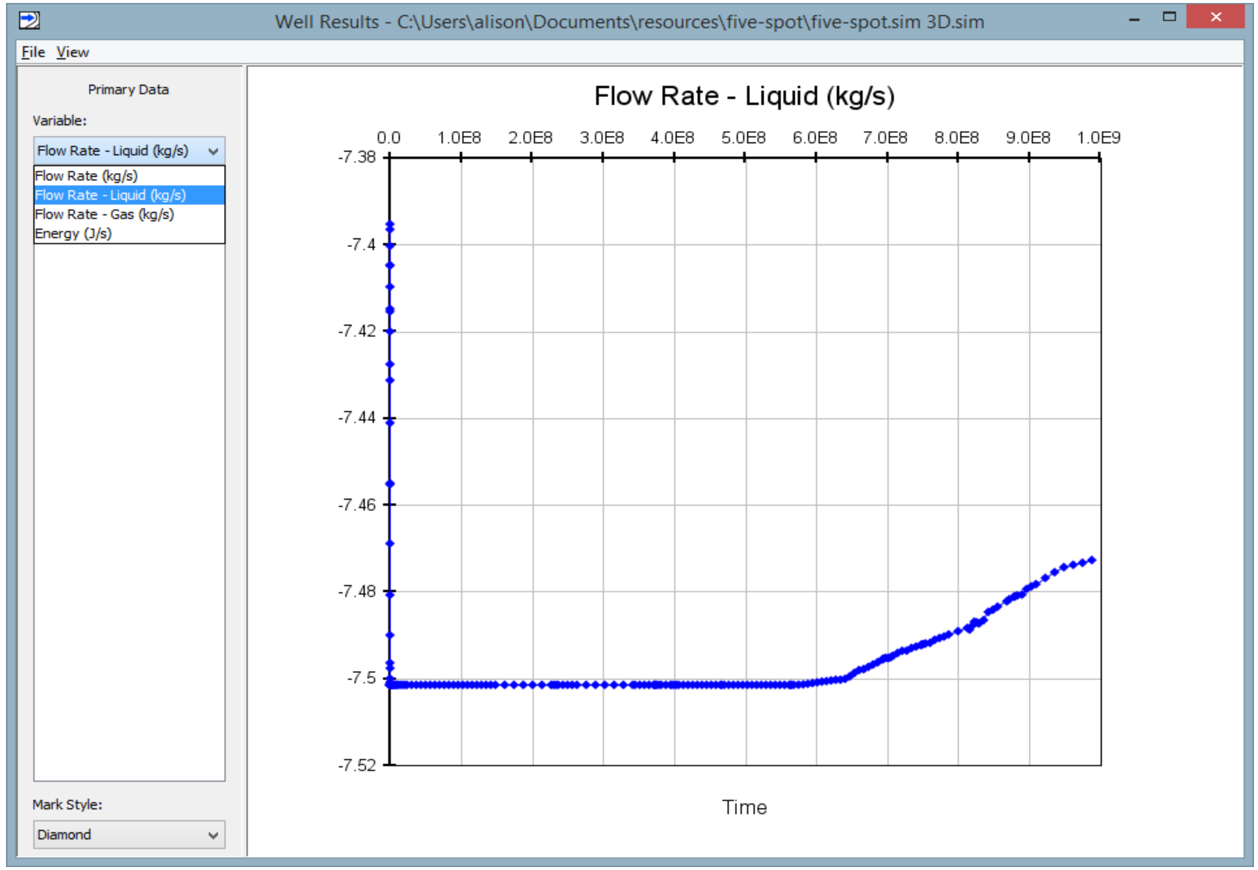
Output Source/Sink Plots

- Created through the Results menu
- Uses chooses variable for plotting and cell
- Axes are adjustable
- Values are based on connection data and are pulled from the goft.csv file
- Data can be exported to CSV file



Output Well Plots

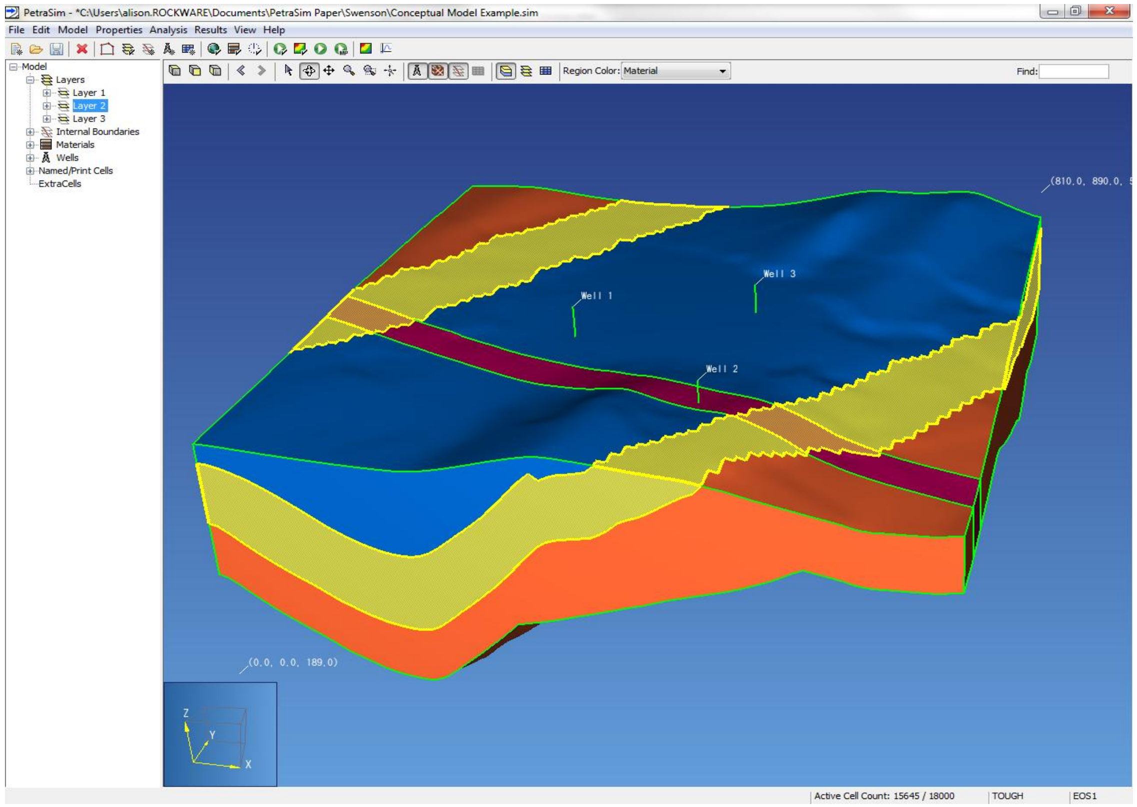
- Created through the Results menu
- Similar selection and plotting options to the other 2D plots
- Summation of data from the Goft.csv file.
- Data can be exported to CSV file
- Print option for well must be enabled



Conceptual Model

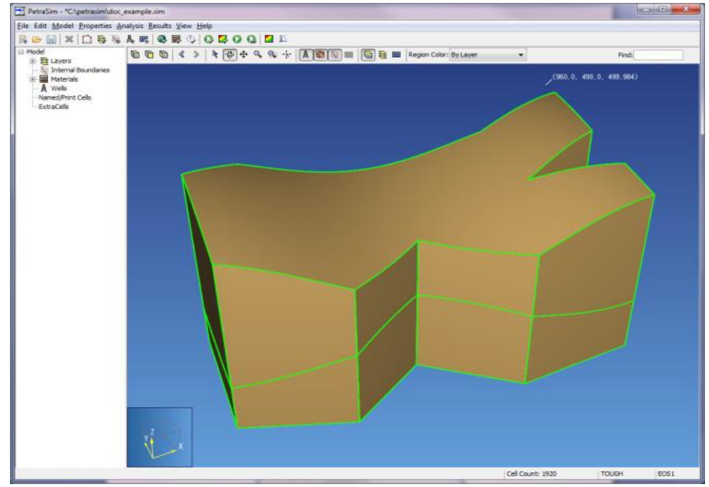
Defines the high-level features of the model and includes:

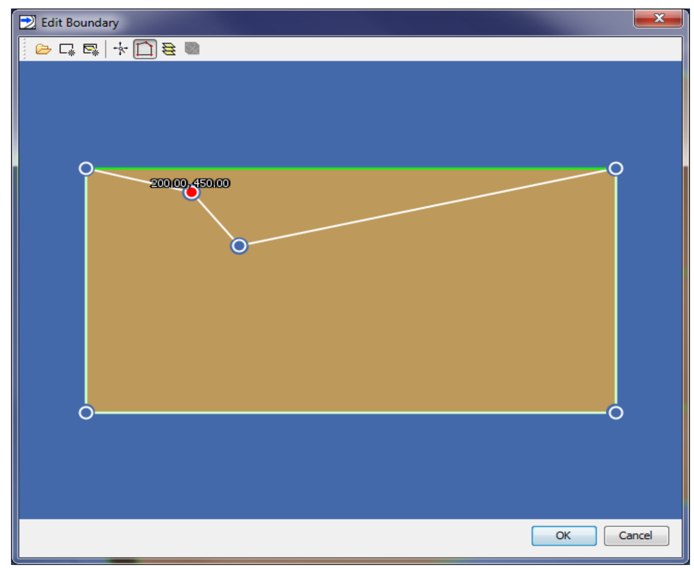
- Model Boundary
- Model Layers
- Internal Boundaries
- Regions
- Wells



Conceptual Model: Boundary

- Is a 2D polygon.
- Can be any shape (concave or convex). Default is a rectangle.
- Accessed through the Boundary Edit item under the Model menu.
- Boundaries can be drawn by hand or can be imported from a list of xy values.





