ROSIM manual v1.2.0

Contents

Summary

Geothermal energy is considered a key renewable energy source in the Netherlands for heat demand in the built environment, anticipated to supply 20PJ in 2030 and 135PJ in 2050 according to the Masterplan Geothermal Energy. More and cheaper resource development (optimization of well design, stimulation design, VOI for white spots) and safe and effective exploitation (induced seismicity, ultimate recovery) requires 2D/3D simulation workflows for reservoirs, honouring structural and stratigraphic complexity and coupled model approach capabilities.

ROSIM is a TNO in-house developed tool, distributed free of charge, to build static 3D geological subsurface models and perform subsequent flow simulation using DoubletCalc 3D (also TNO in-house developed) or OPM [1]. The simulation results are visualized using the free and open source software ResInsight [2].

ROSIM has been benchmarked against Eclipse300 [3]. ROSIM, using DoubletCalc 3D, performs well for the geothermal and high temperature storage configurations tested in this study.

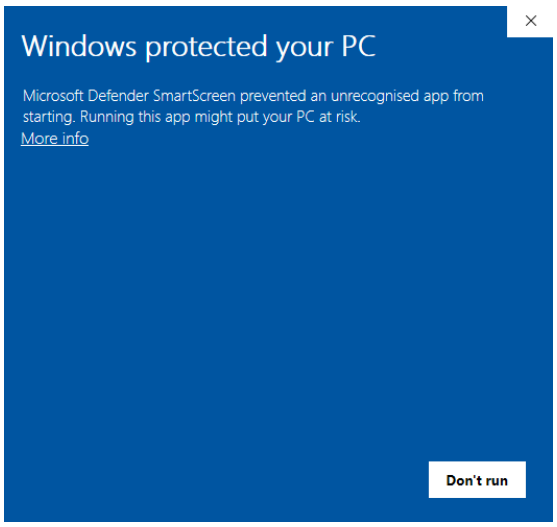
Installation Instructions

ROSIM makes use of two external applications: ResInsight for visualization and OPM as an alternative flow simulator to DoubletCalc 3D. ResInsight must be installed to use ROSIM. OPM is optional, since DoubletCalc 3D can be used as simulator. Below are installation instructions for ROSIM (including DoubletCalc 3D), ResInsight and OPM (optional). Files from the 'ROSIM-vx.x.x_xxx_20xx.zip' are needed in the steps below.

ROSIM installation

Place the 'ROSIM-x.x.x' folder on your local drive, in 'Program Files' for instance. Double click ROSIM.exe to start. ROSIM will start but will not fully work yet since it needs ResInsight and optionally OPM.

NOTE: In some cases the rosim app doesn't start due to the following message:

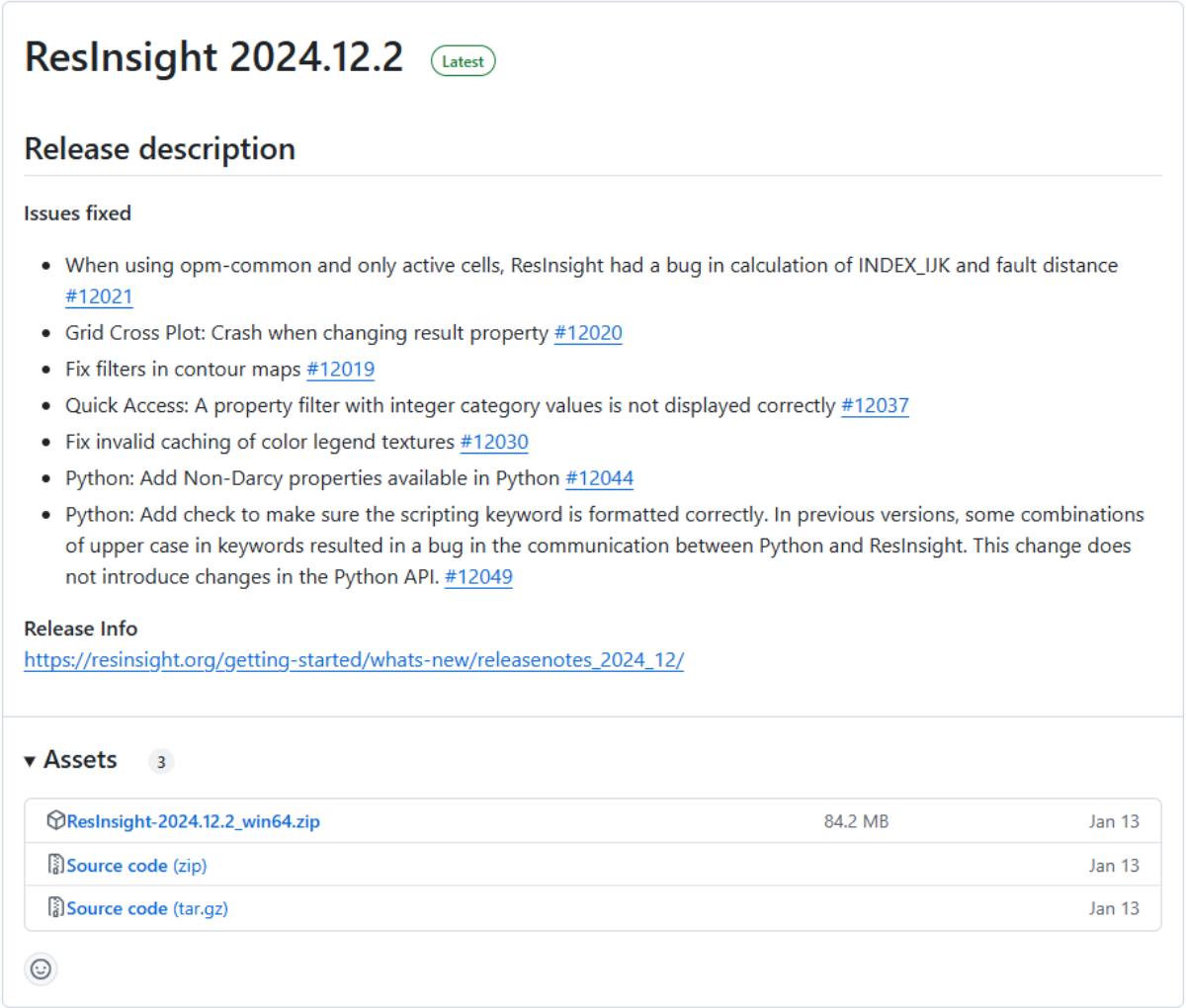


This can be solved by opening a command prompt as administrator and run the .exe file.

ResInsight installation

1. Download ResInsight:
   https://resinsight.org/releases/windows-installation/
   which leads to https://github.com/OPM/ResInsight/releases, and download zip (a later

version could be available, which might not be compatible):



ResInsight 2024.12.2 Latest

Release description

Issues fixed

- When using opm-common and only active cells, ResInsight had a bug in calculation of INDEX_IJK and fault distance #12021
- Grid Cross Plot: Crash when changing result property #12020
- Fix filters in contour maps #12019
- Quick Access: A property filter with integer category values is not displayed correctly #12037
- Fix invalid caching of color legend textures #12030
- Python: Add Non-Darcy properties available in Python #12044
- Python: Add check to make sure the scripting keyword is formatted correctly. In previous versions, some combinations of upper case in keywords resulted in a bug in the communication between Python and ResInsight. This change does not introduce changes in the Python API. #12049

Release Info
https://resinsight.org/getting-started/whats-new/releasenotes_2024_12/

▾ Assets 3

ResInsight-2024.12.2_win64.zip	84.2 MB	Jan 13
Source code (zip)		Jan 13
Source code (tar.gz)		Jan 13

2. Unzip and put in 'Program Files\ResInsight\' for instance
3. Add the folder containing ResInsight.exe (for instance: `C:\Program Files\ResInsight\ResInsight-2024.12.2_win64\bin`) to your PATH: https://www.architectryan.com/2018/03/17/add-to-the-path-on-windows-10/

NOTE: Occasionally, when ResInsight has been started up many times, ResInsight will give an error on start-up and crash. This can be solved by opening the 'Task Manager' and under 'Processes' right click and 'End task' all ResInsight instances. If that doesn't work the last resort is a computer restart.

OPM installation

OPM is a Linux application which can't run directly on Windows. There are two options through which OPM can be run: in WSL (Windows Subsystem for Linux) or inside a Docker container. The installation procedure for both is described below, only one of the two options is needed. Option 1 (Docker Container) will use a fixed version of OPM: for ROSIM 1.2.0 that is OPM '2024.10'. When using option 2 (WSL environment) the latest version of OPM will be installed which might lead to unexpected (better or worse…) results.

Option 1: run in Docker Container (preferred)

- Download and install Docker Desktop: https://www.docker.com/products/docker-desktop.
  - If asked during installation, check 'WSL2' and 'Linux containers'
  - If asked (when starting Docker Desktop), install the Linux kernel update

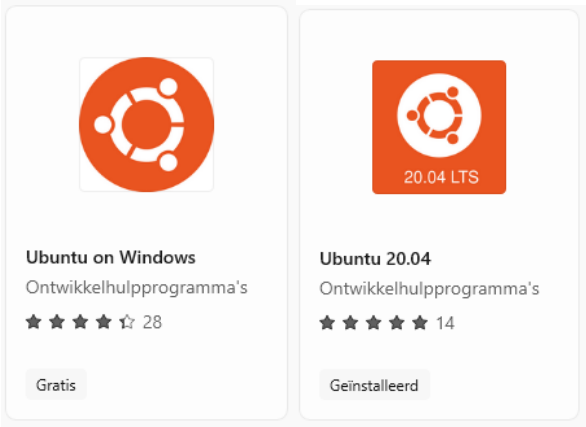
To check if docker is running, in Windows PowerShell type 'docker -v' which should give a message similar to below:

```
PS C:\Users\vrijlandtmaw> docker -v
Docker version 20.10.17, build 100c701
```

NOTE1: Docker desktop must be running for *Run Flow Simulation* with OPM.
NOTE2: Due to restricted rights set by your company's IT policy you might need assistance from your IT department to install Docker Desktop.

Option 2: run in WSL environment

1. (option A) Install 'Ubuntu on Windows' from the 'Microsoft Store' ('Ubuntu 20.04 LTS' works as well):



(option B) If the Microsoft Store is not available (such as on TNO laptops) run Powershell as administrator and:
'wsl --install'
Next check which distributions are available:
'wsl -l -o'
Next install an available distribution:
'wsl –-install -d Ubuntu-22-04'
To check whether the installation was successful:
'wsl -l -v'

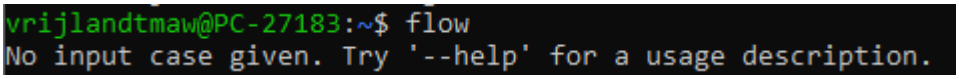
NOTE: Due to restricted rights set by your company's IT policy you might need assistance from your IT department to install WSL.

2. In a Windows PowerShell type: 'wsl -l -v' which should show something like this:

```
PS C:\Users\vrijlandtmaw> wsl -l -v
  NAME                   STATE           VERSION
* Ubuntu-20.04           Running         2
  docker-desktop-data    Running         2
  docker-desktop         Running         2
```

A star indicates the default, it should be at the Ubuntu version that you installed the scripts in. If not run: 'wsl --setdefault Ubuntu-20.04'.

3. Save the ‘installLatestOPM.sh’ script locally
4. In ‘Ubuntu on Windows’ navigate to the directory where the file is saved (‘cd /mnt/c’ directs to your local C drive) and from that directory type:
   ‘sudo ./installLatestOPM.sh’
   This can take a couple of minutes.
5. To test whether it was properly installed type ’flow‘ which should give the message below:



Technical description

Static modelling and prepare flow model

A static model is created with properties per layer (layer-cake model) as described below in the Input 3D Static Model Tab. Optionally a tartan grid refinement can be used around selected wells (Input Wells Tab). The PVTNUM and SATNUM properties (under 'static' properties in ResInsight) give the layer number. The 3D static model is saved as a '.grdecl' file which can be read by DoubletCalc 3D, OPM and Eclipse. Two different '.grdecl' files are saved since DoubletCalc 3D and OPM require slightly different keywords.

DoubletCalc 3D and OPM need a different format of flow input file, both of which created by ROSIM. These files contain the same information which consists of three main components: the flow constraint schedule, the fluid properties and the well indices and connection factors. The flow constraints schedule is straightforward from the input with one key note: the maximum/minimum bottom hole pressure in the flow constraint schedule in the Input Wells Tab is not used for DoubletCalc3D, only for OPM flow. This means that for DoubletCalc 3D the injection and production rates are always equal. The well pressures have to be checked in the DoubletCalc 3D case to check whether they are not too high or low. For OPM the maximum pressure is taken into account. This means that the well flow rates have to be checked, to make sure that the total production and injection rates are equal, which will generally be required for geothermal and high temperature storage applications.

The water density, viscosity and heat capacity properties are dependent on the pressure, temperature and salinity. These are calculated using empirical relationships: Batzle&Wang [4] is used for the density and heat capacity, Kestin [5] is used for the viscosity, which are displayed in Figure 1. These relationships are used directly in DoubletCalc 3D. For OPM property tables are created in the PROPS section using these relationships. A separate table exists per layer (region in OPM).

The well indices and connection factors are determined from the 3D grid geometry, the rock and fluid properties of the 3D model and the well configuration. The calculation method is described in L. Peters et al, [6] and [7]. This calculation results in a list of grid blocks per well, which represents the open (perforated) section for each well. And for each grid block a connection factor describes the connectivity between the well and the grid cell. When specifying the 3D grid and wells, make sure that wells are not located on the boundary between grid blocks, because then it becomes unpredictable in which grid block the well will be placed.

A simple fault functionality is available for vertical faults: some properties grid cells along the fault trajectories are set to a user specified value.

With the static model files and flow input files in place the next step is to perform the flow simulation, using DoubletCalc 3D or OPM flow.

