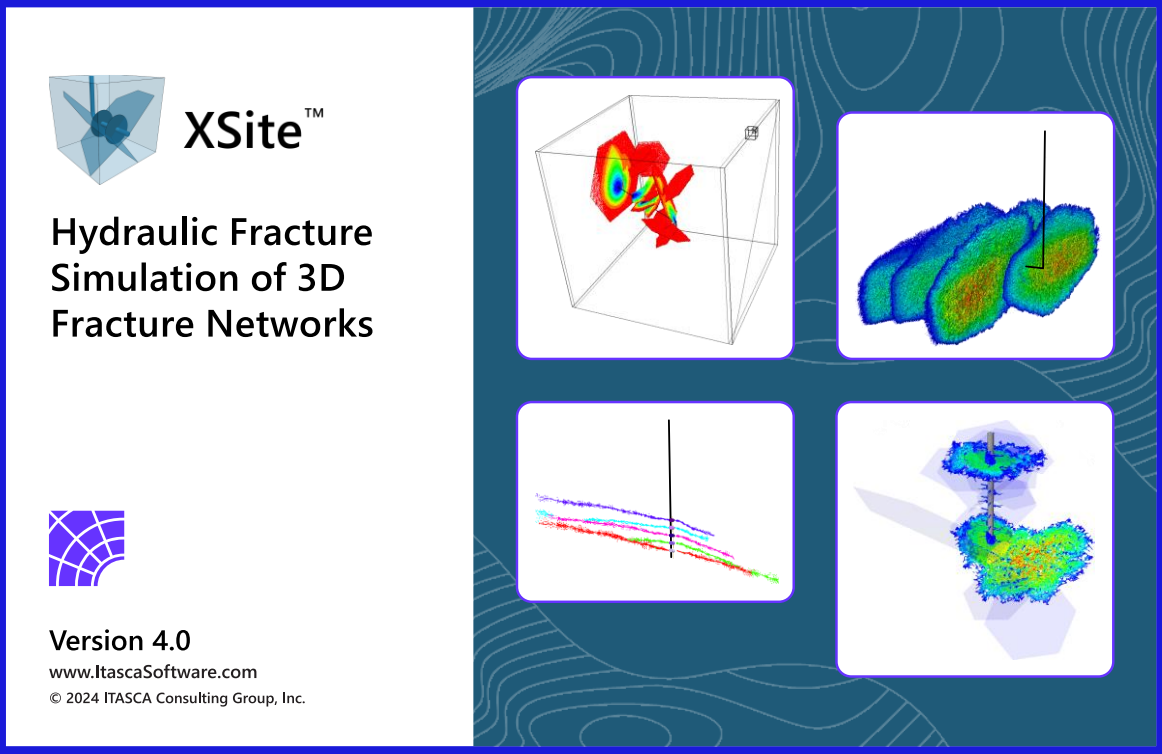
XSite

Version 4.0



User's Guide

Itasca Consulting Group, Inc.
111 Third Avenue South, Suite 450
Minneapolis, Minnesota 55401 USA

XSite

User's Guide and Tutorial

Version 4.0

Table of Contents

Introduction

XSite models deformation and hydraulic fracture (HF) propagation and related microseismicity in naturally fractured reservoirs in three dimensions (3D). There is no restriction on the geometry of new fracture surfaces and their interaction with existing fractures and joints. Cracks may form in any way, in contrast to the restrictive propagation rules imposed by conventional hydraulic simulators. Figure 1 highlights the main elements of an XSite model that consists of any number of segmented boreholes with clusters (i.e., fluid injection points) in a rock mass containing any number of joints, which can be planar, circular or polygonal, or have a general 3D geometry. The code allows simulation of multiple injection points in a heterogeneous reservoir with an initial non-uniform stress and pore-pressure states. Histories and contours plots of the model results of key variables, such as fracture fluid pressure and cracks, can be displayed and are interactive.