Conceptual Model: Wells

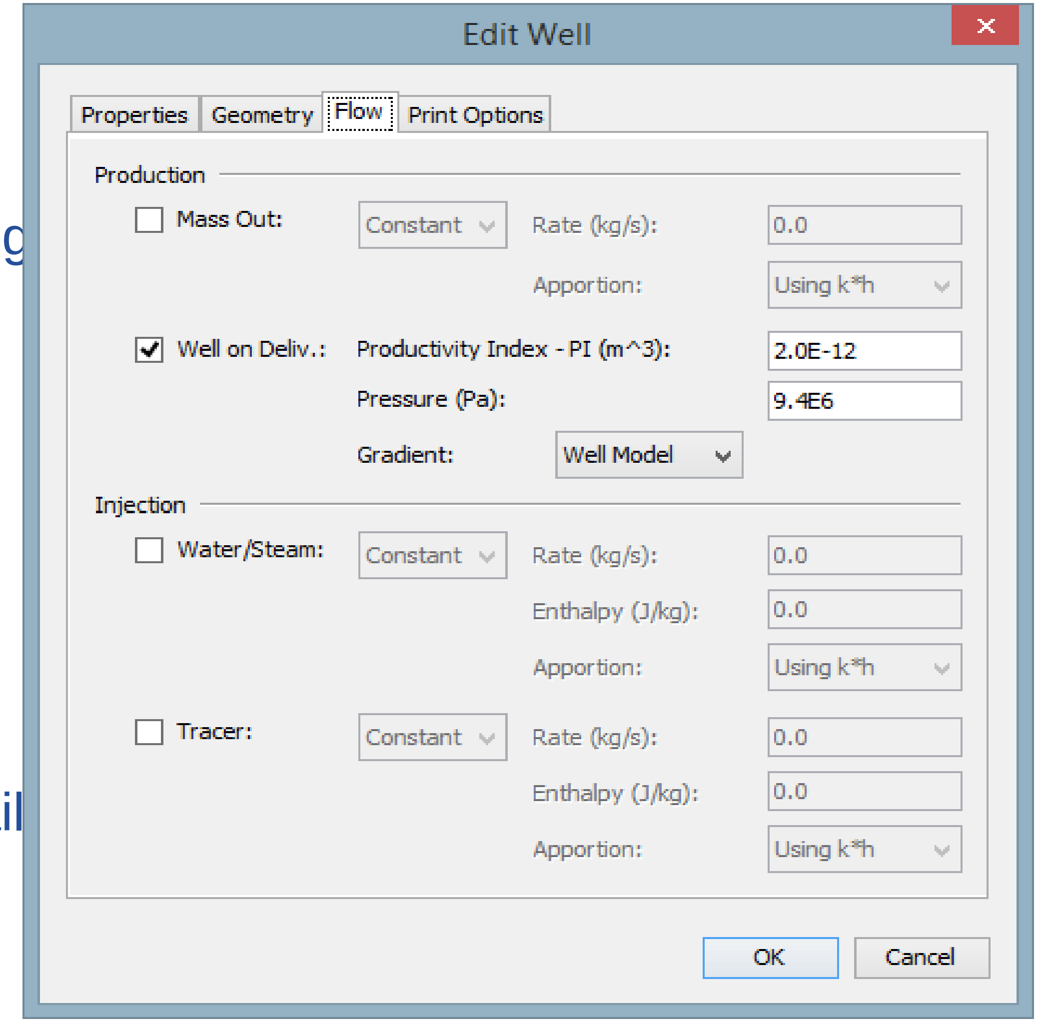
- PetraSim provides a basic option to define wells as geometric objects (lines in 3D space).
- Injection or production options are assigned to the well and PetraSim handles the details of identifying the cells that are intersected by the well and applying the appropriate boundary conditions to each cell.
- This is not a true coupled well model! It is a means of identifying the cells that intersect a well and creating the individual sources/sinks for each cell.
- It also provides a way to label and display wells.

Conceptual Model: Wells

Well definition options include:

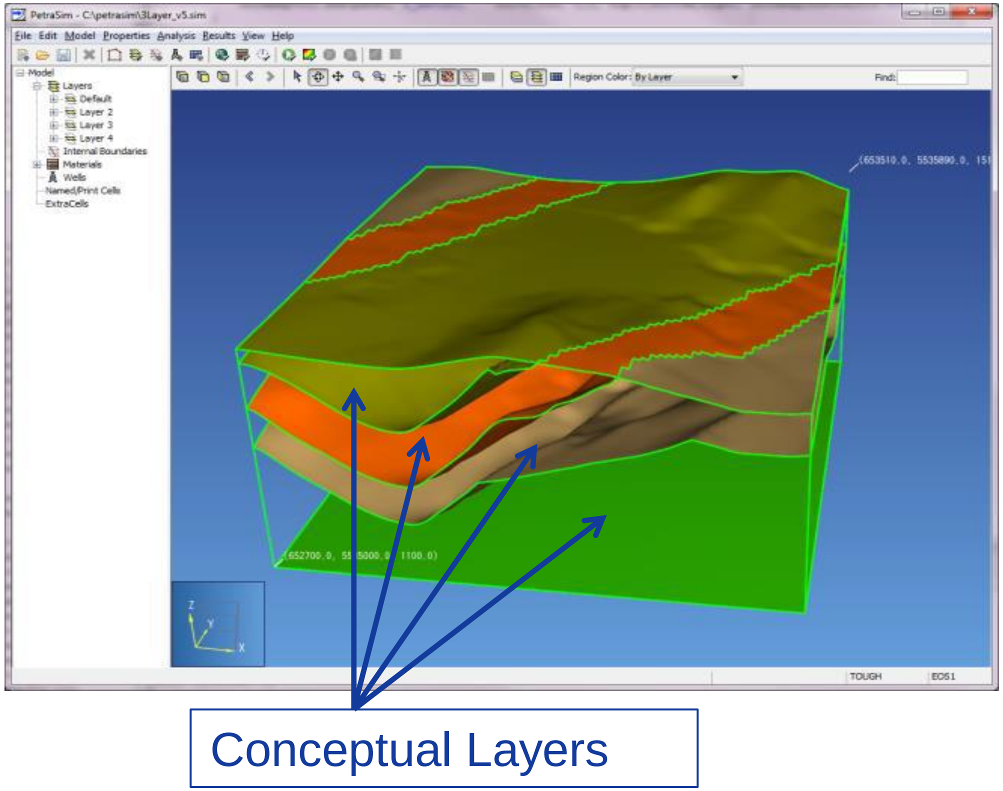
- Location – XY coordinates along the well trace
- Geometry – Top and base elevation of completion interval
- Flow – Injection/Production options
- Print options

Wells will be covered in more detail later in the course!



Conceptual Model: Layers

- PetraSim allows the user to define Layers and Regions as high level geometric entities, independent of the grid.
- Layers can be used to control material properties, initial physical and chemical conditions and the spacing of cells in the z direction.

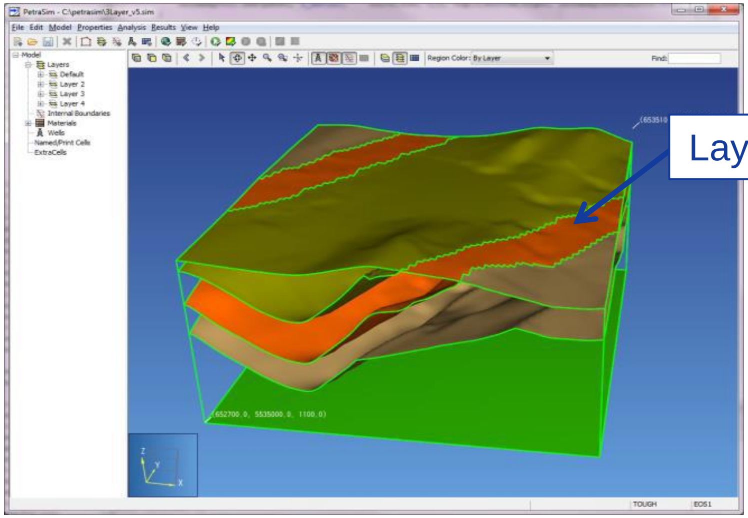


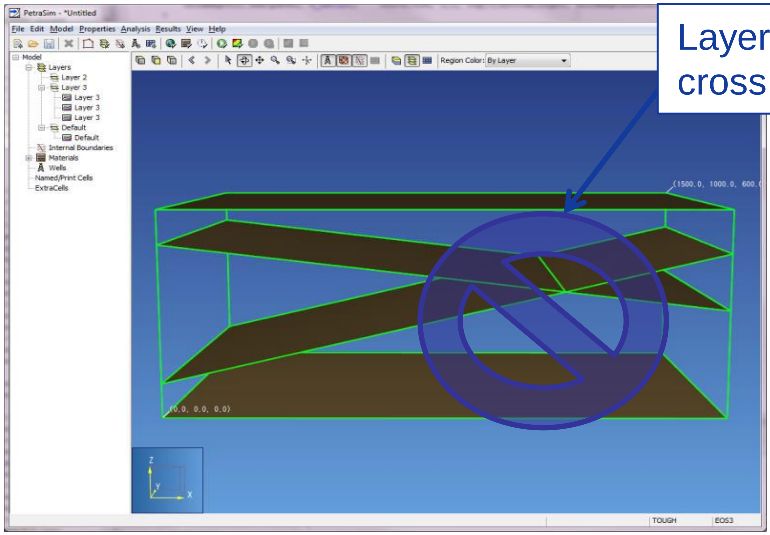
Conceptual Model: Layers

- Layer divisions should extend to the boundary of the model.

- Layer divisions are allowed to touch along areas, pinching the layer, but they should not cross within the model boundary.

- There must always be at least one layer. If you do not define one, the program will create a single default layer based on a planar upper and lower surface.