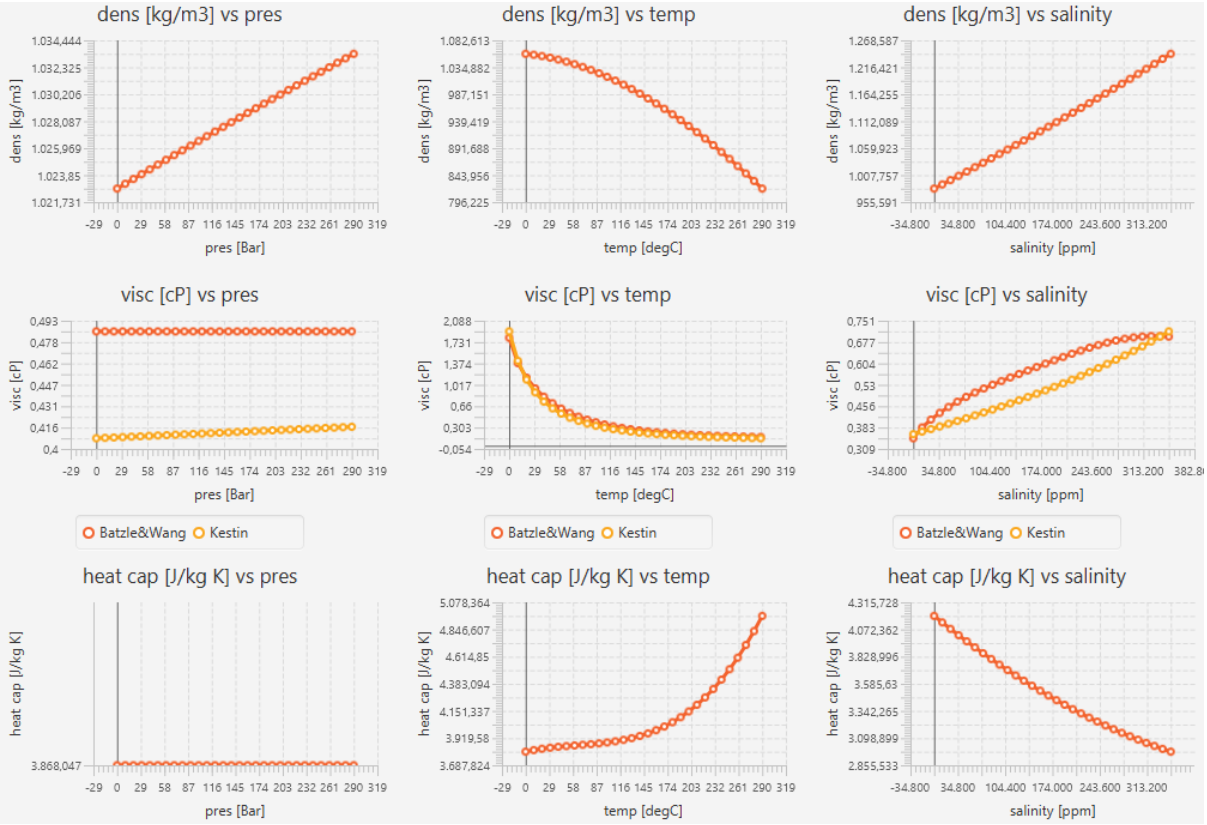


Figure 1: plots of the water properties density, viscosity and heat capacity versus pressure, temperature and salinity. The reference pressure, temperature and salinity are 200 Bar, 80 °C and 70,000 ppm respectively.

Flow Modelling

As mentioned before, there are multiple options for performing the flow simulation, DoubletCalc3D and OPM, for which the input is prepared in the 'Prepare Flow Model' step. Both simulators give the results in a '.EGRID' format: both results can be viewed in ResInsight and can be used for further post-processing, such as a compaction and subsidence calculation.

DoubletCalc3D

DoubletCalc3D is a TNO inhouse developed geothermal simulator which is a 3 dimensional extension of existing tools for 1D and 2D simulation at TNO (NLOG, 2022, Pluymaekers et al., 2016) [8]. It is a dedicated single phase simulator which enables the numerical simulation of temperature and pressure development using temperature dependent density and viscosity.

In a Eulerian reference framework, the transient heat equation is written as

$$\rho c \quad \frac{\partial T}{\partial t} = \nabla \cdot (\lambda \cdot \nabla T) - \vec{v} \cdot \nabla T \qquad (1)$$

See Table 2 for nomenclature. The advective velocity $\vec{v}$ can also be a result of fluid flow inside pores or fractures which strongly affect the thermal distribution. The fluid velocity is resolved from solving the Darcy flow equation:

$$c_h \frac{\partial P}{\partial t} = 0 = \nabla \cdot \left(\frac{\kappa}{\mu} \left(\nabla P + (\rho_f - \rho_0) g \nabla z \right) \right) + Q \qquad (2)$$

Through solving the pressure field in equation 2, the velocities can be determined as

$$\overrightarrow{v_f} = \frac{k}{\mu}\left(\nabla P + \left(\rho_f - \rho_0\right) g \nabla z\right) \quad (3)$$

In DoubletCalc3D, the flow and thermal field are solved with a finite volume finite difference formulation. The finite volume method solves for a potential u_K (e.g. temperature, pressure) at the center of polyhedral cell (index K) at the center point $\boldsymbol{x}_K$, based on continuity equations taking into account the fluxes at the interfaces to other cells [9]. In DoubletCalc3D, the cells are hexahedral in a structured i,j,k grid.

$$\frac{du_K}{dt} = \frac{1}{c_K}\left(\nabla . \boldsymbol{K} \nabla u_K + q_K\right) \quad (4)$$

which can be written in the finite volume formulation

$$\frac{du_K}{dt} = \frac{1}{V c_K}\left(\sum A_\sigma n_\sigma . \boldsymbol{\kappa_K} \nabla u_K + Q_K\right) \quad (5)$$

where $\sum \mathrm{A}_\sigma \mathrm{n}_\sigma . \boldsymbol{\kappa_K} \nabla u_K$ is the sum of fluxes entering the cell, $\mathrm{A}_\sigma \mathrm{n}_\sigma$is area and normal of face σ, $\boldsymbol{\kappa_K}$ is conductivity tensor of cell K. The summation is of fluxes (W for thermal, m3/s for flow). Q_K is source term of cell K (W for thermal, m3/s for flow). The term $\mathrm{A}_\sigma \mathrm{n}_\sigma . \boldsymbol{\kappa_K} \nabla u_K$ assumes the gradient ∇u_K in the cell. In the formulations for a structured hexahedral grid with cartesian coordinates the term $\mathrm{A}_\sigma \mathrm{n}_\sigma . \boldsymbol{\kappa_K} \nabla u_K$ is replaced by a transmissibility formulation in for the six connection in i,j,k. For example, for cell k= (i,j,k) in i direction and we can write:

$$A_{(i,j,k)(i+1,j,k)} \boldsymbol{\kappa_K}_{(i)} \nabla u_K = q_{\,(i,j,k,i+1,j,k)} = T_{(i,j,k)(\,i+1,j,k)}\left(u_{(i+1,j,k)} - u_{(i,j,k)}\right) \quad (6)$$

where $\mathrm{q}_{f\,(i,j,k,i+1,j,k)}$ is the flow rate of the face originating from the neighbouring cell and $T_{(i,j,k)(\,i+1,j,k)}$ is the transmissibility factor of the face connecting cell (i,j,k) and (i+1,j,k). The derivation of transmissibility factors for hexahedral cells is extensively documented in Guo and Languevin [10] and commercial reservoir simulator software (Eclipse [11]). In DoubletCalc3D we adopt the OLDTRAN formulation of the ECLIPSE manual description.

For the gravity driven flow we adopt for eq. 6 a slightly modified form compared to Guo and Languevin, (2002, their eq. 43) [10], assuming density variations can be ignored for correcting the exact mass rates:

$$q_{f\,(i,j,k,i+1,j,k)} = T_{(i,j,k)(\,i+1,j,k)}\left(\left(u_{(i+1,j,k)} - u_{(i,j,k)}\right) + g\left(\rho_f - \rho_0\right)\left(z_{(i+1,j,k)} - z_{(i,j,k)}\right)\right) \quad (7)$$

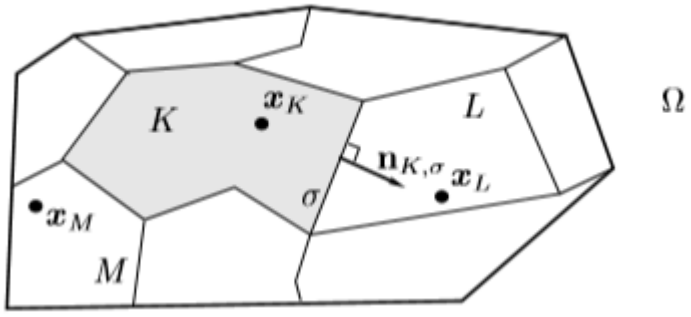


Figure 2: Finite volume formulation for Cell K, constructed from fluxes from the polyhedral faces σ

For the heat equation, the advective fluid mass transfer (eq. 1) is taken into account through

$$q_{h\,(i,j,k,i+1,j,k)} = T_{(i,j,k)(\,i+1,j,k)}\left(u_{(i+1,j,k)} - u_{(i,j,k)}\right) + q_{f\,(i,j,k,i+1,j,k)} C_f \rho_f \left(u_{(i+1,j,k)} - u_{(i,j,k)}\right) \quad (8)$$

where $q_{f\ (i,j,k,i+1,j,k)}$ is the flow rate of the face after solving the pressure equations using eq. 7.

Equations (6-8) result in a sparse matrix system for the time derivative of the potential:

$$\frac{du_K}{dt} = \frac{1}{Vc_K}\left(A_{Kj}u_j - b_K\right) \quad (9)$$

where **A** is a symmetric band matrix, which is positive definite. b_K can differ from $Q_K$as it includes the effects of eliminating cells with Dirichlet (fixed potential) boundary conditions from the system of equations, and can include the effects density driven flow (the last term in eq. 7). Please note that transient

The numerical scheme for solving eqs.8 for pressure and temperature is based on a staggered coupling with discrete time steps (for geothermal years, HT-ATES weeks to month is recommended). For each timestep a steady state solution setting $\frac{du_K}{dt} = 0$. Boundary conditions for the wells are typically mass balance for injection and production rates. Eq. 6-8, whereas at the sides, top and bottom of the mesh no flow boundaries are imposed by default. The system of equations is solved with a preconditioned conjugate gradient solver and results in excess pressure relative to the ambient reservoir pressures.

For the transient heat transfer solution, the flow rates from the cell faces are taken from the pressure solution. Initial temperatures are taken in accordance to a specified geothermal gradient and surface temperature. Boundary conditions are injection temperature in the injection well, and no heat flow at the sides, top and bottom of the grid. The transient solution is obtained using an advanced explicit Runge-Kutta method dedicated to elliptical equations (Verwer, 1977 [12]; Van Wees et al., 2016) [13]. The viscosities and corresponding matrix components (A) are held fixed during the time step.

Symbol	Name	Unit
k	Permeability	m^2 or mD
α	Volumetric thermal expansion coefficient	$°C^{-1}$
ρ	Bulk Density of porous rock	kg m^{-3}
ρ_f	Fluid density	kg m^{-3}
ρ_0	Reference density of fluid	kg m^{-3}
C_h	storage capacity of fluid	Pa^{-1}
C_K	Volumetric heat or storage capacity of porous rock of cell K	J m^{-3} K^{-1}
C_f	Specific heat capacity of fluid	J kg^{-1} K^{-1}
g	Gravitational acceleration	m s^{-2}
T	Temperature	°C
t	Time	s
P	Pressure	Pa
$\vec{v}$	Velocity vector	°C
μ	Fluid viscosity	Pa s
λ	Thermal conductivity	W m^{-1} K
Q	Source term	m^3 s^{-1}

Table 1 parameters in numerical solution of DoubletCalc3D

OPM

Technical information, including a manual, can be found here: opm-project.org. OPM is a linux application while ROSIM is a windows executable. As mentioned in the OPM installation section,

ROSIM runs OPM via Docker Desktop (preferred) or via WSL. Two types of OPM flow decks ('.DATA' files) are created in the 'Prepare Flow Model' step: OPM one_phase_energy, which only models the water phase, and a 'standard' flow deck which models all phases. These two give the similar results but the one_phase_energy mode is faster. The 'standard' OPM flow deck is still present due to legacy reasons.

Since the 2023.04 OPM release, the simulations run well with temperature dependent viscosity and density. The OPM settings created by rosim are not yet optimized. This is planned for a future rosim release.

Post processing

Subsidence calculation

For subsidence calculation nuclei of strain Okada inflation sources and a green function solution are used [14] [15].

For the nuclei of strain, the loci are in agreement with the reservoir cells. The volume change of each cell $\Delta V(t)_i$ is given by temperature and stress change (ΔT and ΔP) of cell *i* with volume V_i

$$\Delta V(t)_i = e_{zz}(t)_i V_i$$

$$e_{zz}(t)_i = \Delta T(t)_i C_{thermal}(l) + \Delta P(t)_i C_m(l)$$

$$C_{thermal}(l) = \alpha \frac{(1+v)}{(1-v)}(l)$$

$$C_m(l) = \frac{(1-v-2v^2)}{(1-v)E}(l)$$

Where *E* is Young's modulus and *v* is Poisson's ratio of layer l in the model. $C_{thermal}(l)$ and $C_m(l)$ are the thermal and mechanical compaction coefficient. α is the linear thermal expansion coefficient of layer l. $C_{thermal}(l)$ and $C_m(l)$ are specified through the user interface.

The subsidence is calculated at surface positions (z=0) in agreement with the horizontal resolution of the reservoir grid cells. For the subsidence calculation in the okada points source formulation we assume uniform subsurface poro-elastic parameters, i.e. specified in the user interface as Young's modulus and Poisson's ratio.