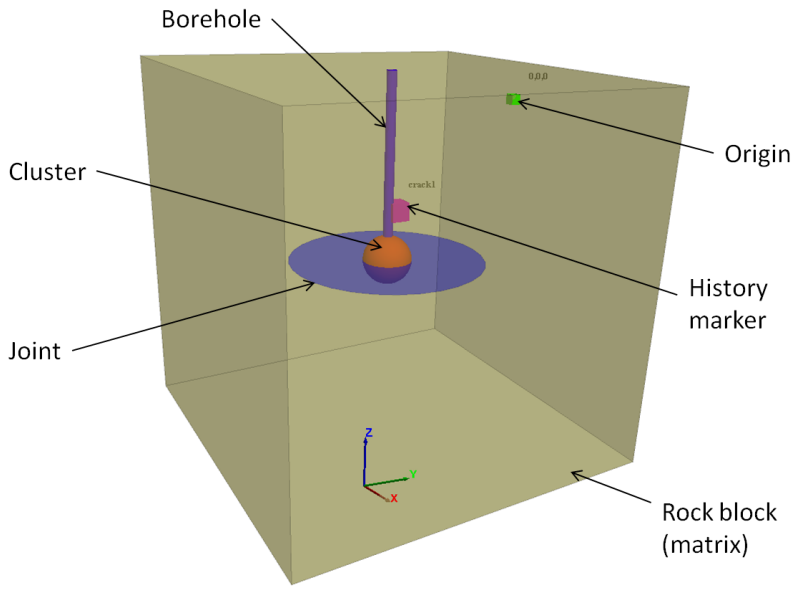


Figure 1 Sketch plot of an XSite model.

The code efficiently simulates the propagation of hydraulic fractures for field applications in unconventional reservoirs by implementing a version of the Synthetic Rock Mass (SRM) approach (Pierce et al., 2007) using a lattice scheme. The lattice consists of relatively uniformly but quasi-randomly (i.e., not in a regular pattern) distributed discrete masses (or lattice nodes) connected by springs to represent the rock matrix and joints. When the average spacing (i.e., the lattice resolution) is relatively small compared to the length scale of interest (i.e., model dimension), the lattice response is equivalent to that of a continuum.

The SRM approach simulates both the fracturing of intact rock (through spring breakage) as well as rock movement on existing discontinuities. The springs that obey the Smooth Joint Model (SJM) in SRM do

not act in the direction of the line connecting the corresponding nodes, but, instead, respect the normal to the joint that intersects that spring. Thus, the SJM allows simulation of sliding of a pre-existing joint in the model, unaffected by any artificial joint surface roughness resulting from lattice resolution or the random arrangement of lattice nodes. The springs between the nodes break when their strength (in tension) is exceeded. Breaking of the springs corresponds to the formation of micro-cracks, and microcracks may link to form macro-fractures. Kinetic energy released by spring breakage and joint slip is estimated, which allows the model to reproduce microseismic emissions (Hazzard et al., 2002). Joints, or an entire discrete fracture network (DFN), can be defined using the code's built-in Joint Set generator. Alternatively, DFN can be imported via data files in one of the following formats: *.CSV, *.XML (i.e., Extensible Markup Language), *.FAB (FRACMAN) or *.DFX (AutoCAD Drawing Interchange Format).

Non-steady fluid flow and pressure are implemented in both the pre-existing joints and stress-induced cracks by discretizing the flow model by (a) penny-shaped reservoirs associated with contacts between particles; and (b) flow pipes between adjacent reservoirs. The fracture flow is fully coupled (i.e., two-way interaction) with mechanical deformation and the pore pressures in the joints acting to load the solid model, whereas the deformation of the solid model causes pore-pressure and aperture (i.e., permeability) changes. Rock matrix flow, which is formulated differently than the fracture flow for computational efficiency, represents leak-off from fractures into intact rock. The leak-off can also be represented using the Carter leak-off formula. Fluid may be injected or withdrawn, or a shut-in condition may be modeled. A pressure or injection rate can be specified independently for each injection point. A new mechanical incompressible fluid (MIF) numerical hydro-mechanical coupling scheme (Itasca, 2023) has been implemented that allows faster simulation of coupled problems compared to other Itasca codes.

Using XSite

1.0 File Conventions

There are three types of files that are used or created by XSite. The files are distinguished by their extensions and are described as follows, where * indicates a user-provided filename.

Setup file (*.xml)

This file contains the complete simulation dataset in XML format (i.e., text) that defines the simulation setup data. At a minimum, the *.xml file is *required* to reproduce any model. This type of file is referred to as a Setup file.