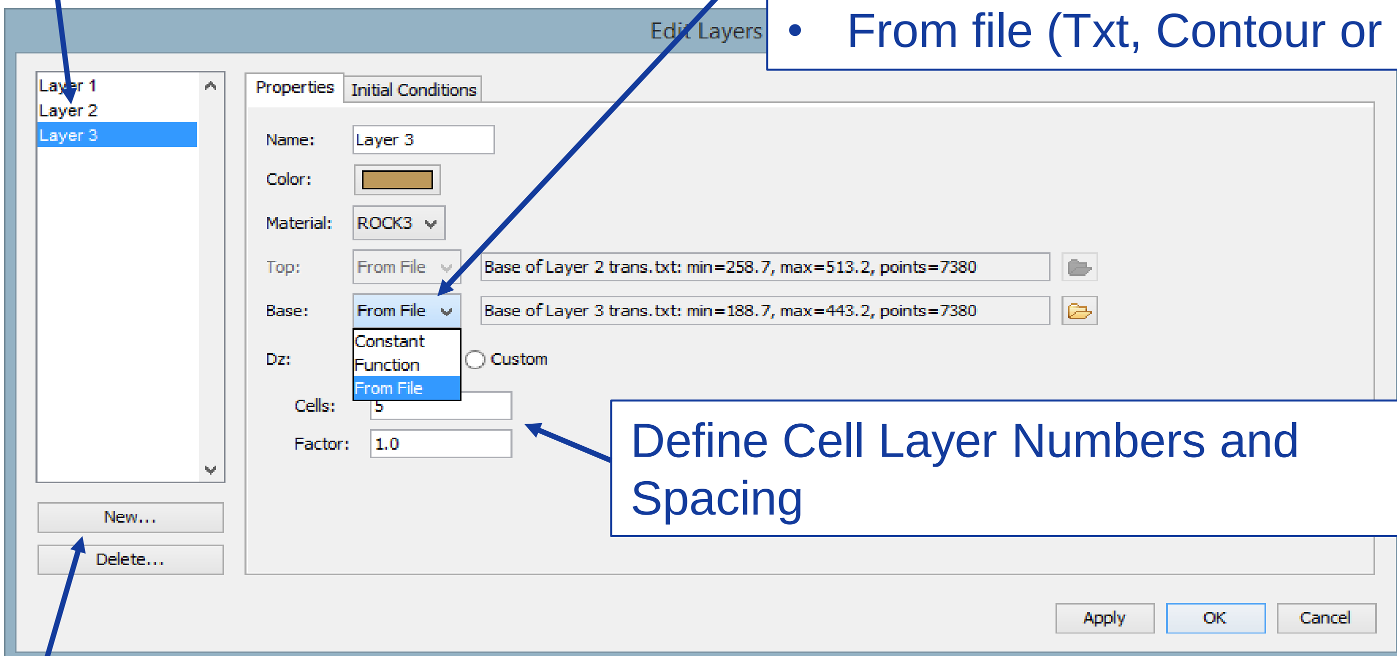


Conceptual Model: Layers

- Used to create and edit layers

Layer List

Define Layer as:
- Constant
- Function
- From file (Txt, Contour or DXF file)



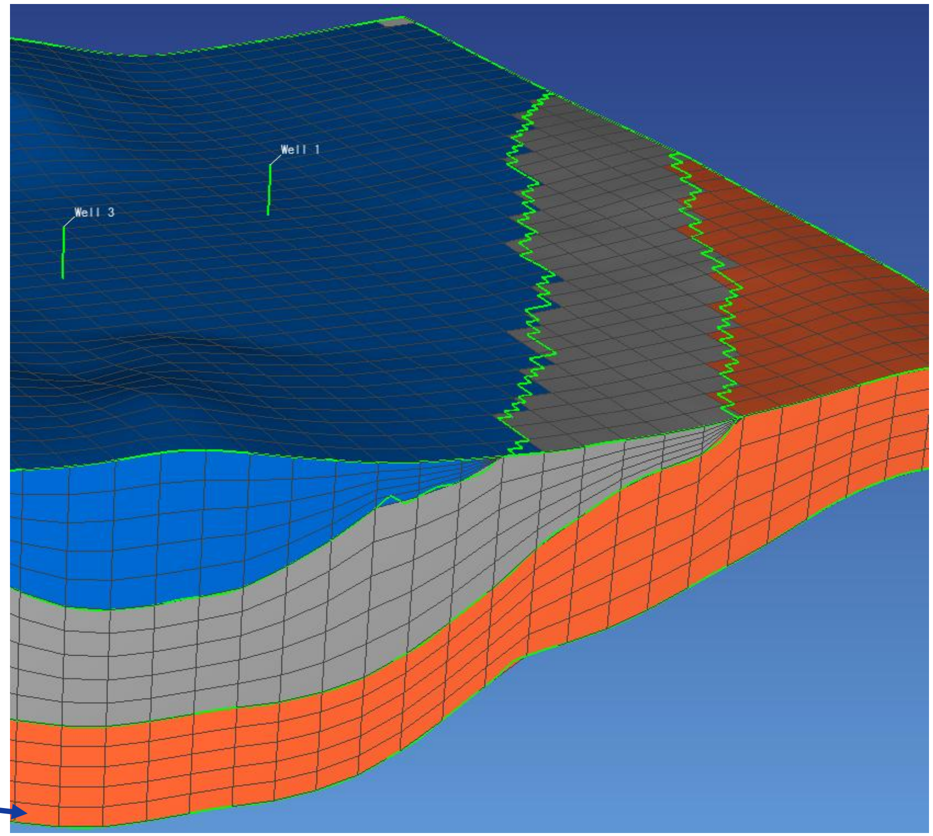
Define Cell Layer Numbers and Spacing

Add or Delete Layers

Conceptual Model: Layers & Mesh

- When a mesh is created, the mesh cell layers mimic the layer elevations and can, in some cases, disappear.

- Warning about possible convergence problems with pinching out layers.



Conceptual Model: Layers & Mesh

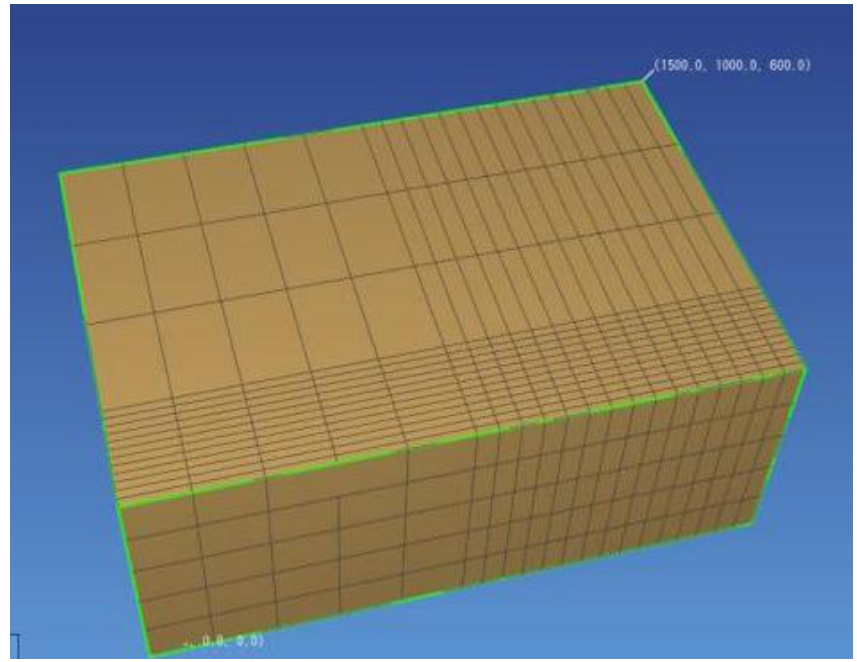
PetraSim provides three types of solution meshes:

- **Regular** – cells are rectangular hexahedrons.
- **Polygonal** – uses extruded Voronoi cells to conform to any boundary and supports refinement around wells.
- **Radial** – represents a slice of an axisymmetric cylindrical mesh. This is based on the **Regular** mesh, but it only allows 1 Y-division.

Conceptual Model: Regular Mesh

- Orthogonal grid cell columns and rows

- Grid cells spacing can vary in each direction and can be refined around wells or other areas where you might expect to see a high flow or heat gradient



- Models are typically stable and grids honor geometric requirements of the TOUGH2 simulators

- Not always efficient – lots of extra grid cells created in areas adjacent to refinement areas.

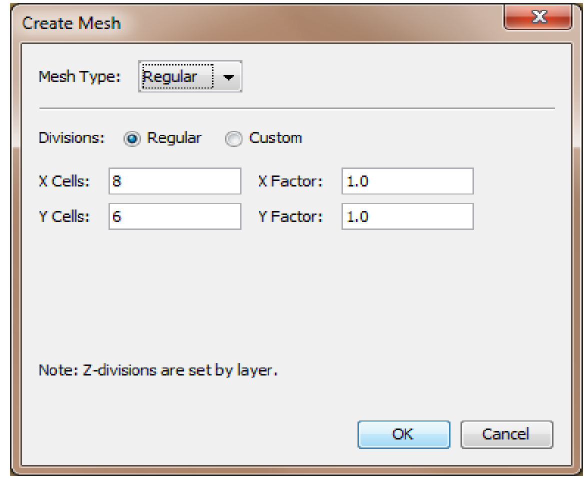
Conceptual Model: Regular Mesh

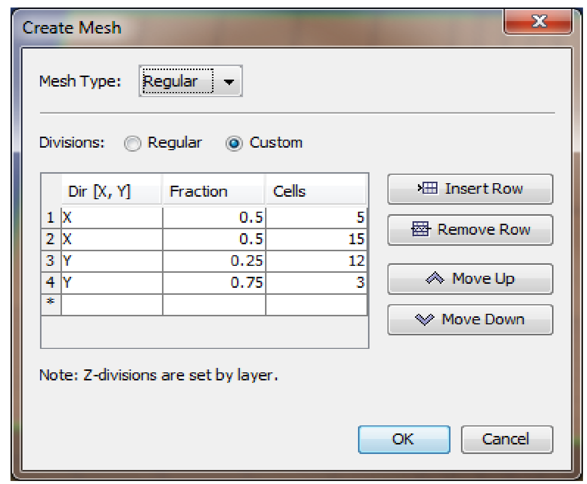
Spacing options include:

- Regular – Constant spacing in each direction

- Regular with a spacing factor – Spacing factor increases or decreases cell size based on equation listed in User's Manual.