Preferentially ROSIM applies an upscaling (lumping) procedure averaging the strains in the reservoir cells in vertical direction, and assuming in horizontal direction no downscaling from the grid dimensions. The upscaling is justified by the fact that the distribution of compaction in a relatively thin and deep reservoir (compared to its depth) an has a negligible effect on the surface subsidence [16]. The green function solution's computational performance benefits considerably from the vertical upscaling option, increasing computational speed by a factor equal to the vertical dimension of the grid.

ResInsight

Information can be found here: resinsight.org. Please review for guidelines how to manipulate and visualize the ROSIM results. Some standard viewing steps are described in the Tutorials section.

Application GUI description

Double clicking ‘rosim.exe’ will open the rosim gui application. the Below all the GUI item are described. Behind the input parameters a default value and the unit are put in square brackets. For table (spreadsheet) inputs data can be copy pasted from and to excel, right click shows extra functionality.
A ‘rosim.log’ file will be created next to the .exe file when the program is started. The logging output, also the messages in the calculation progress window, will be recorded here. This can especially be helpful if something unexpected happens, if the application crashes for instance. Each time the program is restarted this file will be cleared, so make sure to rename this log file if you want to keep it.

Top buttons

See Figure 3.

- ***New Project***
  Create new project with default settings.
- ***Load Project Settings***
  Load a ROSIM settings file (.xml).
- ***Save Project Settings***
  Save settings to a ROSIM settings file (.xml), this saved the settings only, output data is saved automatically after each calculation and stored in the ‘output scenario’ folder.
- ***View Grid***
  View 2d grids, see Input grids for format options.
- ***About***
  App info.
- ***Prepare Flow Model***
  Create flow input decks (containing the 3D static model and well info) for DoubletCalc and OPM. Any existing output for the selected scenario will be deleted first.
- ***View Static Model***
  Shows the 3D static model in ResInsight (from the scenario folder opens the file: <name>_STATIC_MODEL\<name>_STATIC_MODEL.EGRID).
- ***Run Flow Simulation***
  Run the selected simulator.
- ***View in ResInsight***
  Opens selected output in ResInsight (grayed out if not available).

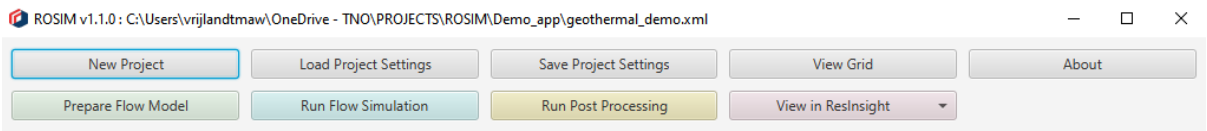


Figure 3: Top buttons

General Settings

See Figure 4.

- ***output (results) directory***
  Directory that contains the ‘output scenario’ data folders.

- ***select output scenario***
  Drop-down showing all 'output scenarios' available in the output (results) directory, the *silent overwrite* toggle disables the warning pop-up for overwriting existing results, the *add new scenario* button creates a new .rosim folder, the *load run settings* button loads the settings that were used for *Prepare Flow Model* of the selected scenario.
- ***opm run environment***
  For running OPM choose between 'WSL Environment' or 'Docker Container', see OPM installation.

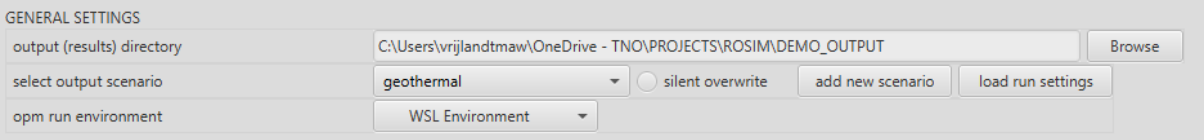


Figure 4: General Settings

Input 3D Static Model Tab

See Figure 5.

CUSTOM GRDECL

- ***use custom grdecl file [no]***
  Option to use an existing 3D static model (for instance from Petrel) in grdecl format.
- ***custom grdecl file***
  File location.

XY EXTENT

- ***x minimum (edge of grid cell) / y minimum (edge of grid cell) [0 m]***
  X-value (Y-value) of the left (bottom) edge of the first X-cell (Y-cell) of the 3D model.
- ***x spacing (grid cell size) / y spacing (grid cell size) [50 m]***
  Standard X and Y grid size, local grid refinement can decrease the spacing around the wells.
- ***x number of cells / y number of cells [60]***
  Standard X and Y number of cells, local grid refinement can increase this number.
- ***x maximum (edge of grid cell) / y maximum (edge of cell) [3000 m]***
  X-value (Y-value) of the right (top) edge of the last X-cell (Y-cell) of the 3D model, this is calculated automatically from the two above.
- ***number of grid cells (excluding grid refinement) [122400]***
  Calculated automatically from the X/Y/Z number of cells
- ***z number of cells [34]***
  Number of vertical cells, sum of the vertical cells in the model layers.

LAYERING

- ***model top depth [2000 m]***
  Start of the model in depth, can be a 2D varying grid.
- ***directory for layers grids (optional)***
  Directory from which grids can be loaded for layer properties in the item below.

- ***model layers***
  Table with each row representing a layer. Many columns can be input from a grid, right click to select a grid. Copy-paste from excel is possible.
  - ***layer name [default layer]***
  - ***vertical cells [20]***
    Number Z cells in the layer.
  - ***geometry option [thickness]***
    Define layer by thickness or base depth.
  - ***thickness/base depth [m]***
    Depending on previous column
  - ***actnum [1]***
    Determine whether (part of) the layer is active in the simulation, can be a grid. Can be 0 (not active) or 1 (active).
  - ***initial (surface) temperature [11 °C]***
    Surface temperature, can be a grid. This is equal to the initial temperature if the gradient in the next column is 0: Temperature = initial temp + gradient * depth.
  - ***gradient [34 °C/km]***
    Can be a grid.
  - ***porosity [0.12]***
    Can be a grid.
  - ***net to gross [0.9]***
    Can be a grid.
  - ***perm x [400 mD]***
    Permeability in X direction, can be a grid.
  - ***perm y [400 mD]***
    Permeability in Y direction, can be a grid.
  - ***perm z [40 mD]***
    Permeability in Z direction, can be a grid.
  - ***rock matrix thermal conduct [4.0 W/Km]***
    Rock matrix thermal conductivity, can be a grid. The bulk conductivity is derived from the rock matrix and water thermal conductivities and the porosity.
  - ***water thermal conduct [0.6 W/Km]***
    Water thermal conductivity, can be a grid.
  - ***salinity [70000 ppm]***
    Constant per layer.
  - ***rock matrix heat capacity [1000 J/kgK]***
    Rock matrix heat capacity, constant per layer. The bulk heat capacity is derived from the rock matrix and water (from Batzle&Wang) heat capacities, the porosity and the rock matrix density.
  - ***rock matrix density [2700 kg/m^3]***
    Constant per layer.
- ***faults (optional)***
  Table with each row representing a fault.
  - ***fault name [fault 1]***
  - ***shape file [select shape file..]***
    Shape file ('.shp') in the 'directory for layers grid' defining the trajectory of the fault. Multiple faults are possible, the top fault has preference in case of crossing faults.

- ***porosity [fraction]***
  Porosity to be given to the cells on the fault trajectory.
- ***permeability [mD]***
  Permeability to be given to the cells on the fault trajectory.
- ***rock matrix thermal conduct [W/Km]***
  Rock matrix thermal conductivity be given to the cells on the fault trajectory.

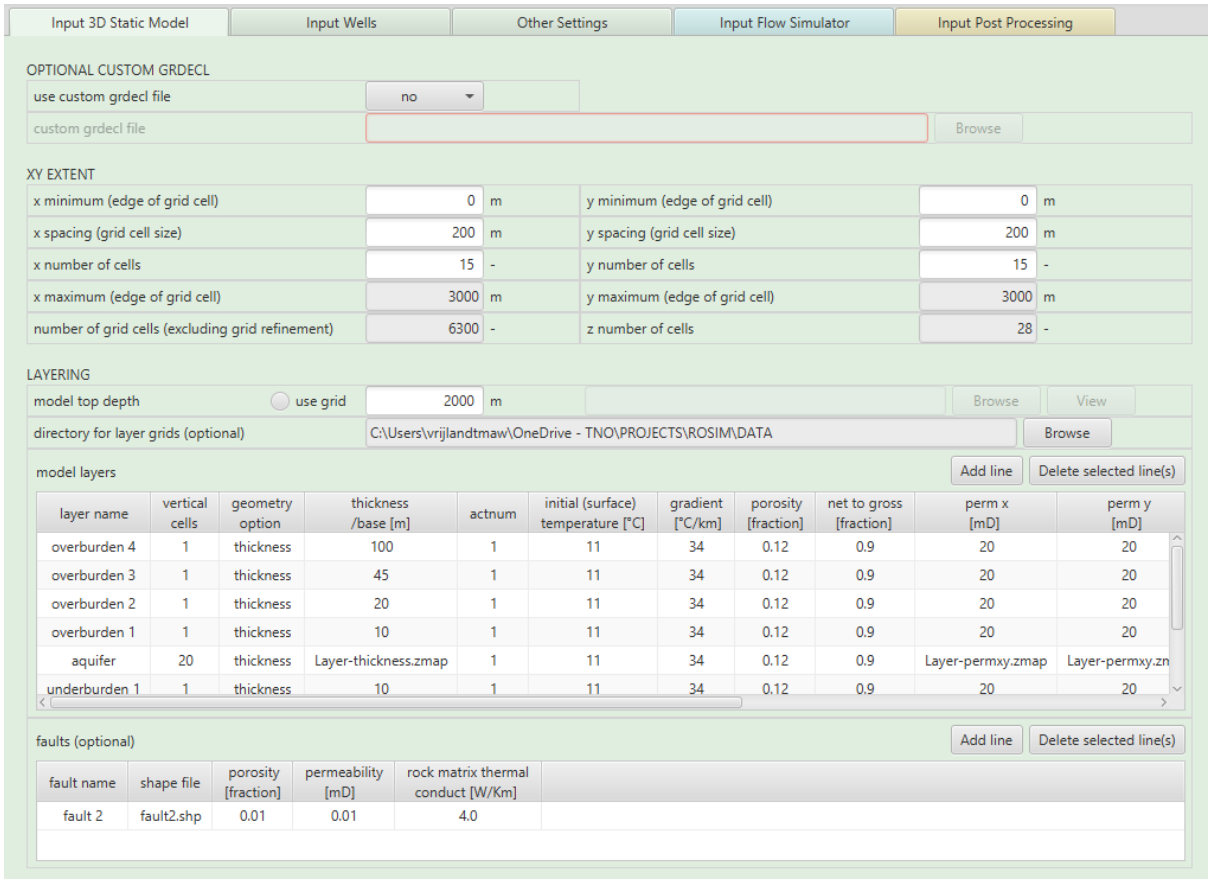


Figure 5: Input 3D Static Model Tab

Input Wells Tab

See Figure 6.

SIMULATION PERIOD

- ***simulation start date [1-1-2020]***
- ***simulation end date [1-1-2050]***
  This will override any later items in the flow constraints schedule.
- ***reporting step time unit [year]***
  Unit for the reporting step increment (next item).
- ***reporting step increment (=doubletcalc calc step) [1]***
  Number of units (previous item) to be used for the reporting step, meaning the output interval for temperature and pressure. Dates at which the flow constraints change are added as reporting step. For DoubletCalc this reporting step is also the calculation step.

WELLS

- ***well name [name]***
- ***x coordinate well head [1025 m]***
  X-coordinate of the top of the well trajectory. To avoid issues with the well connectivity factor calculation, well head X and Y are shifted slightly if they fall exactly on top of a grid cell edge.
- ***y coordinate well head [1525 m]***
  Y-coordinate of the top of the well trajectory.
- ***rotary kelly bushing (RKB height above msl) [0 m]***
  height of the drilling floor above mean sea level, this is the point from which the vertical depth (below RKB) in the well trajectory is measured.
- ***xy grid size target (0=no grid refinement) [0 m]***
  XY grid size around the well, achieved by (tartan) local grid refinement. The actual grid spacing can deviate somewhat because the original grid cell is divided into equal sized cells. A value of 0 means no grid refinement.
- ***grid refinement range [3 grid cells]***
  Number of original grid cells in X and Y to which the grid refinement is applied. The refinement is fading away from the well.
- ***diameter [8.5 inch]***
  Diameter of the well bore in the open section.
- ***skin [0]***
  Well skin in the open section.
- ***well trajectory (deviation survey)***
  Table with the well trajectory or deviation survey, each row is representing a deviation point. Straight lines are assumed between the deviation points. Only the part of the trajectory between the top and base of the open hole section is used. Copy-paste from excel is possible. Wells should not be defined to follow the boundary between grid blocks, because then it becomes unpredictable in which grid block the well will be placed.
  - ***vertical depth (below RKB) [3000 m]***
    Positive vertical depth below rotary Kelly bushing.
  - ***X offset [0 m]***
    X distance from the well head X coordinate, positive is East, negative is West.
  - ***Y offset [0 m]***
    Y distance from the well head Y coordinate, positive is North, negative is South.
  - ***Z [3000 m]***
    Positive vertical depth below mean sea level, calculated automatically.
  - ***MD [3000 m]***
    Positive measured depth along hole below RKB, calculated automatically.
- ***Open section selection mode [Along hole MD]***
  Options: `Along hole MD`, `TVDSS`, `By Layer`.
- ***top open section MD's, comma separated [2200]***
  Measured depth along hole from RKB to the top of the open (perforated) section, can be multiple sections, comma separated.
- ***base open section MD's, comma separated [2250]***
  Measured depth along hole from RKB to the base of the open (perforated) section, can be multiple sections, comma separated.