Project file (*.prj)

This file contains any plot views and settings generated by the user. This graphical information is not associated with a specific model setup or model state. This type of file is referred to as a Project file.

Save file (*.sav)

This file contains all the state information (i.e., results) of the lattice model, provided that a simulation has been started and a model state has been saved. Multiple simulation states may be saved. This type of file is referred to as a Save file. Every Save file is accompanied by a Setup file created at the same time as the Save file. A Save file cannot be loaded in XSite without a Setup file.

The *.sav and *.prj files are generated automatically with the same name as the *.xml file when the Setup file is saved. As *.sav files are generally very large, when transferring a model to another user, usually only the *.xml, and optionally the *.prj, files are needed, as these files enable the model to be regenerated. The *.sav file may be deleted in order to restart the simulation from scratch (with possible changes to the model beforehand).

2.0 Graphical User Interface

The XSite Graphical User Interface (GUI) consists of three main panels, as shown in Figure 2. The left-hand simulation setup panel contains tabbed controls to define the model geometry domain, geology (e.g., rock matrix, joints, seams, and fluid), borehole layout and numerical solution parameters. The central plot view panel displays graphical information about the model and simulation results. Multiple tabbed plots can be generated. The right-hand plot control panel contains plot items and attribute settings to create and manipulate plots. The arrangement and display of panels may be changed by using the Windows and/or Layout menu bar items, by positioning the mouse cursor between panels and resizing them horizontally, or by positioning the mouse cursor on the panel title and dragging it to a new location. Shortcuts to various common operations are provided via two contextual toolbars (i.e., the toolbars will change depending on whether the mouse focus is on the simulation setup panel or the plot view panel).

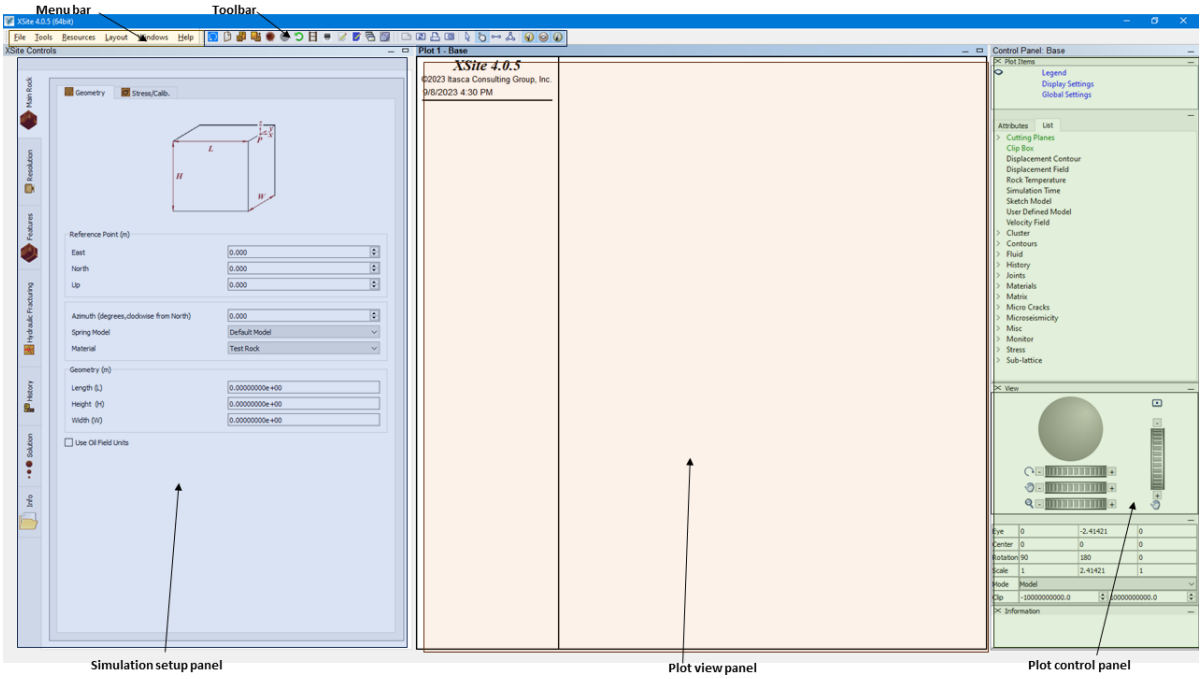


Figure 2 XSite GUI.