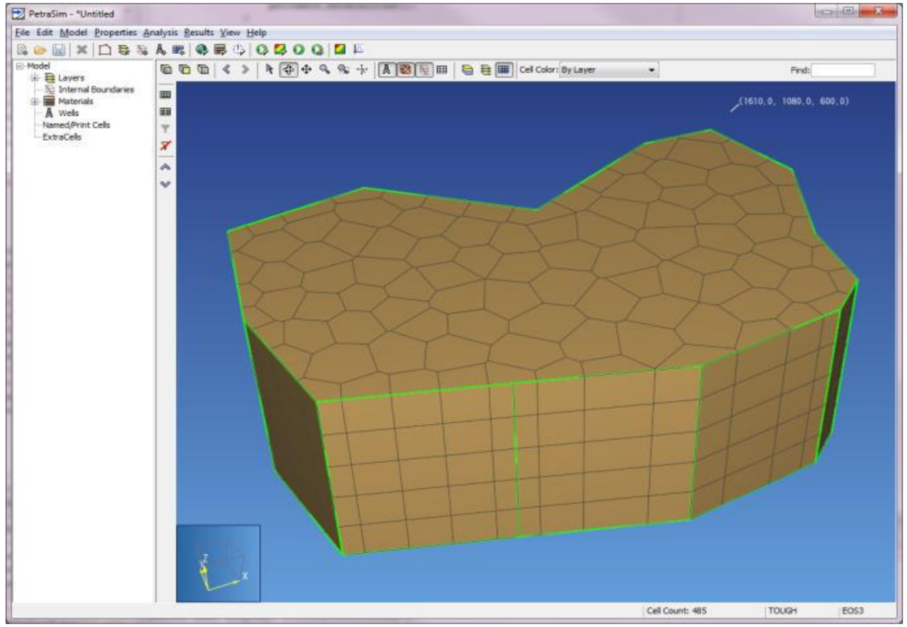
- Custom – Cell spacing is specified using a format similar to the TOUGH2 MeshMaker format.





Conceptual Model: Polygonal Mesh

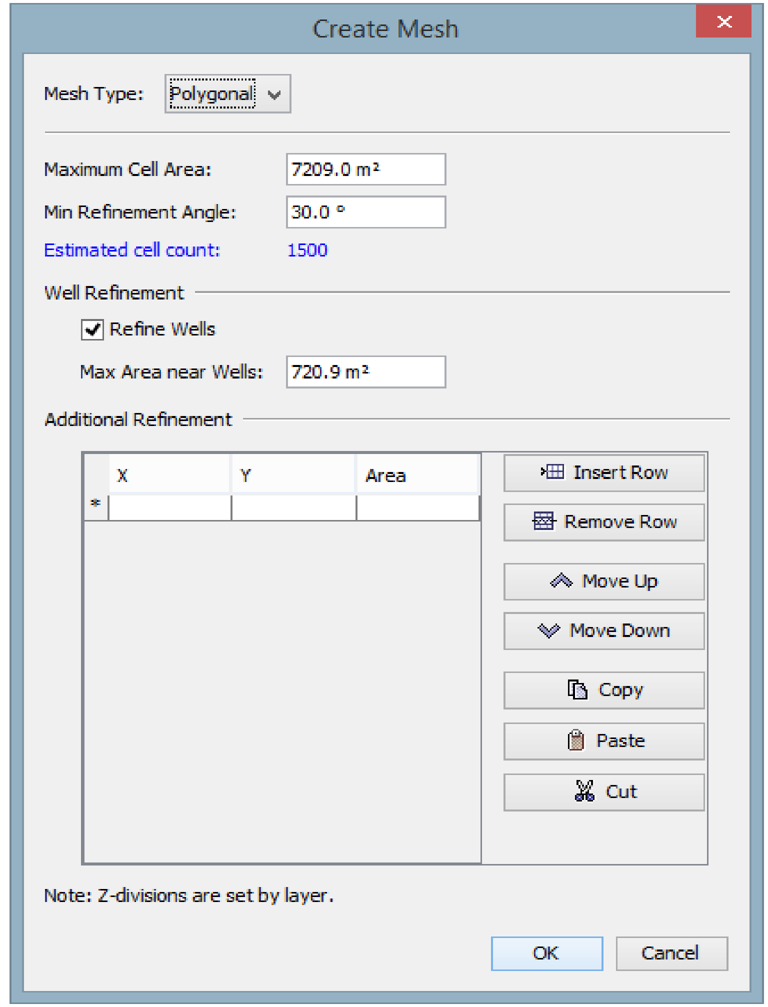
- Uses extruded Voronoi cells
- Cells can conform to any boundary
- Cells can be refined around wells or other refinement points defined by the user
- More efficient way to model larger areas – only refine the mesh in areas where you need to
- Con – Post-processing contours not as smooth
- Con – Small edge length might cause convergence problems



Conceptual Model: Polygonal Mesh

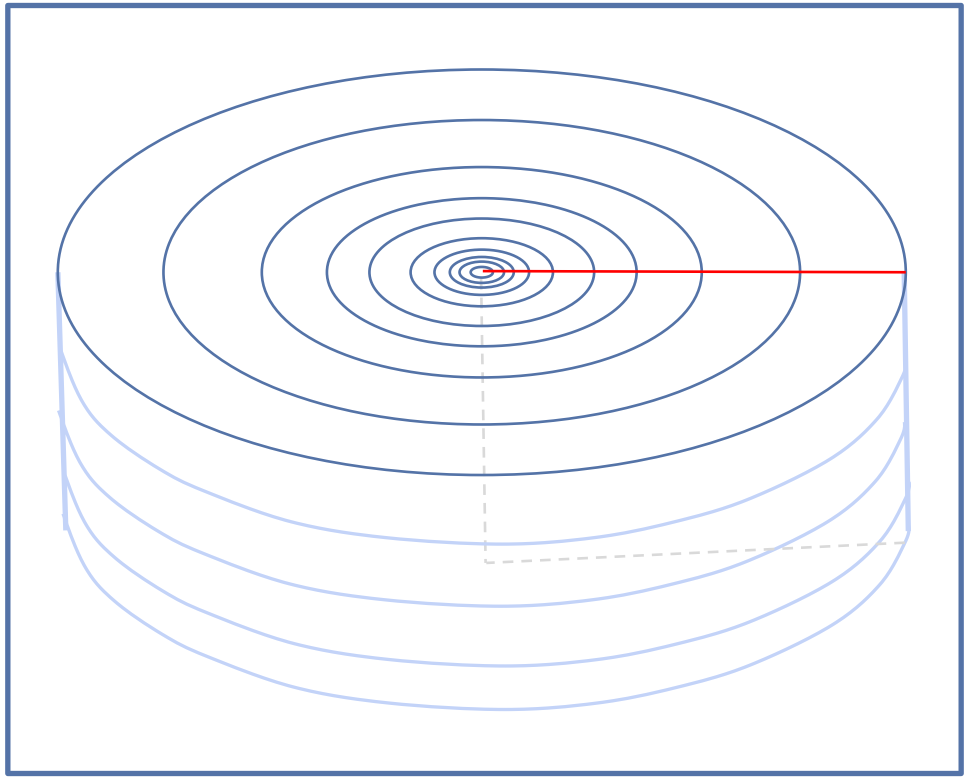
Parameters defined during mesh creation:

- Maximum Cell Area (approximate) in XY Plane
- Minimum Refinement Angle – controls how quickly the area near wells disperses.
- Maximum Area near Wells
- Additional Refinement –defines X and Y coordinates (and approximate areas) at which to apply refinement to the mesh.



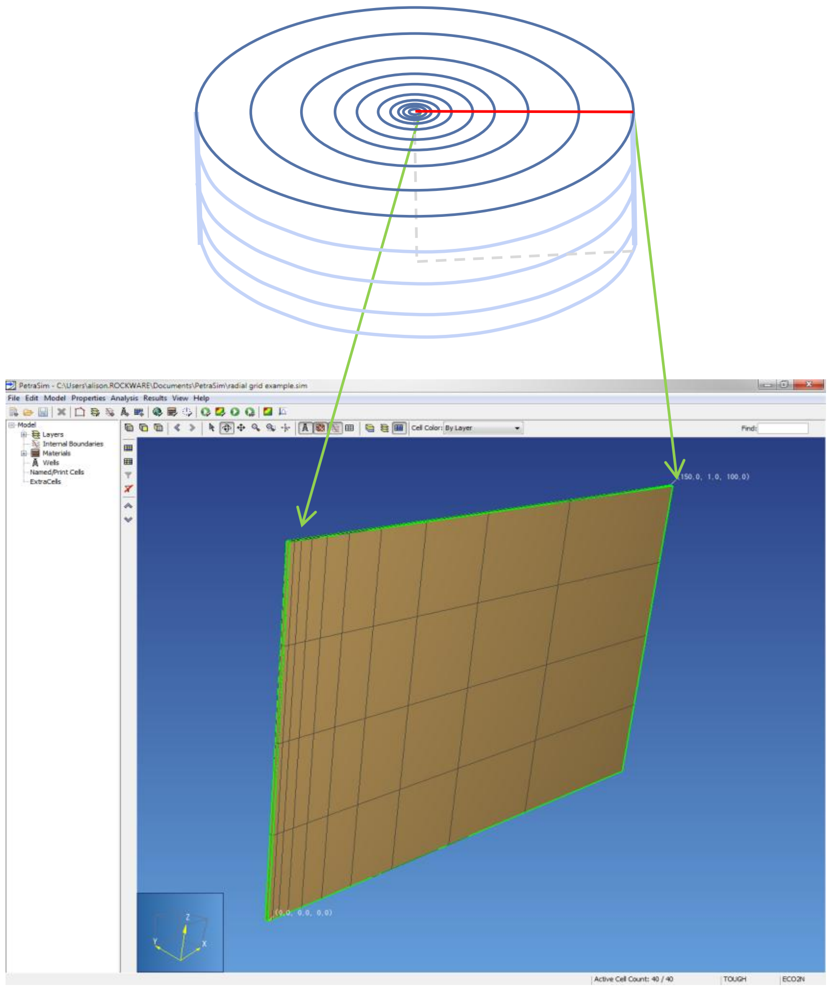
Conceptual Model: Radial Mesh

- Same as a TOUGH2 Meshmaker R-Z (radially symmetric) mesh
- Represents a group of 1D or 2D cylindrical model cells (shaped like doughnuts)
- Wells are typically placed in the "center" of the grid to simulate injection or production