- ***top open section depths, comma separated [2200]***
  True vertical depth below sea level to the top of the open (perforated) section, can be multiple sections, comma separated.
- ***base open depths, comma separated [2250]***
  True vertical depth below sea level to the base of the open (perforated) section, can be multiple sections, comma separated.
- ***open section layer names, comma separated [default layer]***
  Names of the layers that are open (perforated) in the well, can be multiple layers, comma separated.
- ***open section layer margins, comma separated [10]***
  Margin [m] from the top and base of the layer that are open in the well, can be multiple sections, comma separated. If the layer extends from 2000 to 2200 m depth, and the margin is 25, then the open section will be from 2025 to 2175 m depth.
- ***flow constraints schedule***
  Table with the flow constraints schedule. Each row indicates a change in flow constraints. Copy-paste from excel is possible.
  - ***year [2022]***
    The starting date of the specified constraints. For DoubletCalc this date is rounded down to the preceding reporting step.
  - ***month [1]***
  - ***day [1]***
  - ***target flow rate (+ve inj) [200 m3/h]***
    Target flow rate, positive is injection, negative is production. For DoubletCalc this is the actual flow rate. For OPM this is the target flow rate, obtained if the pressure constraint (next column) allows it. For DoubletCalc it is required to have 0 net flow.
  - ***bhp min (prod) or max (inj) [175 Bar]***
    Pressure constraint, minimum pressure for production, maximum pressure for injection. This value is ignored by DoubletCalc.
  - ***injection temp (prod: -1) [30 °C]***
    Injection temperature of the injected water. During production this value is ignored by OPM. For DoubletCalc this value during production is the cut-off temperature: the production rate is set to 0 after the first time step that the production temperature drops below the cut-off temperature. A value of -1 means that no cut-off temperature will be used.

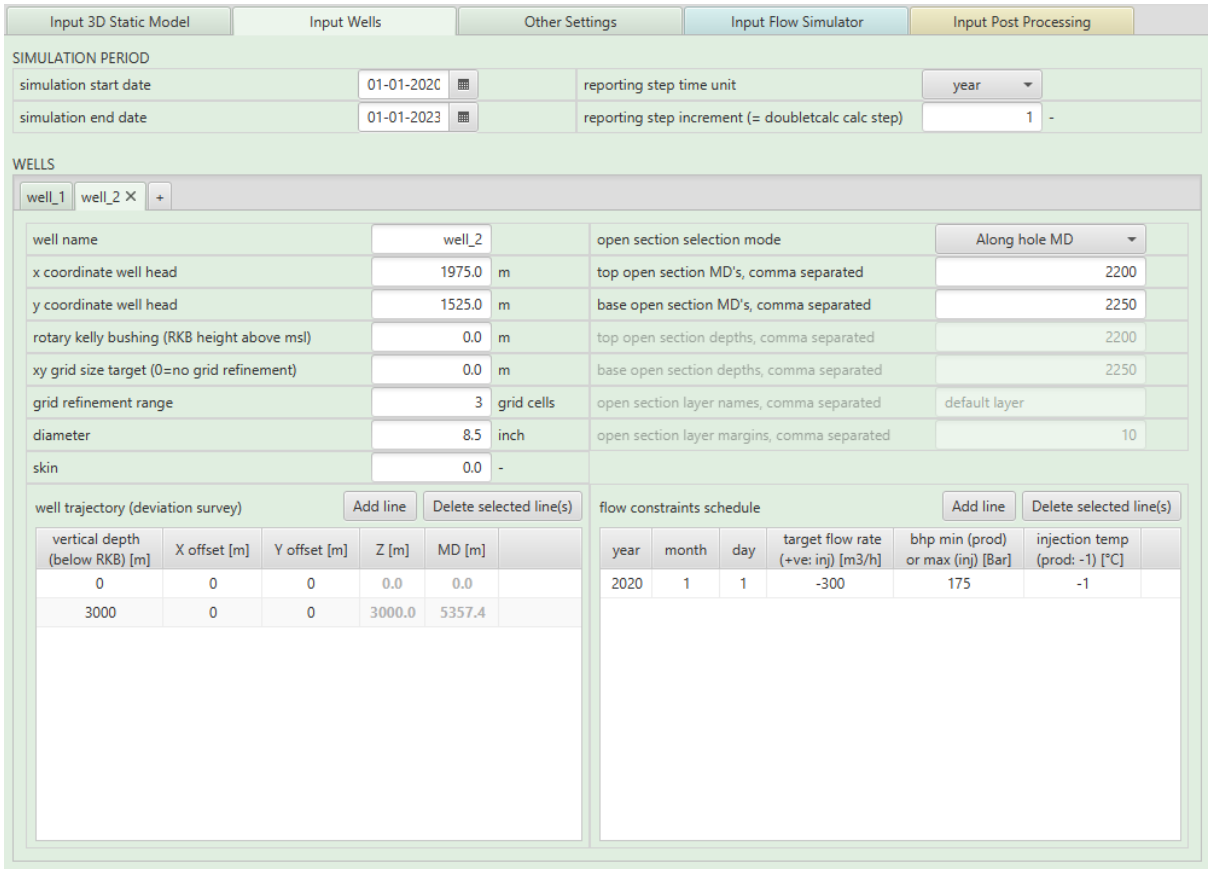


Figure 6: Input Wells Tab

Other Settings Tab

See Figure 7.

OUTPUT OPTIONS

- ***auto-show static model in resinsight [yes]***
  Option automatically open the created static model in ResInsight after *Prepare Flow Model*.

EQUILIBRIUM INITIALIZATION DATA AND BOUNDARY CONDITIONS

- ***initialization reference depth [2200 m]***
  Depth for the initialization reference pressure, see below. This value has to be within the depth range of the 3D static model. Calibration point for the initial 3D pressure field.
- ***initialization reference pressure [220 Bar]***
  Pressure at the initialization reference depth, see above.
- ***no fluid flow boundary (doubletcalc) [yes]***
  Option to set the fluid flow boundary, only for DoubletCalc. If 'no' then a constant pressure boundary condition is used, and fluid is allowed to flow out of and into the model. OPM always uses a no fluid flow boundary.
- ***model top/bottom temperature (doubletcalc) [no heat flow]***
  Option to set the top/bottom temperature boundary condition, only for DoubletCalc. Options: `no heat flow`, `fix top/bottom temp`, `wall cooling`.

LICENSE OUTLINE

- ***license shape file (optional)***
  A 3D static property 'OPERNUM' will be created whose cells will be:
  - 0: cell outside the shape file polygon
  - 1: cell in the shape file polygon
  - 2: cell partly in the shape file polygon
  Using a property filter in ResInsight the pressure or temperature evolution inside the license (or on the boundary) can be displayed.

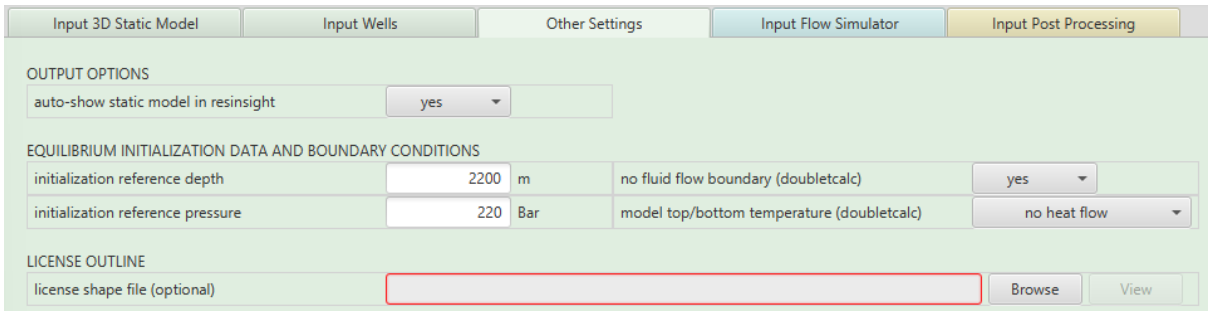


Figure 7: Other Settings Tab

Input Flow Simulator Tab

FLOW INPUT DECK

See Figure 8.

- ***flow simulator option [DoubletCalc]***
  Select a flow simulator option: DoubletCalc, OPM (ROSIM deck) or OPM custom deck. DoubletCalc and OPM use the flow input deck created by ROSIM. For OPM you can choose to run a custom deck: created outside ROSIM or a modified ROSIM deck.
- ***opm flow option [flow_onephase_energy]***
  Select OPM flow version. Flow_onephase_energy for single phase flow (relatively recent addition to OPM), quicker than the standard 3-phase OPM flow version and is preferred.
- ***opm flow custom input deck file***
  Possibility to run a custom OPM deck (not created by ROSIM, or an altered ROSIM deck). A new folder with the simulation output is created next to the selected input deck.

OUTPUT

- ***auto-show output in resinsight [yes]***
  Option to automatically open the dynamic model in ResInsight after *Run Flow Simulation*.

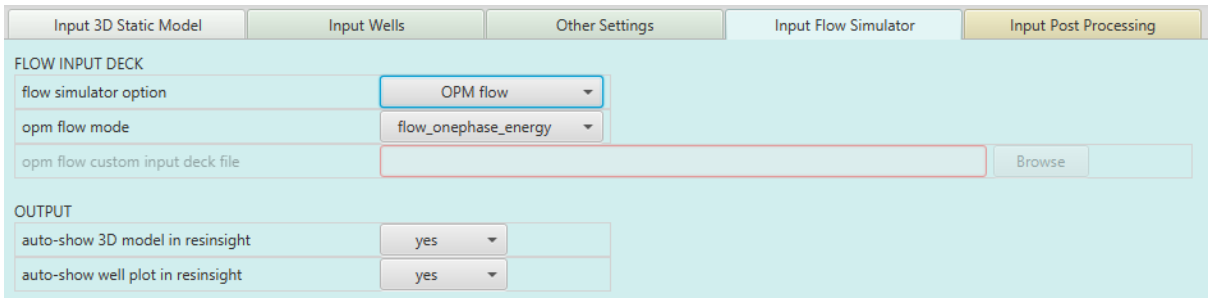


Figure 8: Input Flow Simulator Tab

Input Post Processing Tab

INPUT OUTPUT

See Figure 9.

- ***post processing input .egrid file***
  Select the input file on which to calculate the post processing step, can be any (post processed) egrid file, as long as it has the required properties as created by a DoubletCalc3D or OPM flow simulation.
- ***post processing output name [scenario_name_post_processed]***
  Output file name of the post processed .egrid file, the output will file will be transformed to all caps and .egrid extension will be added.
- ***auto-show results in resinsight***
  Option to automatically open the dynamic model in ResInsight after *Run Post Processing*.

CALCULATION MODE

- ***post processing mode [Subsidence]***
  Select which post processing calculation to run. In the standard (non-beta) version, there is only one option.

CONSTANT PARAMETERS (Subsidence option)

- ***subsidence Young's Modulus [20 GPa]***
  Young's modules value used in the compaction and subsidence calculation.
- ***subsidence Poisson's Ratio [0.35]***
  Young's modules value used in the compaction and subsidence calculation.
- ***subsidence lump sources z-dir [yes]***
  Option to lump the sources in the z direction for the compaction and subsidence calculation.

LAYER PROPERTIES (Subsidence option)

- **layer properties (number of layers must match number of 3D static model layers)**
  Table with each row representing a layer. The number of rows must be equal to the number of rows in de 'model layers' table in the 'Input 3D Static Model' tab. For non-rosim egrid: the number of rows must match the number of SATNUM regions (i.e the maximum INT value in the SATNUM keyword).
    - **compaction coefficient [1 [1e-5/Bar]]**
      Compaction coefficient: should be zero outside the reservoir, a warning will be given if the top and/or bottom layer has a non-zero value.
    - **thermal compaction coefficient [1.67 [1e-5/K]]**
      thermal compaction coefficient