3.0 Main Menu Bar

The top menu bar consists of several menu items and icons. The menu items are reviewed as follows.

File

Load Dataset

The **Open File** dialog box will be displayed, enabling a saved dataset to be reloaded. You will be able to navigate to the desired folder where a previously created .xml file is located. If a *.prj file of the same main name exists, the saved plot states will be restored. The toolbar button  provides the same functionality.

Save Dataset

The current state of the model including the *.prj file will be saved — the file previously named in a **Load Dataset** or **Save Dataset As** commands will be overwritten. The toolbar button  provides the same functionality.

Save Dataset As

The **Save File** dialog box will be displayed, allowing to navigate to the desired folder where the model will be saved. The current state of the model including the *.prj file will be saved to a named file. Note the comments above regarding file types.

Load State

The **Open File** dialog box will be displayed, enabling a saved dataset to be reloaded. You will be able to navigate to the desired folder where a previously created .xml file is located. Even if a *.prj file with the same name as its corresponding .xml file exists, the saved plot states will not be restored, and the current plot items will be preserved.

Save State

The current state of the model, excluding the *.prj file, will be saved. The file previously named in the **Load Dataset** or **Save Dataset As** commands will be overwritten.

Save Results File

The current state of the model will be saved in a file that has a smaller size than a regular save file. This file can be used for post-processing but cannot be used for continuation of simulation. This feature is available in the MPI version of the code only.

Start Simulation

The toolbar **Start Simulation** button  provides the same functionality.

When this option is selected, the **Start Simulation** dialog is displayed (Figure 3). If this is the first time the command has been given (or the corresponding toolbar button has been clicked), then the lattice model is built. Depending on the size of the model, it may take some time to build the model. The different steps of the model initialization are shown in an information box located at the lower left corner of the main window. In the lower right corner of the main window, a status bar is displayed indicating the completed percentage of the current initialization step. The simulation time is determined by the time (in seconds) entered in the **Start Simulator** dialog box. If zero time is requested, only the model-building phase is performed; this is useful for checking the model setup.

There are six check boxes in this dialog: **Mechanical active**, **Elastic mode**, **Fluid active**, **Simplified toughness-dominated regime active**, **Approximate pressure gradient** and **Thermal active**.

Mechanical active

This field will activate the mechanical simulation (i.e., numerical solution of evolution of motion of nodes and forces in the springs).

Elastic mode active

If elastic mode is activated, springs will have an elastic behavior and will not break.

Prevent micro-cracking after elastic equilibrium

This option can be checked only if the **Elastic mode** option is selected. It should be used only during model initialization, either when a complex stress state is imported (from continuum model or previous XSite simulation) or when the stresses are initialized using stress initialization commands built into XSite. The option prevents spurious cracking in the initial stress state due to inherent force dispersion in the lattice.

Full flow active

Fluid Active can only be checked if the Activate Fluid Flow checkbox in the **Solution/Fluid** tab is also checked. This option will be disabled otherwise. **Full Flow** computation is active if the fluid active checkbox is selected. This option solves fluid diffusion in the entire pipe network. See the section describing the Fluid time step factor on Solution/Fluid tab for further explanation on how the explicit or implicit fluid computation is activated.

If the radio-button **Simplified toughness-dominated regime** is selected, the hydraulic fracture propagation is solved, assuming that there is no pressure drop between the injection point and any of

the connected fluid nodes. If **Approximate pressure gradient** is active, an outward incremental pressure loss is applied radially from the injection point to the connected fluid elements. See the section describing the simplified toughness-dominated regime options on Solution/Fluid tab for further explanation.

Thermal Active

The check box Activate Thermal in the **Solution/Thermal** tab must be checked for the thermal simulation parameters to be enabled. This option will be disabled otherwise. A tightly coupled thermo-hydro-mechanical simulation is conducted if all three boxes (**Mechanical active**, **Fluid flow active** and **Thermal active**) are checked. The code automatically conducts sub-stepping, keeping all models synchronized but executing each at its own critical time step. In cases with processes driven by heat transfer, and particularly by heat conduction when the mechanical model evolves quasi statically and the flow model evolves through a sequence of steady states, tight coupling would lead to unnecessary computational overhead. In those cases, the user may specify intervals for equilibrating the model mechanically (due to induced thermal strains) and bringing the fluid flow to steady state (for new fracture apertures). The field **Equilibrium frequency** is specified in days, and it should be a fraction of the total thermal simulation time. If the value specified is zero, this step is not performed. When the simulation time reaches the **Equilibrium frequency** value, the code will cycle mechanically for the number of seconds specified in the field **Mechanical equilibrium time** (time estimated to be sufficient to reach mechanical equilibrium) followed by fluid simulation for the number of seconds specified in the field **Time to fluid steady-state** (time estimated to be sufficient to reach fluid flow steady state).