Conceptual Model: Radial Mesh

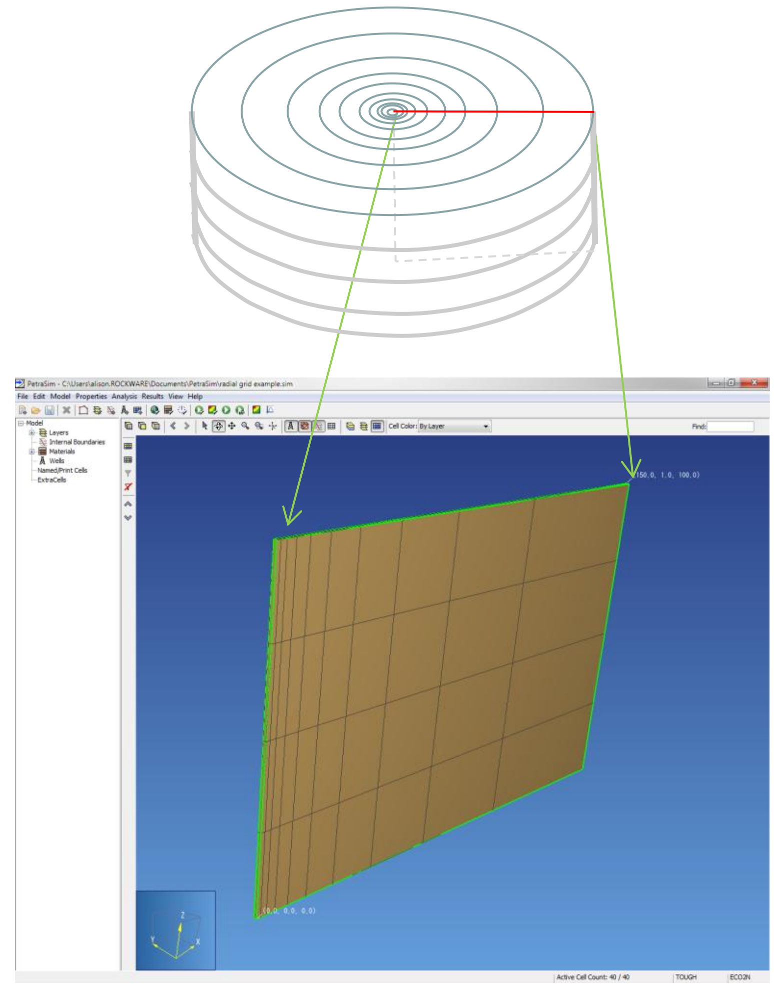
- In PetraSim, displayed as a 2D slice through the radius of the cylinder
- Good for simple models of injection/production (often used for CO2 modeling)
- Impossible to accurately represent non-horizontal geological units



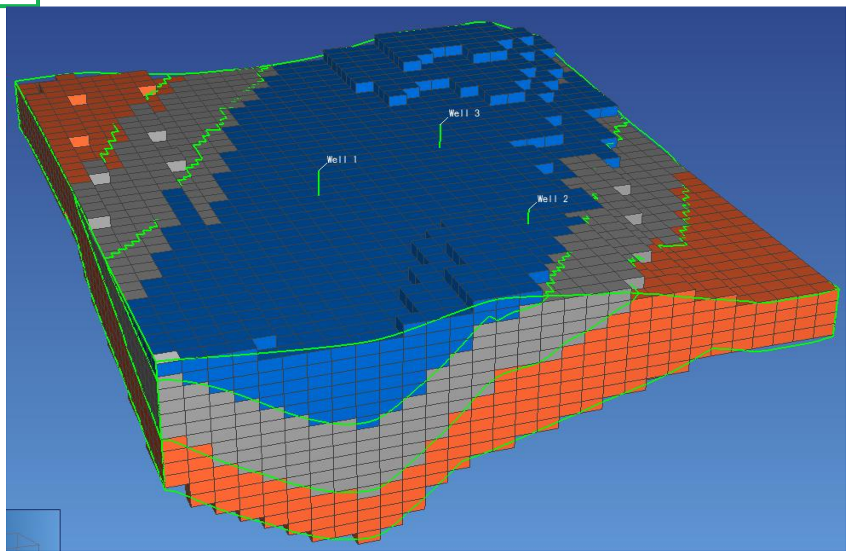
Conceptual Model: Radial Mesh

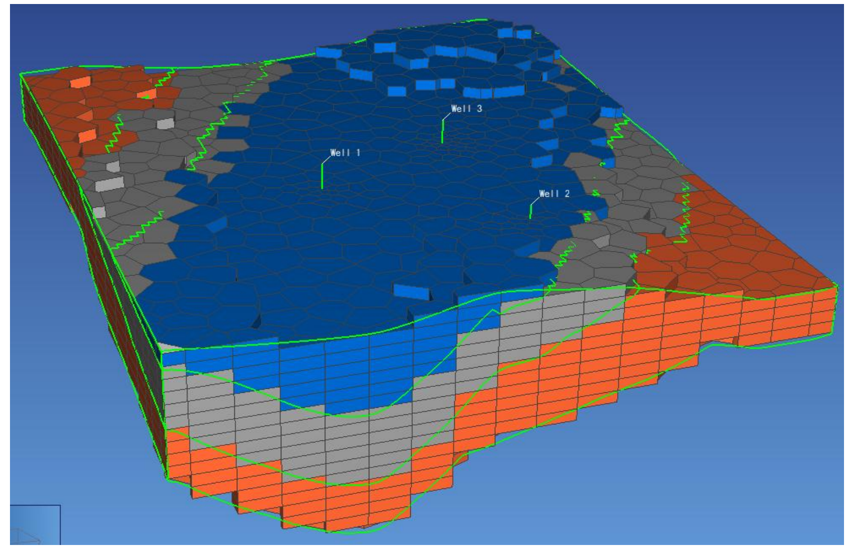
- The only parameter needed to create the mesh is the radial divisions, which correspond to the X divisions in the resulting mesh.

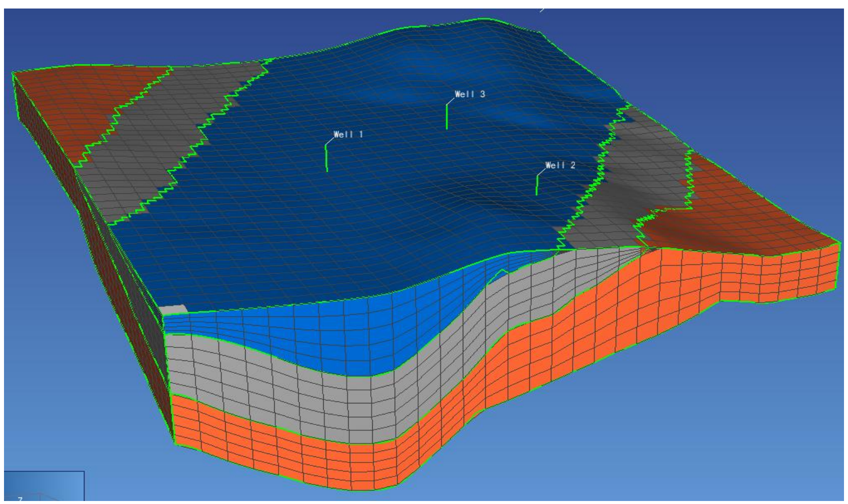
- When creating this type of mesh, you should make the spacing in the Y direction 1 m.