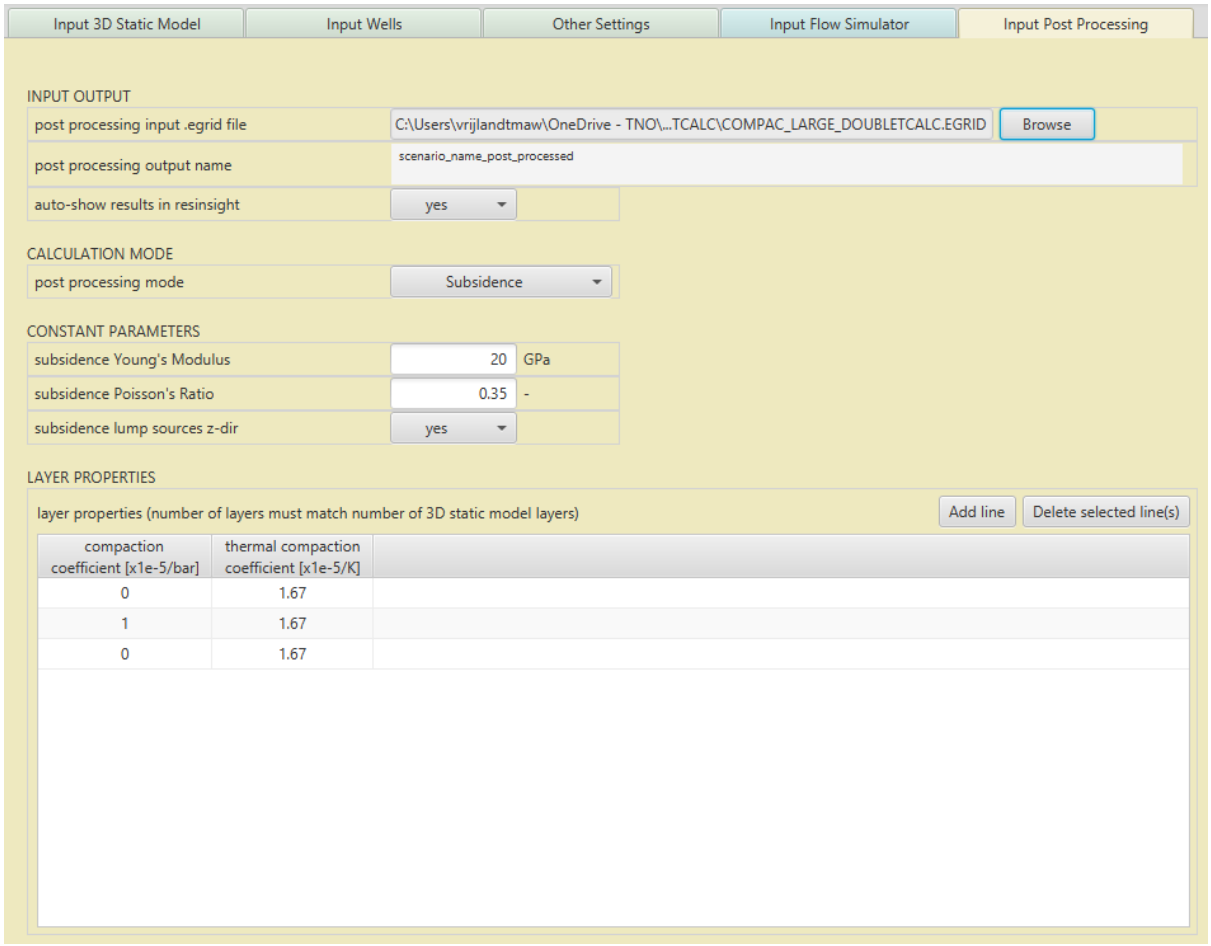


Figure 9: Input Post Processing Tab

Run in Batch Mode

It is possible to run rosim in batch mode, with Java 17 installed, using the 'rosim-batch-x.x.x.jar' file, the '-h' help option gives possible arguments, see below (The '*WARNING: sun.reflect.Reflection.getCallerClass is not supported. This will impact performance.*' message can be ignored.):

```
C:\Users\vrijlandtmaw\OneDrive - TNO\PROJECTS\ROSIM\batch>java -jar rosim-batch-1.1.0.jar -h
WARNING: sun.reflect.Reflection.getCallerClass is not supported. This will impact performance.
usage: java -jar <rosim jar> -f <rosim settings xml file> [options]"
Run GUI application: java -jar <rosim jar>.
ROSIM, a TNO geothermal subsurface model builder and reservoir simulator."

Recognized options:
 -f,--rosim-settings-file <arg>   ROSIM settings xml file (required)
 -h,--help                        Print this help message and exit
 -l,--log-level <arg>             Log level options: 'debug', 'info'
                                  (default), 'warn', 'error'
 -lf,--log-file <arg>             Log level file path: no logging to file
                                  if not specified
 -o,--output-dir <arg>            output directory, if not given use
                                  directory and scenario name in settings
                                  file
```

The 'Prepare Flow Model' and 'Run Flow Simulation' steps can be run automatically:

`

java -jar rosim-batch-x.x.x.jar -f newProject.xml -lf test-batch.log -o "C:/Users/vrijlandtmaw/OneDrive - TNO/PROJECTS/ROSIM/batch/test.rosim"

`

This will create a static and flow model and perform a flow simulation with the settings in 'newProject.xml'. A log file, 'test-batch.log', is created, and the results will be placed in the output rosim scenario folder specified in the last argument '-o':

```
C:\Users\vrijlandtmaw\OneDrive - TNO\PROJECTS\ROSIM\batch>java -jar rosim-batch-1.1.0.jar -f newProject.xml -lf test-bat
ch.log -o "C:/Users/vrijlandtmaw/OneDrive - TNO/PROJECTS/ROSIM/batch/test.rosim"
WARNING: sun.reflect.Reflection.getCallerClass is not supported. This will impact performance.
2023-Oct-06 11:57:33,386 INFO  rosim.ROSIMBatch:168 - OUTPUT FOLDER: C:\Users\vrijlandtmaw\OneDrive - TNO\PROJECTS\ROSIM
\batch\test.rosim
2023-Oct-06 11:57:33,393 INFO  rosim.ROSIMBatch:174 - =====  generate static model and flow input deck  =====
2023-Oct-06 11:57:33,395 INFO  tno.geoenergy.gui.ProgressTrackerConsole:55 - Initializing static model


2023-Oct-06 11:57:34,157 INFO  tno.geoenergy.gui.ProgressTrackerConsole:55 - Creating 3D GRDECL grid

2023-Oct-06 11:57:34,333 INFO  tno.geoenergy.gui.ProgressTrackerConsole:55 - Calculating well indices

2023-Oct-06 11:57:34,341 INFO  tno.geoenergy.gui.ProgressTrackerConsole:55 - Creating flow input files

2023-Oct-06 11:57:34,413 INFO  tno.geoenergy.gui.ProgressTrackerConsole:55 - Creating Static Model EGRID files

2023-Oct-06 11:57:34,825 INFO  tno.geoenergy.gui.ProgressTrackerConsole:55 -
The following output has been created:
C:\Users\vrijlandtmaw\OneDrive - TNO\PROJECTS\ROSIM\batch\test.rosim\test_OPM_FLOW.GRDECL
C:\Users\vrijlandtmaw\OneDrive - TNO\PROJECTS\ROSIM\batch\test.rosim\test_DOUBLETCALC.GRDECL
C:\Users\vrijlandtmaw\OneDrive - TNO\PROJECTS\ROSIM\batch\test.rosim\test_OPM_FLOW.DATA
C:\Users\vrijlandtmaw\OneDrive - TNO\PROJECTS\ROSIM\batch\test.rosim\test_OPM_FLOW_ONEPHASE.DATA
C:\Users\vrijlandtmaw\OneDrive - TNO\PROJECTS\ROSIM\batch\test.rosim\test_OPM_FLOW.DATA
C:\Users\vrijlandtmaw\OneDrive - TNO\PROJECTS\ROSIM\batch\test.rosim\test_OPM_FLOW_ONEPHASE.DATA
C:\Users\vrijlandtmaw\OneDrive - TNO\PROJECTS\ROSIM\batch\test.rosim\test_DOUBLETCALC.DCINPUT
C:\Users\vrijlandtmaw\OneDrive - TNO\PROJECTS\ROSIM\batch\test.rosim\test_STATIC_MODEL (directory)
C:\Users\vrijlandtmaw\OneDrive - TNO\PROJECTS\ROSIM\batch\test.rosim\test_OPM_FLOW_last_run_settings.xml


2023-Oct-06 11:57:34,826 INFO  rosim.calc.GenStaticModelDeck:345 - ===== GenStaticModelDeck.java finished =====

2023-Oct-06 11:57:34,894 INFO  rosim.ROSIMBatch:184 - =====  flow modeling  =====
2023-Oct-06 11:57:35,912 INFO  tno.geoenergy.gui.ProgressTrackerConsole:55 - simulating timestep 0 (of 30)

2023-Oct-06 11:57:37,206 INFO  tno.geoenergy.gui.ProgressTrackerConsole:55 - simulating timestep 1 (of 30)

2023-Oct-06 11:57:40,368 INFO  tno.geoenergy.gui.ProgressTrackerConsole:55 - simulating timestep 2 (of 30)
```

File formats

ROSIM can use 2D property grids as input, uses an xml file to store settings and outputs data to a .rosim folder which contains simulation results in egrid format.

Input grids

The *View Grid* button can be used to check whether a grid can be read by ROSIM. Input/output from several software packages:

- Petrel: read and export as Zmap (.zmap)
- ArcMap/QGIS: read and export to ESRI Asciigrid (.asc), Zmap (.zmap) and GeoTIFF (.tif)

Below is an overview of compatible grid file formats (in the title parenthesis are the associated file extensions).

Surfer (.grd)

A Golden Software Surfer ASCII grid contains 5 lines of of header information at the beginning of the file followed by the cell value data stored in row major order, starting with the southernmost row from west to east. Each row has the same y-coordinate, from ymin (1st row) to ymax (nth row). The x-coordinates in each row start with xmin and end with xmax.

line	contents
1	DSAA
2	Number of colums, number of rows
3	Minimum x-coordinate, maximum x-coordinate
4	Minimum y-coordinate, maximum y-coordinate
5	Minimum z-value, maximum z-value

Example Surfer ASCII grid

DSAA
101 101
624325 629325
5788025 5793025
10 30
23.42 21.1 14.56 [..] 17.54
26.09 29.51 28.42 [..] 21.76
10.57 26.2 17.26 [..] 19.66
[..]
10.98 23.16 21.81 [..] 20.3

ESRI Asciigrid (.asc, .arc)

An ESRI ASCII grid contains 6 lines of of header information at the beginning of the file followed by the cell value data stored in row major order, starting with the northernmost row from west to east. Each row has the same y-coordinate, from ymin (1st row) to ymax (nth row). The x-coordinates in each row start with xmin and end with xmax.

line	parameter	description	requirements
1	NCOLS	number of cell columns	integer greater than 0
2	NROWS	number of cell rows	integer greater than 0
3	XLLCENTER or XLLCORNER	X coordinate of the origin (by center or lower left corner of the cell).	match with Y coordinate type
4	YLLCENTER or YLLCORNER	Y coordinate of the origin (by center or lower left corner of the cell)	match with X coordinate type
5	CELLSIZE	cell size	greater than 0
6	NODATA_VALUE	the input values to be NoData in the output raster	optional. Default is -9999

Example ESRI ASCII grid:

ncols 101
nrows 101
xllcorner 624300
yllcorner 5788000
cellsize 50
nodata_value -9999
10.98 23.16 21.81 [..] 20.30
27.70 22.54 13.96 [..] 29.48
10.40 26.87 22.26 [..] 25.64
[..]
23.42 21.10 14.56 [..] 17.54

ZMap/ZYCOR (.zmap, .zycor)

A Landmark ZYCOR Zmap+ ASCII grid contains 5 lines of header information at the beginning of the file. The header both starts and ends with the '@' character. Optionally the header is preceded by a number of comment lines all starting with '!'). The header is followed by the cell value data stored in column major order, starting with the westernmost column from north to south. The number of cell values per line does not exceed the last number in the first header line.

line	parameter description
1	'@', user defined text, 'GRID' keyword, maximum number of cell values per line
2	'15', no data value, empty field, '1', '1'
3	number of columns, number of rows, minimum and maximum x-coordinates, minimum and maximum y-coordinates
4	cell size in x-direction, cell size in y-direction, cell size in z-direction

Example Zycor ZMAP+ ASCII grid:

@testgrid_out.asc HEADER, GRID, 5
15, 0.1000000E+31 , ,7, 1
101, 101, 624325, 629325, 5788025, 5793025
50, 50
@

10.98 23.16 21.81 [..] 20.30
27.70 22.54 13.96 [..] 29.48
10.40 26.87 22.26 [..] 25.64
[..]
23.42 21.10 14.56 [..] 17.54

GeoTIFF (.tif, .tiff)

This format can be used to export from QGIS, a description can be found here: https://en.wikipedia.org/wiki/GeoTIFF.

Settings XML file

The ROSIM GUI settings are stored (*Save Project Settings*) in a .xml file.

Output scenario folder

When a new output scenario is created, a .rosim scenario folder is created. This folder will hold the output data for this scenario. This folder can contain the following files and folders:

Created by *Prepare Flow Model*:

- **<name>_OPM_FLOW.DATA**
  OPM flow input deck for 3-phase flow
- **<name>_OPM_FLOW_ONEPHASE.DATA**
  OPM flow input deck for onephase flow
- **<name>_OPM_FLOW.GRDECL**
  3D static model, part of the OPM flow input deck
- **<name>_DOUBLETCALC.DCINPUT**
  DoubletCalc input data
- **<name>_DOUBLETCALC.GRDECL**
  3D static model, part of the DoubletCalc input data
- **<name>_last_run_settings.xml**
  setting file of the last *Prepare Flow Model* run
- **<name>_STATIC_MODEL (folder)**
  Static model folder
- **<name>_STATIC_MODEL\<name>_STATIC_MODEL.EGRID**
  Static model egrid file, this file is loaded by ResInsight after *Prepare Flow Model* (if selected in the *Other Settings Tab*) and *View Static Model*.

Created by *Run Flow Simulation*:

- **<name>_DOUBLETCALC\<name>_DOUBLETCALC.EGRID** and other EGRID files
  For DoubletCalc simulation output
- **<name>_OPM_FLOW\<name>_OPM_FLOW.EGRID** and other EGRID files
  For OPM flow 3-phase simulation output
- **<name>_OPM_FLOW_ONEPHASE\<name>_OPM_FLOW_ONEPHASE.EGRID** and other EGRID files
  For OPM flow single phase simulation output

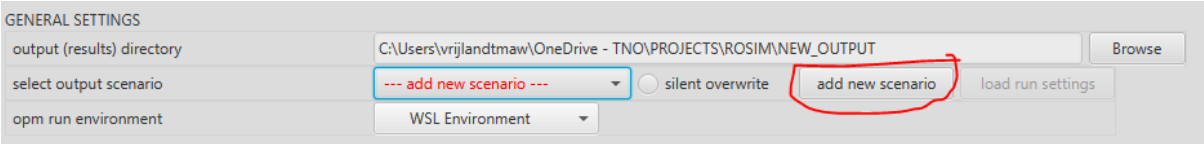
Tutorials

The aim of this section is to introduce new users to the software. First we go through a simple project (constant grid spacing, constant flow, vertical wells) which runs quickly. Then we look at two demo projects (files are provided in the download) with varying grid spacing, deviated wells, property values from grid and varying flow rate. We will go through the ROSIM app and some basic ResInsight viewing functionality. For more ResInsight information please visit https://resinsight.org/getting-started/.

Simple Project

After completing the installation Instructions follow the steps below to run this simple project:

1. Start ROSIM.exe, click the 'New Project' button and select an 'Output Directory' in which the data for the output scenarios will be placed, click 'Finish'.
2. Add a new scenario, for instance 'default_values'. (This will create a 'default_values.rosim' data directory in your 'Output Directory'.)