Mechanical, fluid or thermal simulations, or any combination of these, may be done if the corresponding boxes are checked. If the mechanical and fluid boxes are checked, then a fully coupled hydro-mechanical simulation is performed once OK is clicked. Background resolution length, lattice grid edge length, rotation scheme, aperture status and the maximum aperture setting are also displayed.

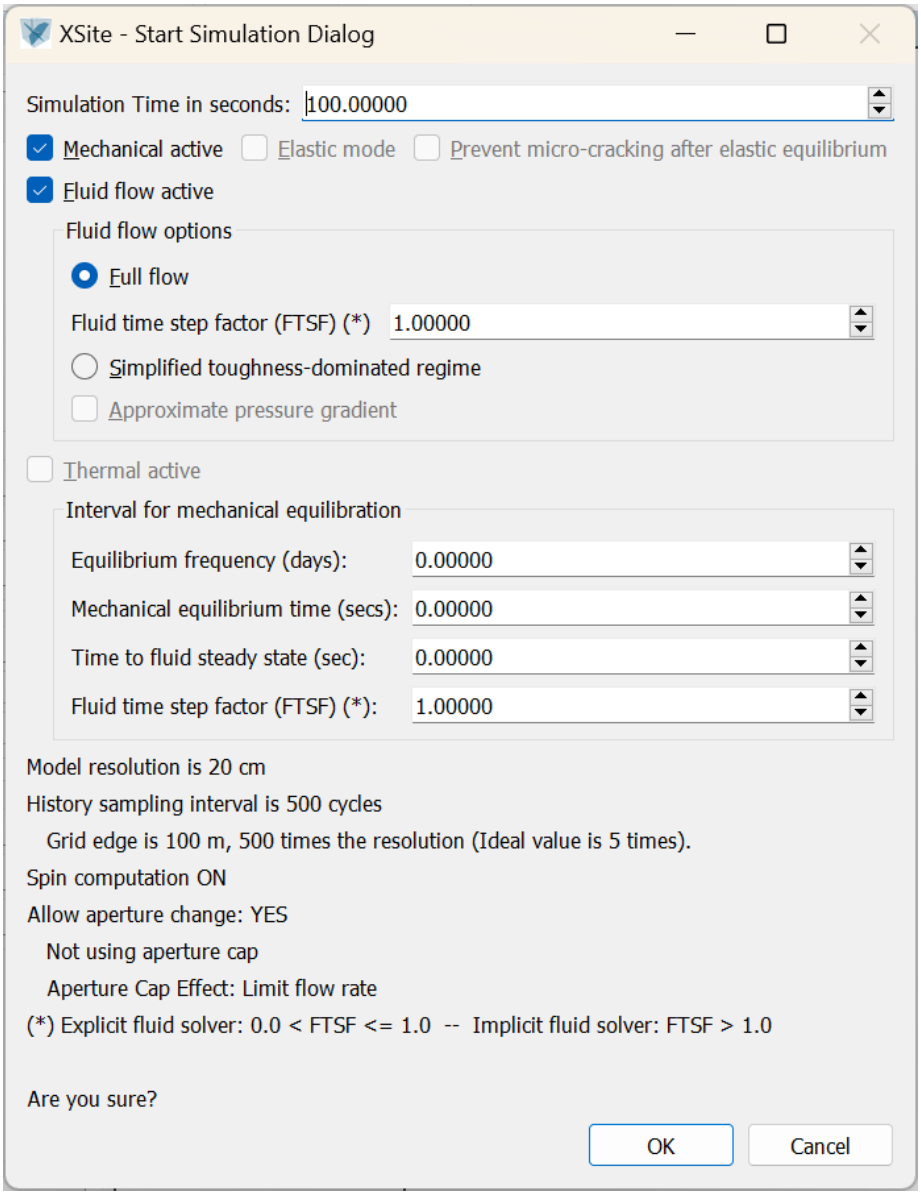


Figure 3 Start Simulation Dialog.