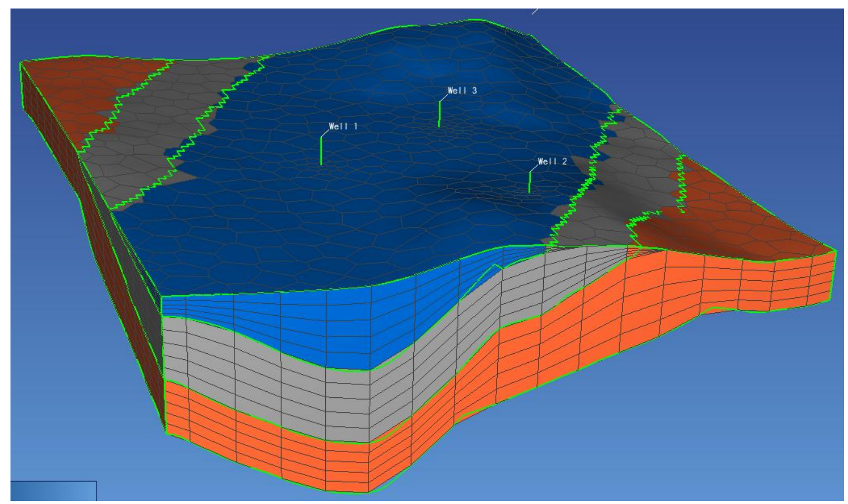


Conceptual Model: Review



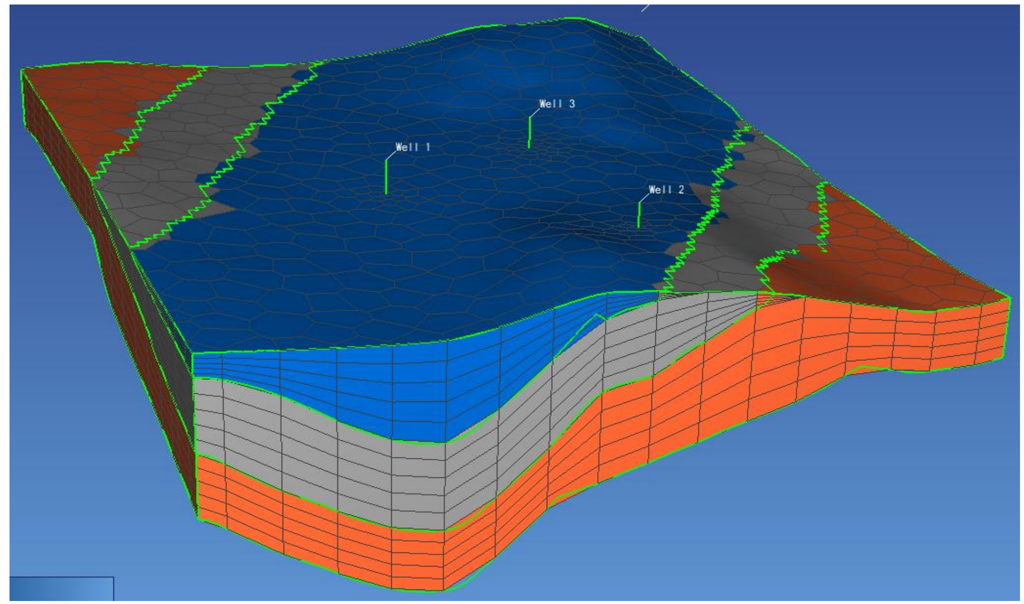






Conceptual Model: Review

- Polygonal Mesh (refined around wells)
- Multiple Conceptual Layers
- Cell layer thickness varies with Conceptual Layers



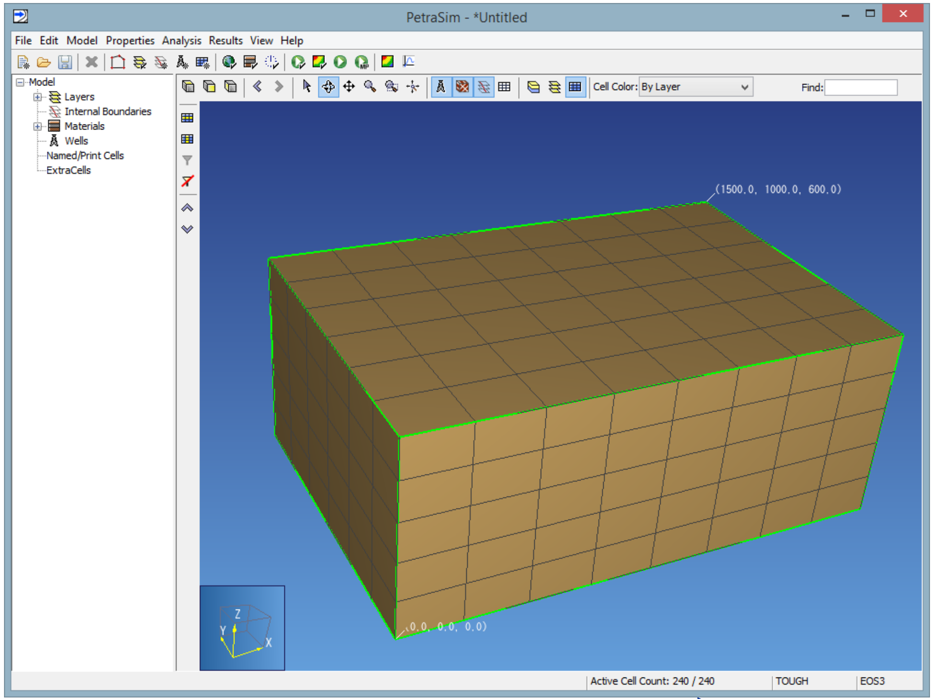
Conceptual Model: Boundary

Three types of boundary conditions available in PetraSim/TOUGH2:

- No Flow (Neumann)
- Constant (Dirichlet)
- Sinks/Sources for fluid, gas, heat, etc.
- Time-based

Conceptual Model: closed boundary

- By DEFAULT, all boundaries of a TOUGH2 model are closed.
- Injection/production in and out of a closed model can cause unrealistic pressures that will cause the simulation to stop.
- Solution is to use a very large model extents, or to open up the boundary of the model to allow flow in and out