3. Click the 'Prepare Flow Model' button.
   Uses data from the three green input tabs, for more info on the parameters here see Input 3D Static Model Tab, Input Wells Tab and Other Settings Tab.
   Creates the static model and flow input files: i.e. the files needed for running the flow simulations in either DoubletCalc or an OPM flow version. For more details on which files are created see Output scenario folder.
   Shows the static model automatically ResInsight, this option can be turned off in the 'Other Settings' tab.
4. Click the 'Run Flow Simulation' button. You can also run with OPM flow with the `flow simulator option` in the `Input Flow Simulator` tab.
   Using the input files created in the previous step 'Prepare Flow Model'.
   Runs a flow simulation using the selected simulator in the blue 'Input Flow Simulator' tab, for output files see Output scenario folder section.
   Shows the results automatically shown in ResInsight.
   IMPORTANT: when changing input settings in the green tabs, 'Prepare Flow model' has to be (re)run before 'Run flow simulation'.
5. View results in ResInsight: two windows are opened for the flow modelling results, a 'Plots' window with the well data and a '3D View' showing the 3D model evolution with time.
   - The 'Plots' window shows well pressure, temperature and flow rate with time. Expand the 'Summary Curves' to select which plots to display, see Figure 10. Right click on a curve gives the option to 'Show Plot Data'.
   - The '3D View' window show the 3D models parameters with time. Left mouse click to zoom, middle mouse click to rotate, right mouse click to move.
   - In the 'Project Tree' highlight 'Cell Result' which brings up the 'Property Editor' below where 'dynamic' or 'static' properties can be selected. For now select 'TEMP'.
   - In the 'Project Tree' right click 'Cell Filters' and select 'Range Filter – J Slice'. Highlight this new 'Range Filter' and below in the 'Property Editor' set 'J Start' to '31' and

'Width' to '60'. Now hit the green triangle play button in the top bar to show the temperature evolution with time, see Figure 11.

- When selecting a grid cell (see purple outlined cell in the 3D viewer) the evolution of the grid cell value through time is displayed in the plot below.

6. Alternative scenarios can be run by changing some parameters, adding another scenario and rerunning.
7. To do a Subsidence calculation go to the 'Input Post Processing' tab and select the DoubletCalc output egrid file as 'post processing input .egrid file'. This file can be found in the scenario .rosim folder created in step 2, in the 'scenario_name_DOUBLETCALC' folder. Optionally rename the 'post processing output name', for instance to '<scenario_name>_subsidence'. Click the 'Run Post Processing' button.
   This will create COMPAC and SUBS dynamic properties, see Figure 12.
8. 'Save/Load Project Settings' can be used to store the current settings in an xml file. This will only store settings, not save any generated data. The data is saved on completion of the calculations and stored in the scenario folder created in step 2.

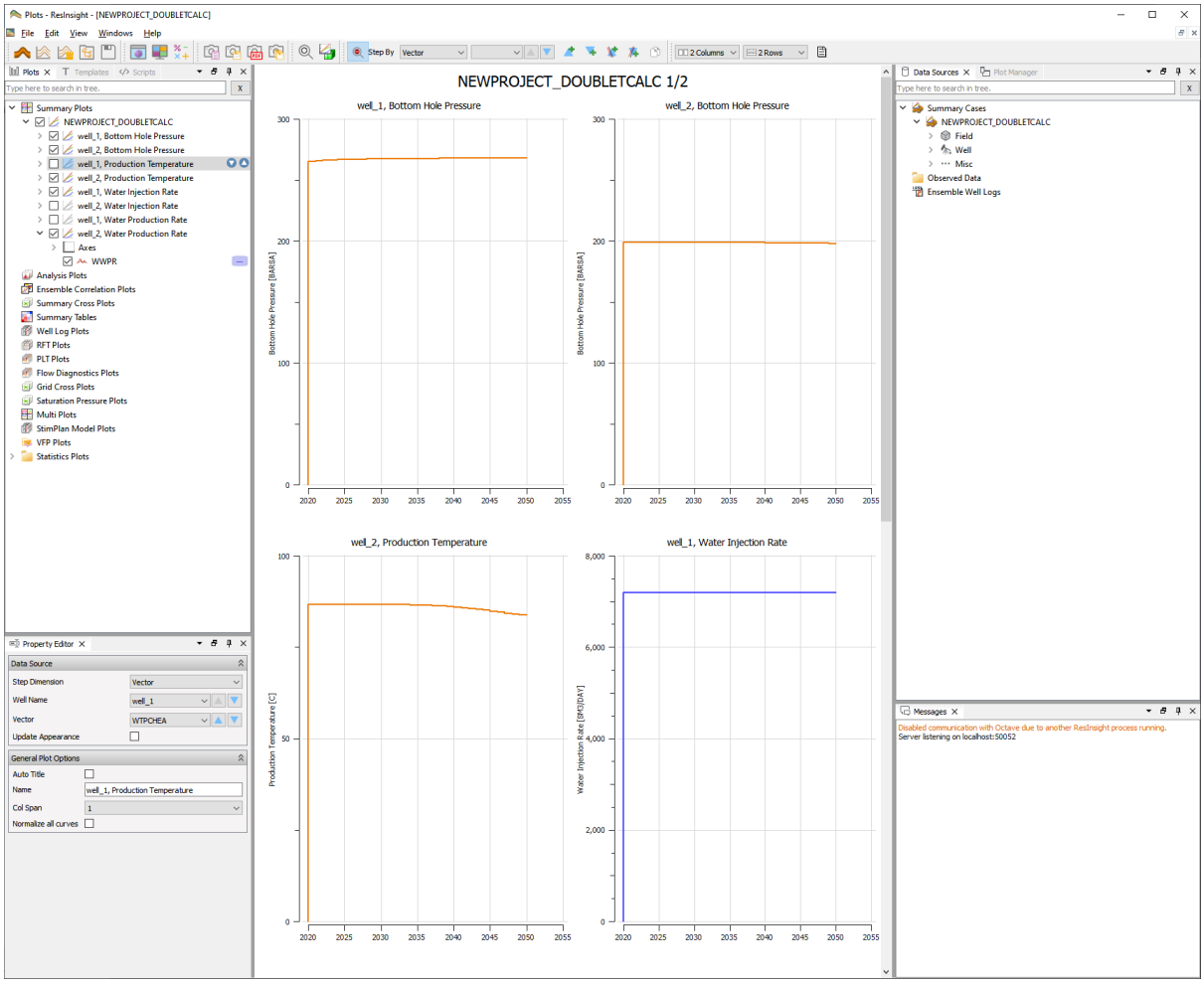


Figure 10: Example ResInsight 'Plots' window

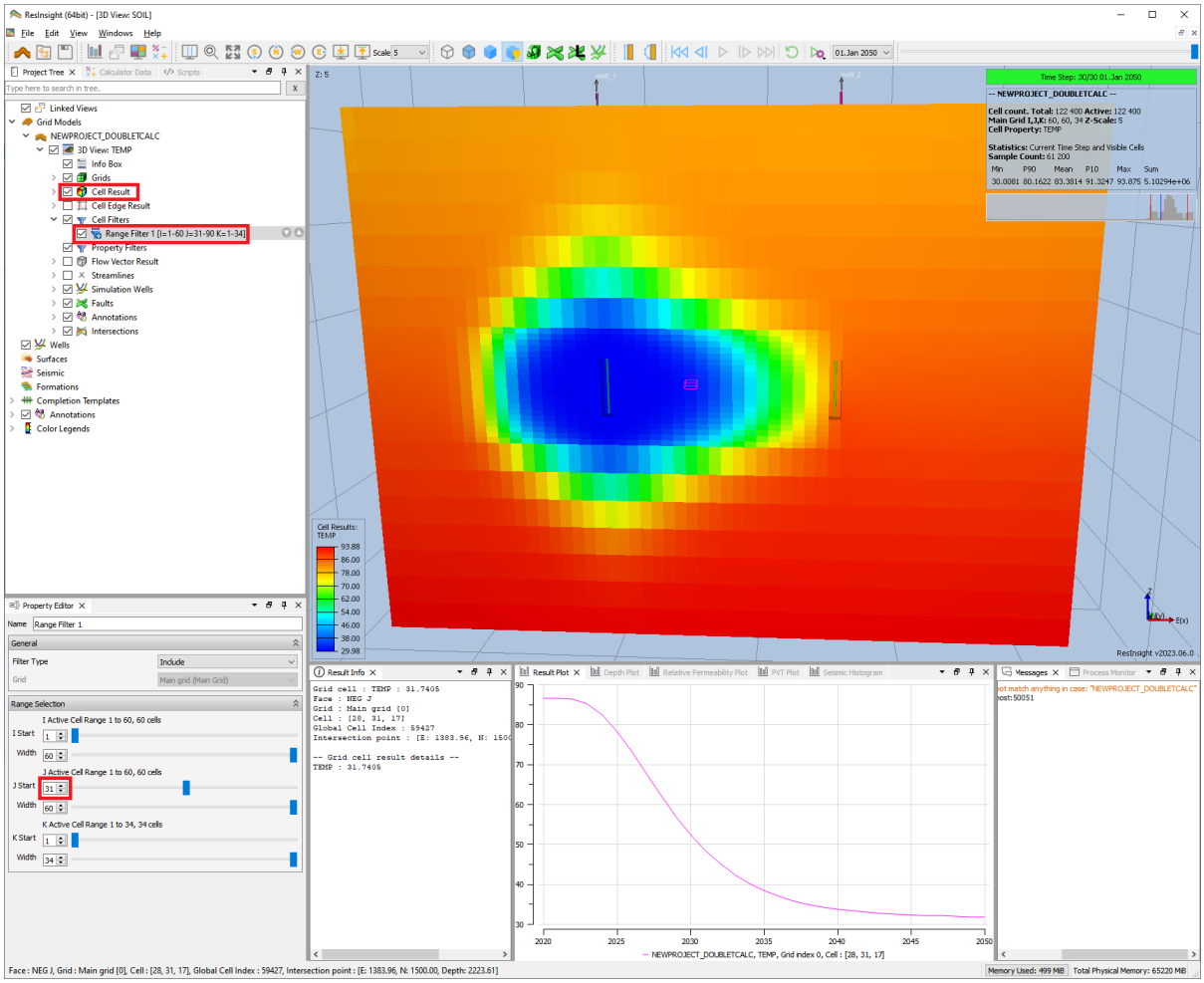


Figure 11: ResInsight '3D View' window for the Simple Model

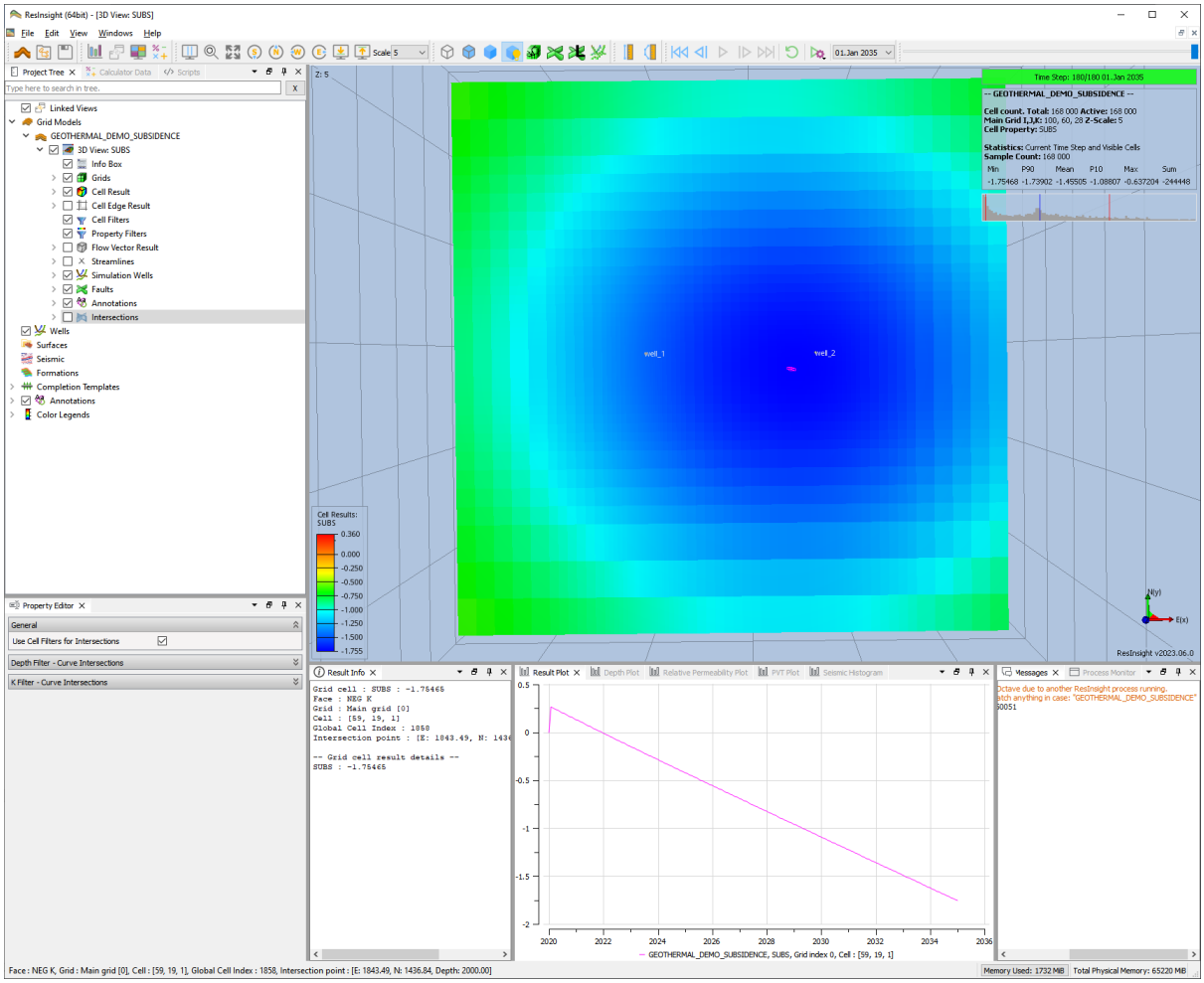


Figure 12: ResInsight '3D View' window for the calculated subsidence after 15 years

Demo Geothermal Project

This geothermal demo project has local grid refinement around the wells (tartan grid), deviated well paths and processes a license boundary shapefile to show the temperature inside, outside and on the license boundary.