During simulation, a progress bar is displayed on the lower right corner of the main window, and the simulation may be canceled by clicking the Stop Simulation button on the toolbar. Note that this button is only active when a simulation is running. All active plot items are updated periodically during the simulation (see Tools/Options section).

Export Menu Option

Unless noted, the data will be exported to a .CSV (Comma-Separated Value) type of file, which may be imported into programs such as Excel to be analyzed. The first line of the file contains the column name of the exported fields with their respective units.

Export Clusters

This option exports the following information of a cluster to a .CSV file:

- Stage name
- Cluster active flag
- Location
- Pressure
- Radius
- Number of internal fluid nodes and springs
- Average fluid node aperture
- Propped volume
- Volume

Export Displacement History

If displacement histories were specified in the history tab, they may be exported to a .CSV file. This file will contain the displacement of a node over time. The node closest to the specified coordinate is selected to be traced.

Export fluid nodes (pre-existing joints and microcracks)

When this menu option is selected, the dialog box in Figure 4 will be displayed.

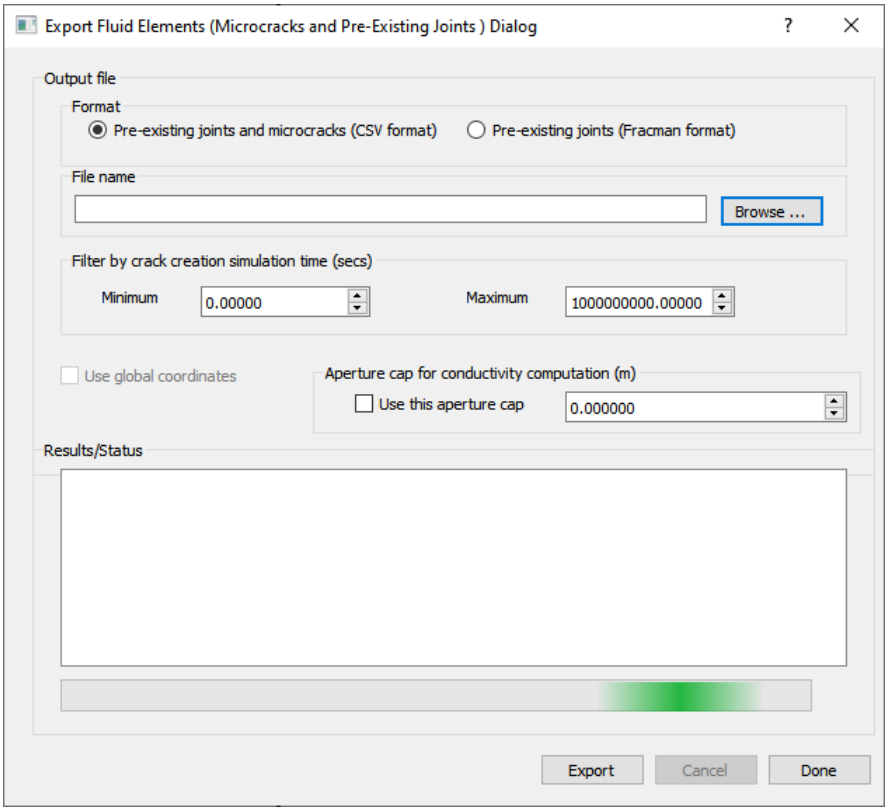


Figure 4 Export Fluid Elements Dialog.

Data can be exported in .CSV or FRACMAN formats. If the user selects the .CSV option, information on fluid nodes (microcracks and pre-existing joints) will be exported.

If the user selects the .CSV format, the PEJ-MC column will indicate the fluid node is at the location of a microcrack (MC) or a pre-existing joint (PEJ). In case of pre-existing joints, the data in this column will also indicate state of the joint: open or slipped. The data will also contain the location of the fluid node, the name of the cluster to which the fluid node is connected, shear displacement in case of joints, pressure, temperature, proppant concentration, aperture, conductivity and the name of the layer in which the fluid node is located.