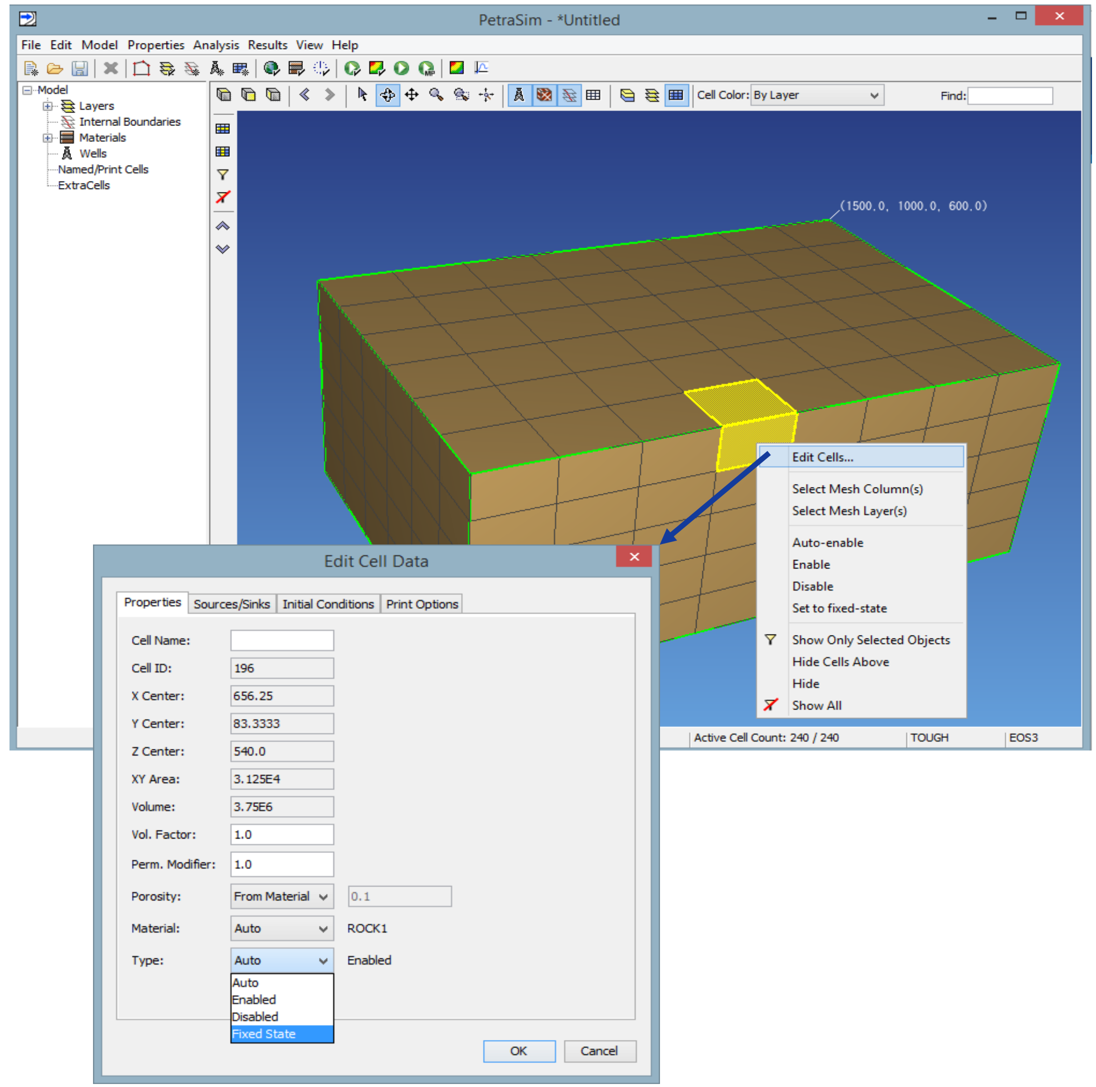


No connections, closed boundary

Conceptual Model: fixed value boundary

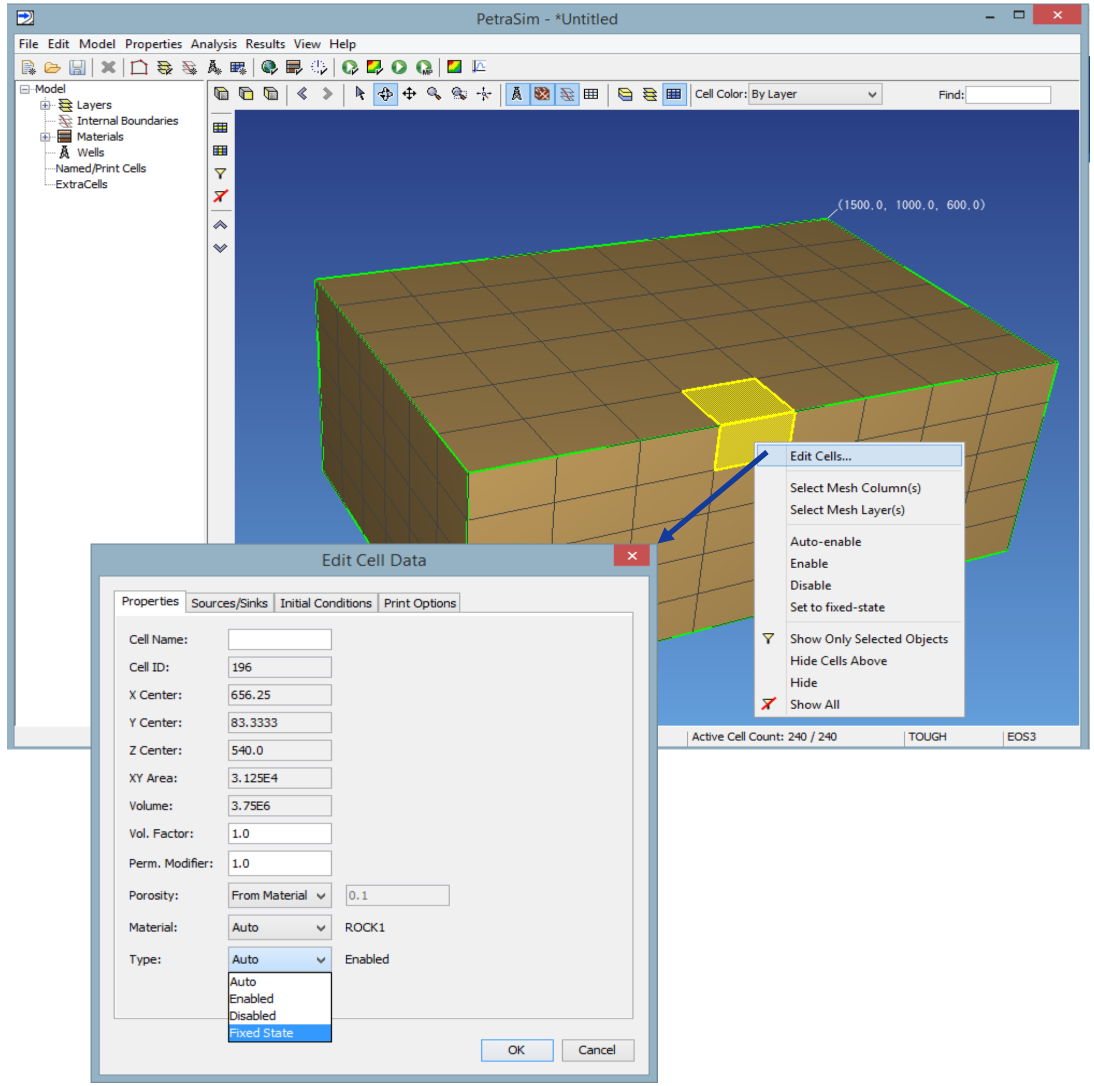
- Dirichlet boundaries are typically created using the “Fixed State” cell option in PetraSim

- Depending on the simulator, PetraSim will either make the volume of a Fixed State cell very large, or will make it an inactive cell in the input file



Conceptual Model: fixed values boundary

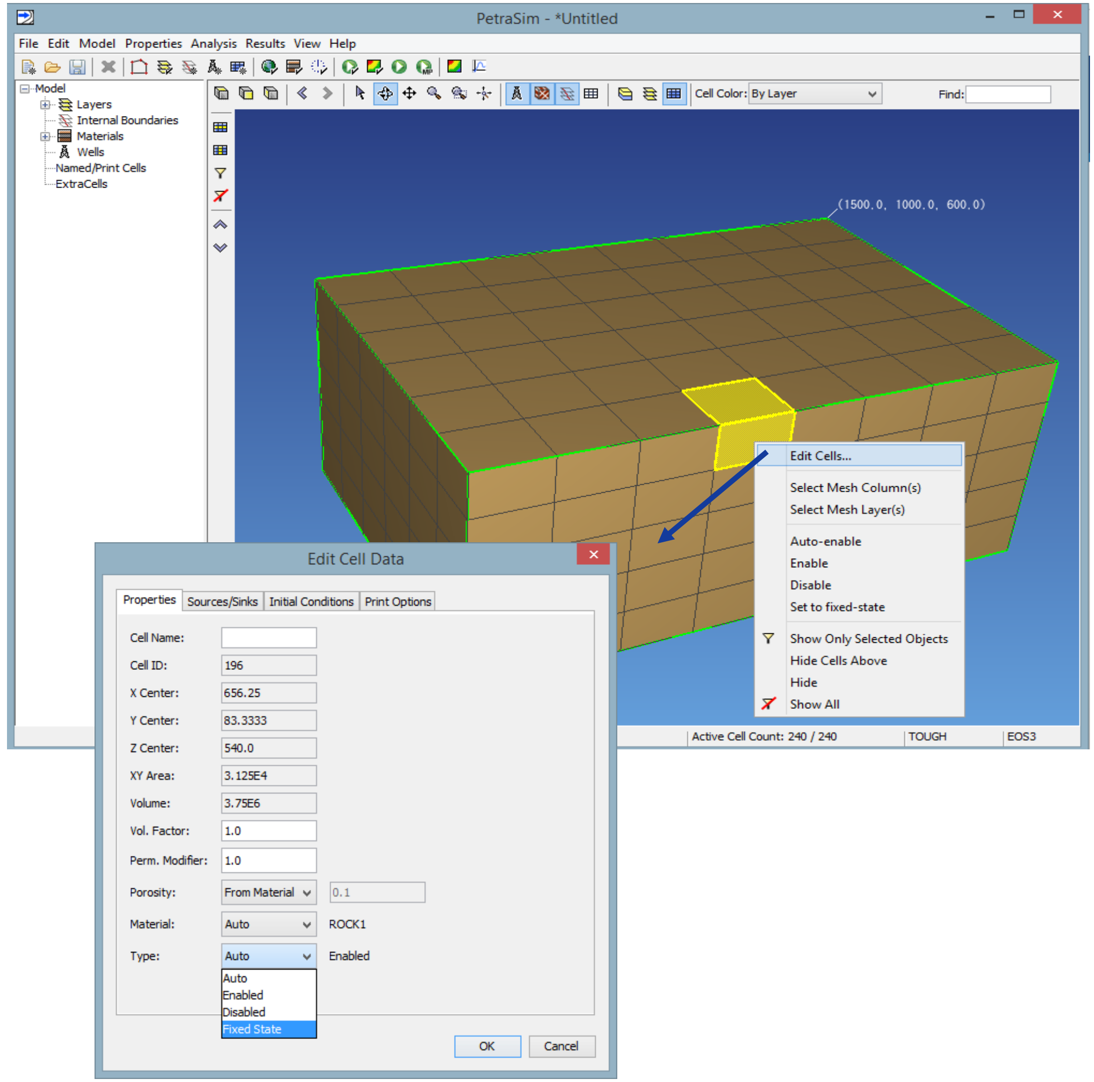
- Fixed State cells will:
  - Be open to fluid/gas and heat flow.
  - Will have a fixed pressure and temperature (and state) based on the initial condition of the cell
- Flow in and out of the cell has no affect on the state of the cell because of the very large volume



Conceptual Model: fixed values boundary

- Requires that you create special materials that are assigned to the boundary cells.

- For visual purposes, we recommend that you make these cells very thin along the boundary of the model (or used “Extra Cells”).

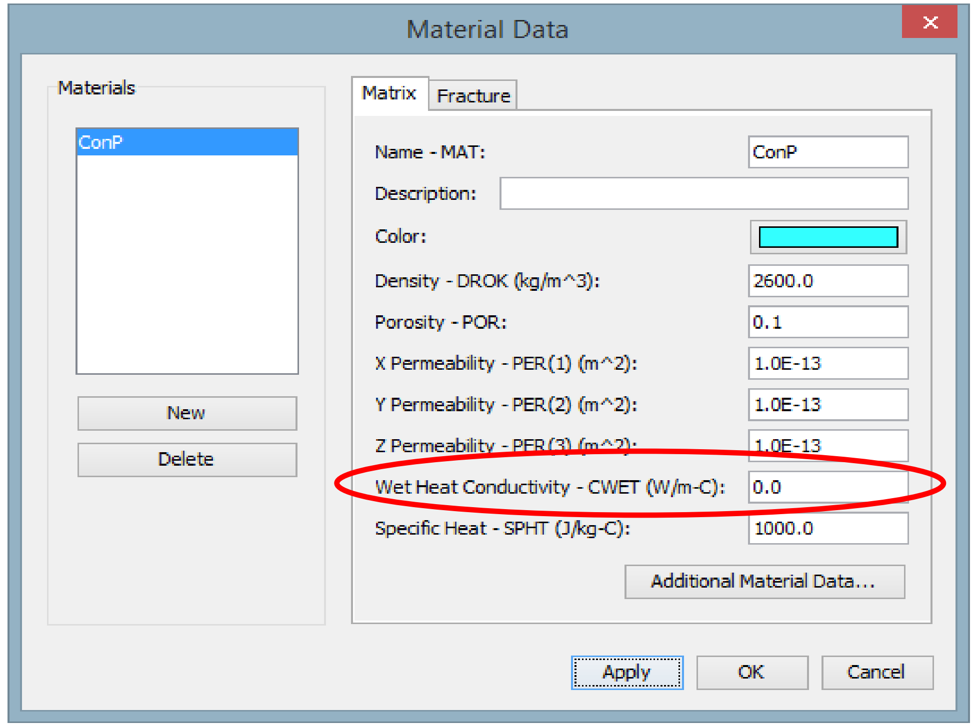


Conceptual Model: fixed pressure boundary

- Make the thermal conductivity of the cell equal to 0 and make the cell “fixed state”