1. Click 'Load Project Settings' and select the 'geothermal_demo.xml' file from the 'Demo Projects' folder in the ROSIM download zip.
2. Next select an 'output (results) directory', create or add an output scenario.
3. As 'directory for layer grids (optional)' select the 'Data' folder under 'Demo Projects'. This data folder is necessary because in the 'model layers' spreadsheet 'grid file names' are used as input and a fault 'fault2.shp' is used. Right click to select or view another grid or fault. See Input grids for supported file formats.
4. Under the 'Other Settings' tab, at 'license shape file (optional)' point to the 'license_outline.shp' file in de 'Data' folder.
5. Now continue with 'Prepare Flow Model' and 'Run Flow Simulation', this can take some time. It also possible to simulate with OPM flow.
6. When displaying the temperature evolution as described for the simple project above you can also: right click 'Property Filters', select the newly created filter and below select the 'static' type property 'OPERNUM'. This property has a value of '0' outside the license polygon, '1' inside the polygon and '2' on the polygon edge. By selecting the appropriate values in the

'Filter Settings' the min/max temperature can be displayed within the license area. Make sure to filter only the grid layers (k-values in the range filter described above) of the reservoir, see Figure 13.

7. Please note that only the well path within the 3D model is displayed (in green). In this case the model starts at a depth of 2000m. So even though the well deviation starts at the surface, this is not shown in ResInsight.
8. When viewing from above (Range filter, select K Start '12') the effect of the fault can be displayed, see Figure 14.

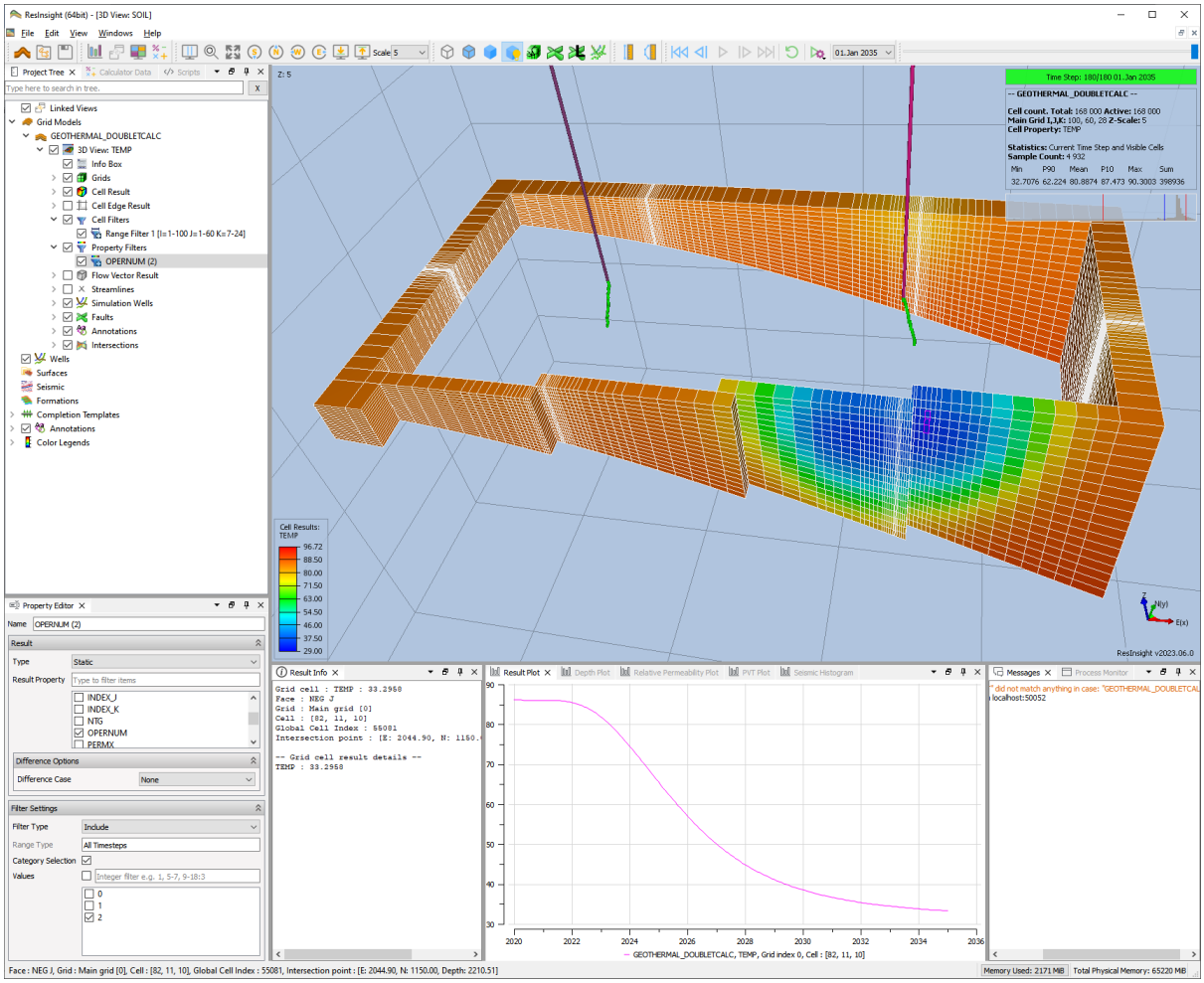


Figure 13: ResInsight '3D View' for the Geothermal Demo model on the license outline cells

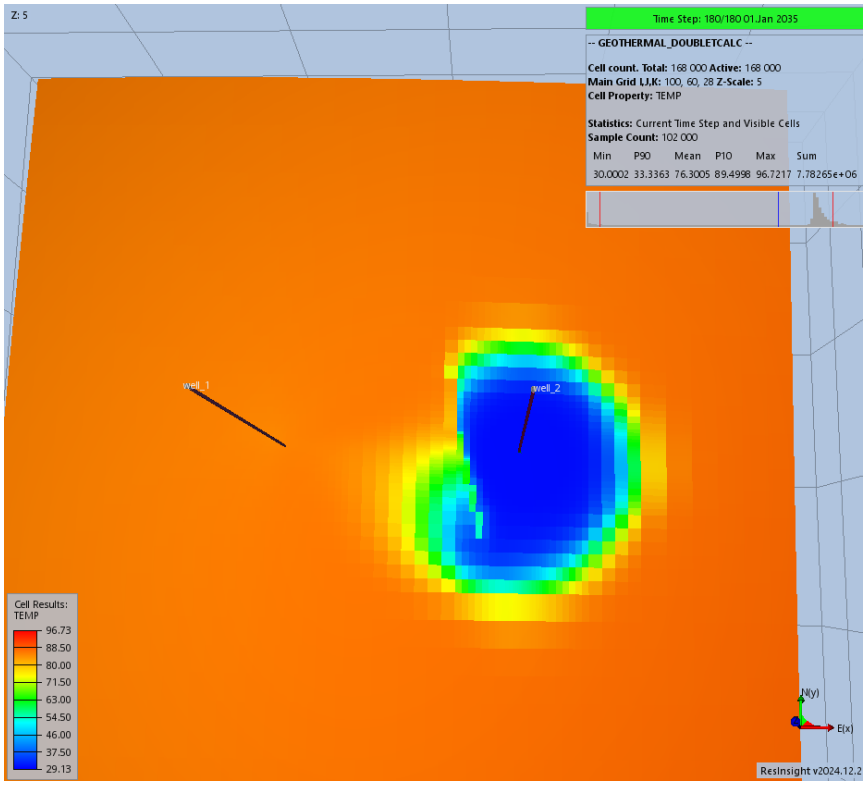


Figure 14: ResInsight '3D View' for the Geothermal Demo model from the top

Demo HTO Project

This high temperature storage demo project has 8 wells: three warm operating wells and five cold compensation wells. Every year consists of three months of injection of hot water in the operating wells, 3 months rest, 3 months of production of hot water in the operating wells and 3 months rest.

1. Click 'Load Project Settings' and select the 'hto_demo.xml' file from the 'Demo Projects' folder in the ROSIM download zip.
2. Next select an 'output (results) directory', create or select an output scenario.
3. Now continue with 'Prepare Flow Model' and 'Run Flow Simulation', this can take some time since there are many months to run. See 'flow constraint schedule' in the 'Input Wells' tab. When simulating with OPM flow it will take a lot of time (> 24h).
4. See Figure 15 for the production temperature (WTPCHEA) evolution in the wells. The production temperature is zero during rest and injection. The operating wells, for instance 'well_3', shows a decreasing production temperature during each winter, when the hot water bubble is drained. The production temperature increases over the years, showing the 'charging' of the HTO. Figure 16 shows the temperature field after 10 years of production.

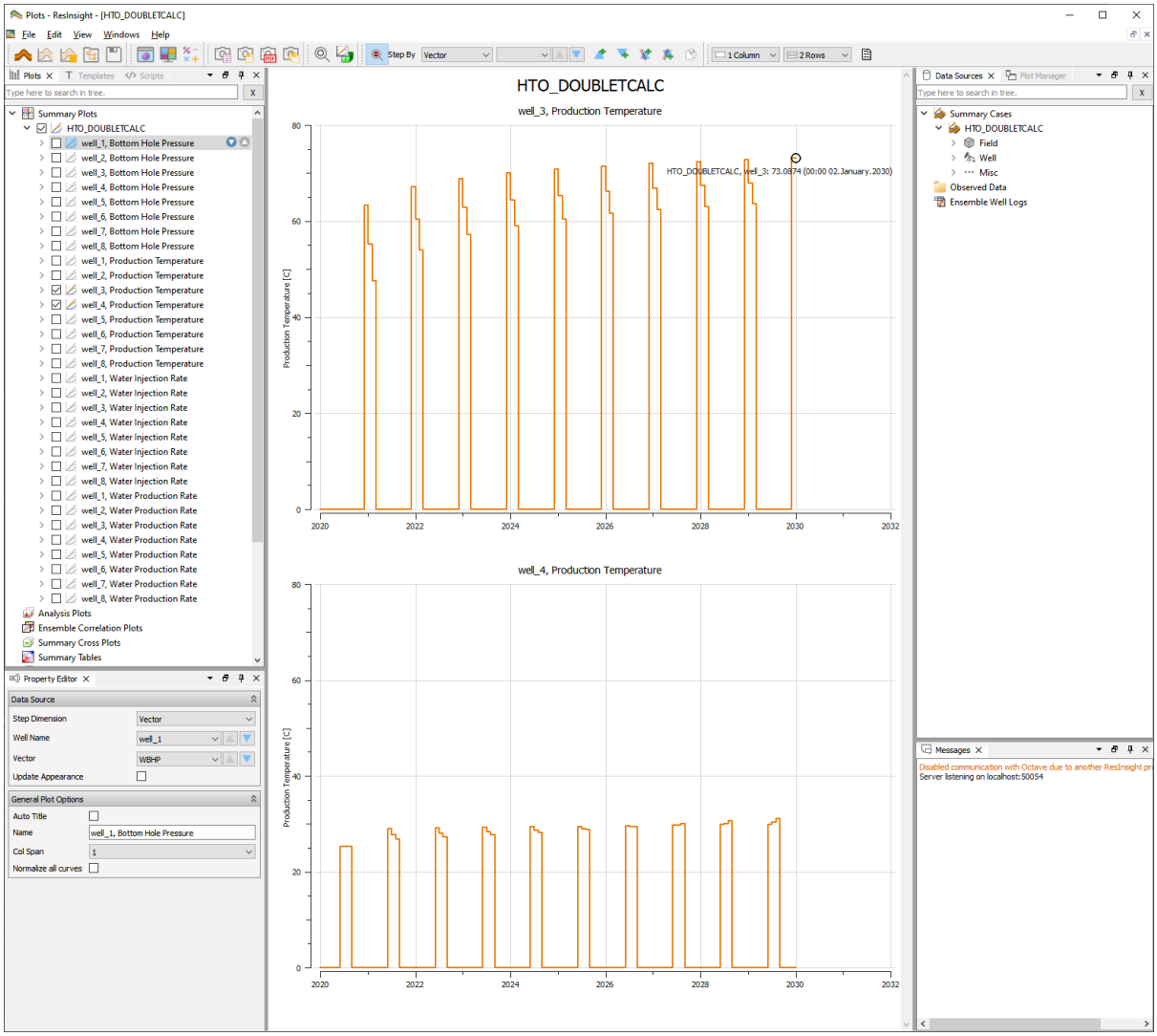


Figure 15: ResInsight 'Plots' for the HTO Demo model

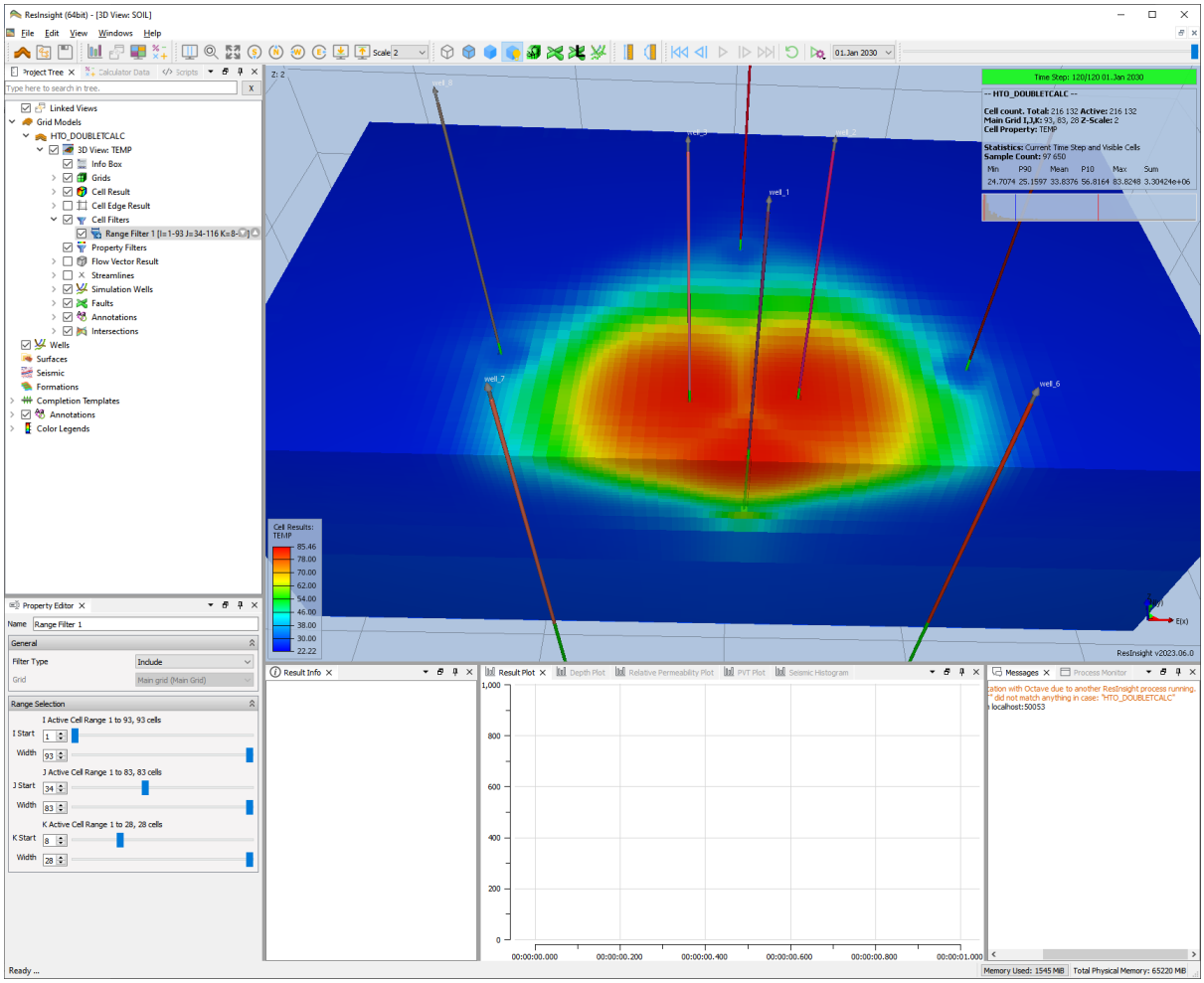


Figure 16: ResInsight '3D View' for the HTO Demo model

Demo Subsidence Post Processing

To illustrate how to perform the subsidence calculation for a non-standard project the workflow is demonstrated for the demo geothermal project. To this end in the postprocessing tab, set the 'post processing input .egrid file' and adapt the 'layer properties' the number of layers in the model:

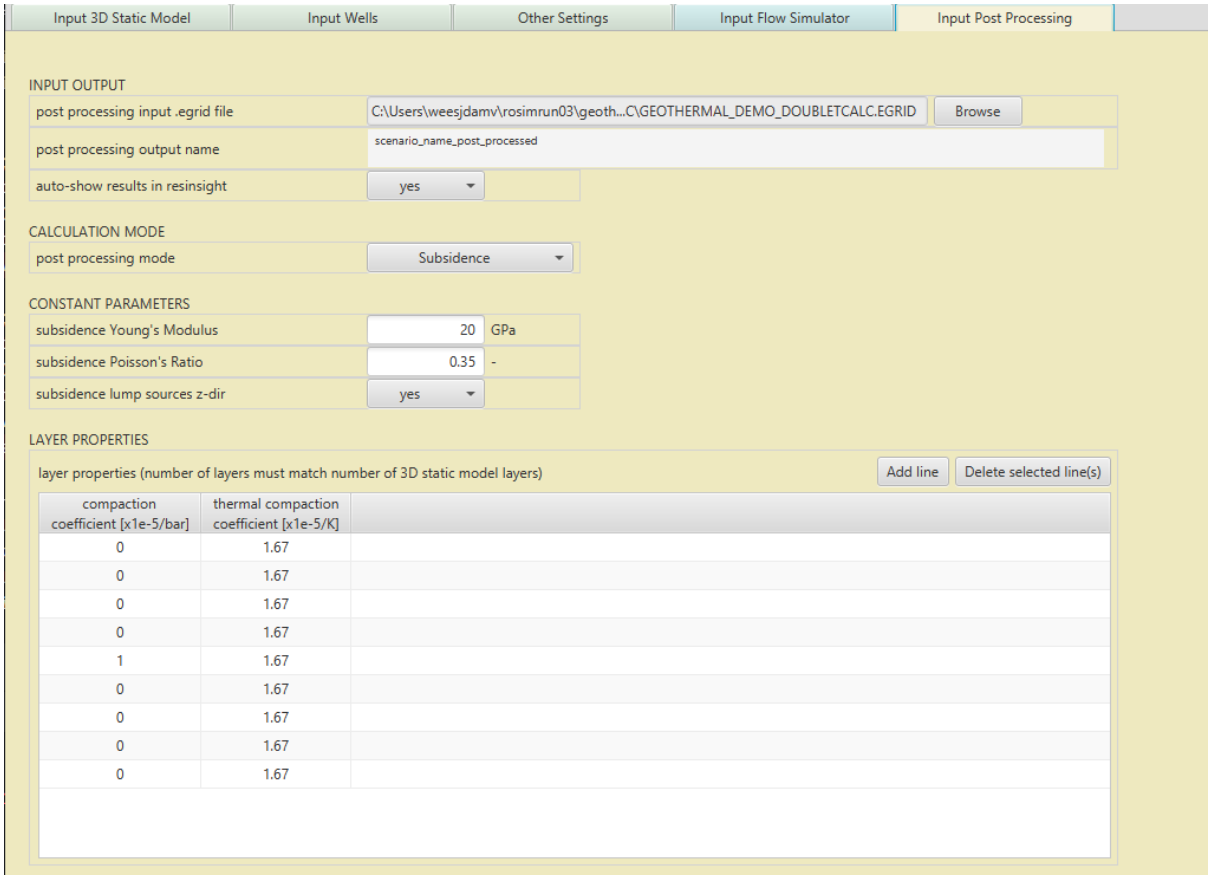


Figure 17: Input Post Processing Tab for the geothermal demo case with nine layers

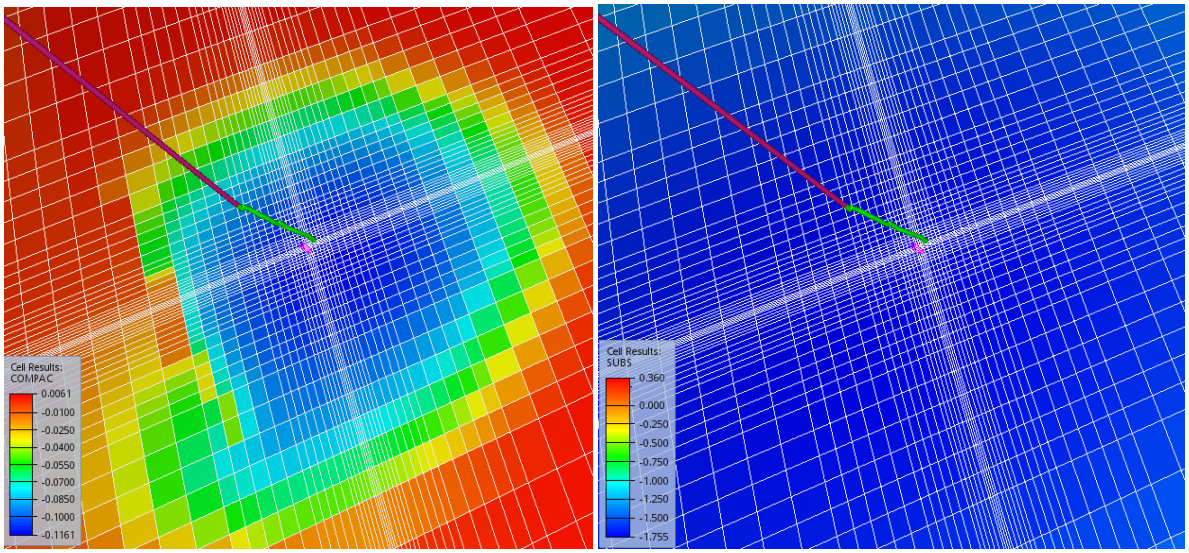


Figure 18: Compaction (left) and resulting subsidence [mm] (right) after 15 years.

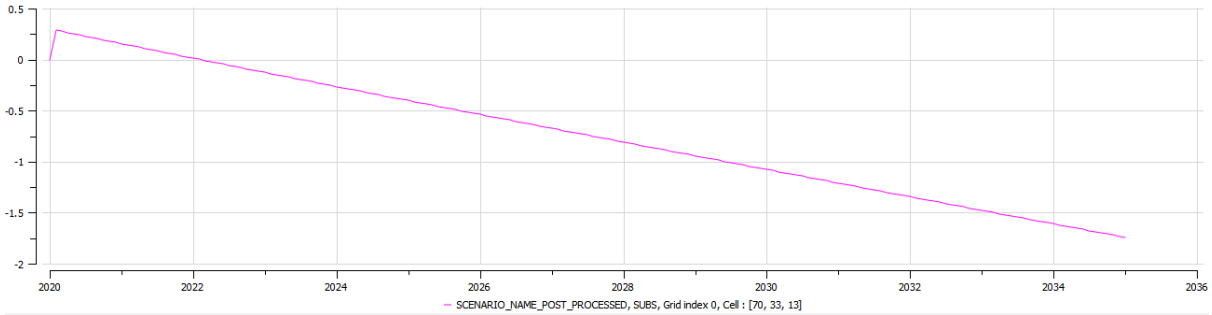


Figure 19: Compaction [mm] at surface close to injector indicated in Figure 18

Versions

1.2.0: March 2025

Added alternative options for the selection of the open section in the wells (MD, TVDSS, by layer), also allowing multiple open sections. Enabled setting the temperature boundary conditions in the stable version (previously only in the beta version).

1.1.1: November 2023

Fix well trajectory DX, DY input: these are cumulative now (deviation from well head), similar to for instance Petrel. Previously it was with respect to the previous deviation point.

1.1.0: October 2023

Added simple vertical fault capability and post-processing subsidence calculation. Use OPM version 2023.04 seems to work well. The OPM settings created by rosim are not yet optimized.

1.0.0: December 2022

Version used in the December 2022 benchmark comparing the results for DoubletCalc3D and Eclipse300. The 2022.10 OPM version was not usable with temperature dependent density and viscosity.

References

[1] The Open Porous Media Initiative, „OPM | The Open Porous Media Initiative," [Online]. Available: https://opm-project.org/.

[2] C. Solutions, „ResInsight," [Online]. Available: https://resinsight.org/.

[3] M. V. J. v. W. E. Peters, „ROSIM benchmark December 2022," 2022.

[4] Z. W. M. Batzle, „Seismic properties of pore fluids," *Geophysics,* vol. 57, nr. 11, pp. 1396-1408, 1992.

[5] H. E. K. R. J. C. J. Kestin, „Tables of the Dynamic and Knematic Viscosity of Aqueous NaCl Solutions in the Temperature Range 20-150 C and the Pressure Range 0.1-35 MPa," *J. Phys. Chem.,* vol. 10, nr. 1, pp. 71-87, 180.

[6] G. B. S. S. P. J. E. M. C. Elisabeth Peters, „Modelling of multi-lateral well geometries for geothermal applications," *Adv. Geosci.,* vol. 45, pp. 209-215, 2018.

[7] E. E. P. C. A. N. R. S. S. B. G. a. C. M. Peters, „Upscaling of RJD for incorporation in reservoir simulators," *The Horizon 2020 SURE Project: Deliverable 7.4,* 2019.

[8] M. V. H. V. W. J. Pluymaekers, „Doubletcalc 2D: A free geothermal flow simulator. European Geothermal Congress, EGC2016, Straatsburg. (https://pangea.stanford.edu/ERE/db/IGAstandard/record_detail.php?id=27526)," 2016.

[9] J. Droniou, „Finite volume schemes for diffusion equations: introduction to and review of modern methods," *<hal-00813613v1>,* 2013.

[10] G. a. Langevin, „User's Guide to SEAWAT (https://fl.water.usgs.gov/PDF_files/twri_6_A7_guo_langevin.pdf)," 2002.

[11] SLB, „ECLIPSE Industry Reference Reservoir Simulator," [Online]. Available: https://www.software.slb.com/products/eclipse.

[12] J. Verwer, „A class of stabilized three-step Runge-Kutta methods for the numerical integration of parabolic equations.," *Journal of Computational and Applied Mathematics,* 1977.

[13] e. a. J.D. Van Wees, „Numerical modelling of thermal convection in the Luttelgeest carbonate platform, the Netherlands.," *Geothermics,* vol. doi:10.1016/j.geothermics.2016.05.002.

[14] F. e. al., „Surface Movement Induced by a Geothermal Well Doublet (geothermal-energy.org).," in *WGC*, Melbourne, 2015.

[15] J. P. M. O. S. F. P. V. T.-V. K. O. B. W. B. H. D. C. T. Van Wees, „3D Mechanical Analysis of Complex Reservoirs: a novel mesh-free approach.," *Geophysical Journal International,* 2019.

[16] TNO, „DoubletCalc2D: Tools | NLOG," [Online]. Available: https://www.nlog.nl/en/tools.

[17] TNO, „DoubletCalc1D: Tools | NLOG," [Online]. Available: https://www.nlog.nl/en/tools.