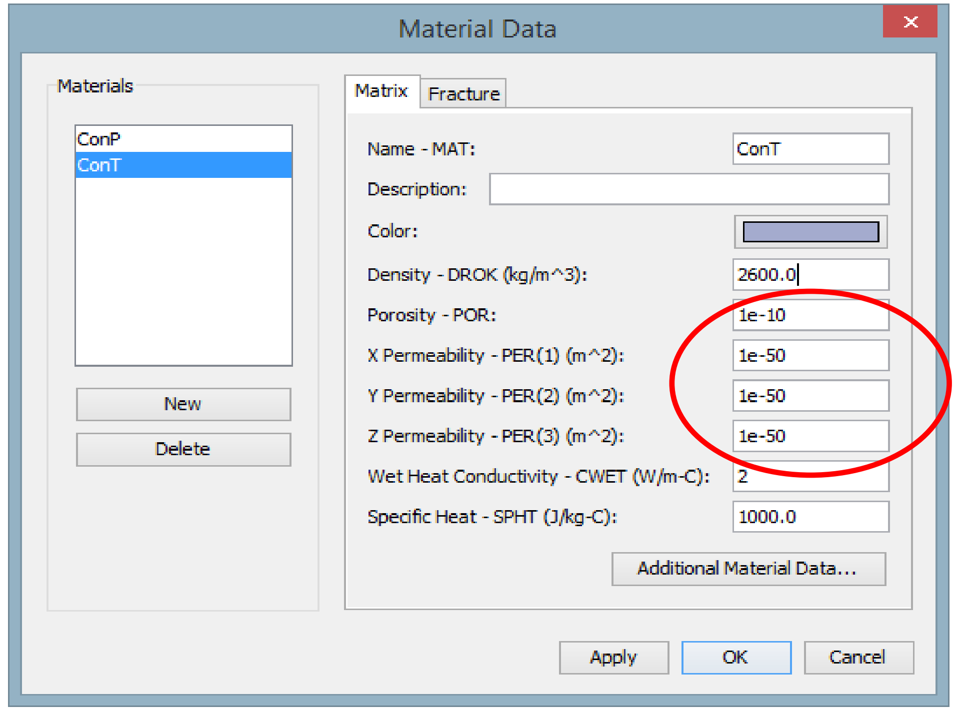
- Fluid will flow in and out of the cell with a very large volume, and pressure will not change

- Cell will not contribute heat to the model or absorb heat, and the heat in the cell will not change



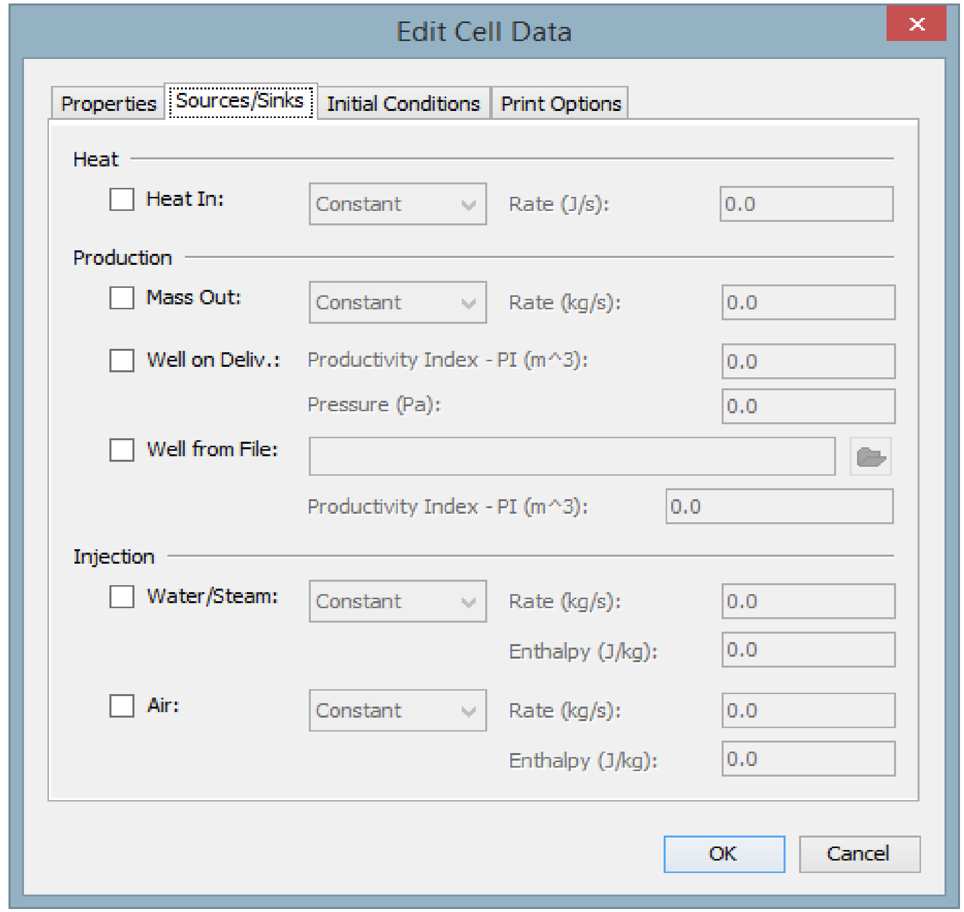
Conceptual Model: fixed temperature boundary

- Make the thermal permeability and porosity of the cell very small and make the cell “Fixed State”
- Cell will act as a closed boundary to flow
- Cell will act as a constant sink or source of heat based on the initial temperature of the cell



Conceptual Model: Source/Sink

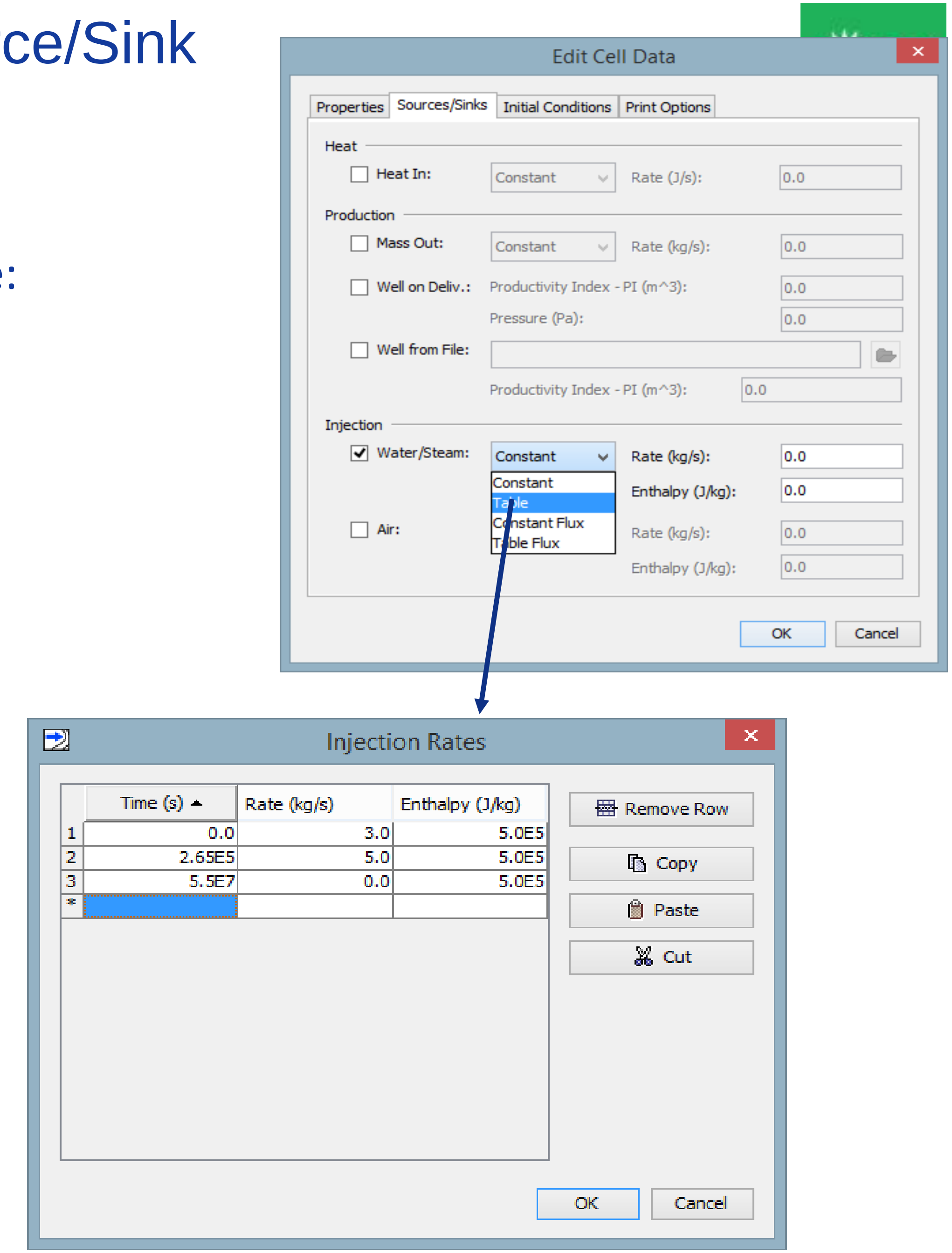
- Used to define flow into or out of the cell
- Used to represent injection, production, recharge, a heat source, etc.
- Right click on a cell or group of cells to Edit the Properties and add sinks/sources.
- Sinks/sources available for heat, fluid, gas, NAPL, etc. (dependent on the EOS module)



Conceptual Model: Source/Sink

Heat, Injection, Mass Out options include:

- Constant (J/s or Kg/s)
- Table-Based
- Constant Flux ($^{J/s}/m^2$ or $^{Kg/s}/m^2$) – based on top area of cell
- Table Flux



THANKS FOR YOUR KIND ATTENTION