The user can select the fluid nodes that will be exported based on the range of time when the fluid node was created. For example, to output the microcracks created for a given stage, enter the initial and final time in which the stage was stimulated. An aperture cap will be used to calculate fracture conductivity if this option is activated. (The purpose of this option is to prevent loose nodes created by intersections of multiple joints, resulting in unreasonably large permeabilities.)

If the user selects the FRACMAN format, only the pre-existing joints will be exported. Users may opt to export the data in the global coordinate system.

Events or errors that might occur during generation of the export data will be printed in the **Results/Status** window. When the program finishes exporting the data, the total number of exported pre-existing joints will be printed.

Export Histories

All history data defined in the history tab may be exported. The first column of the file contains the time that the history was recorded in seconds. The other columns are displayed in the order they were created in the history tab. A column with a name that identifies the history and type of history is displayed in the first line of the file.

Export Joint and Microcracks Normal Displacement

This option exports (as *.CSV) the fluid pressure, normal displacement, aperture and location of the joints and microcracks.

Export Matrix

The dialog in Figure 5 will be displayed when this option is selected.

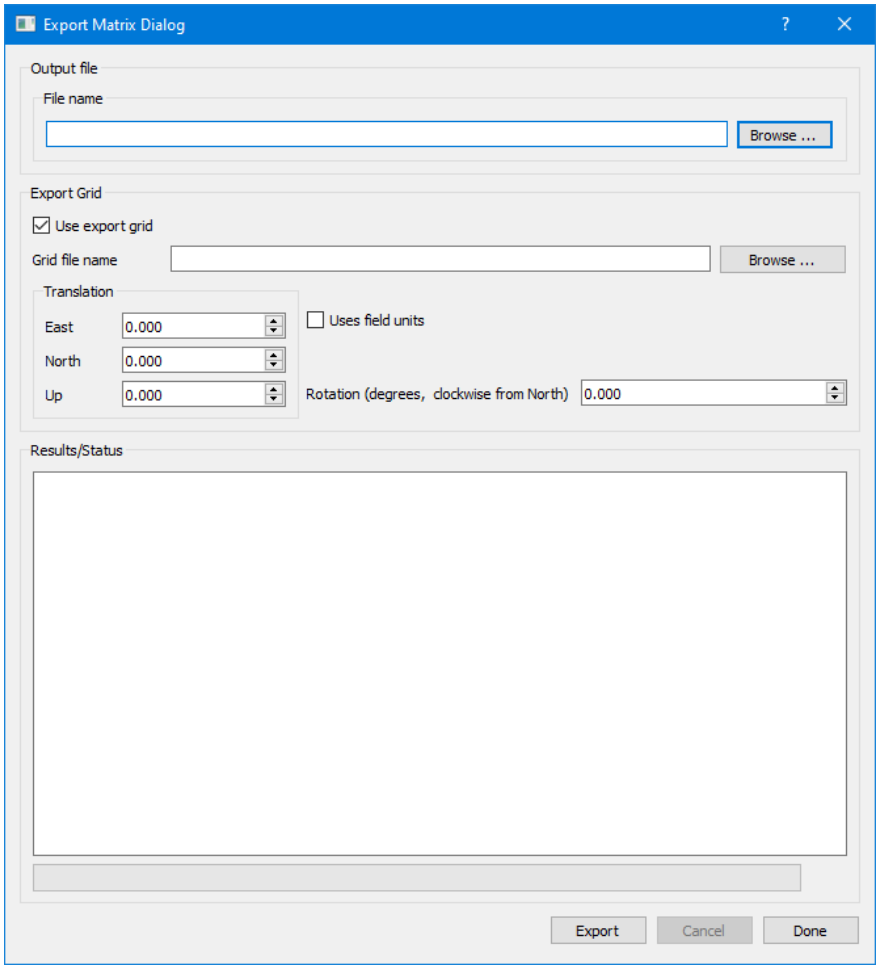


Figure 5 Export Matrix Dialog.

If the user chooses to export matrix data using an existing grid file, the grid file name must be specified. The grid file is explained in the File/Create Export Grid section. In this case, matrix flow rates from the zones that belong to the internal XSite three-dimensional grid (see Lattice grid edge) will be upscaled and mapped into the given grid zone. Make sure to select **Uses field units** if the export grid file is in global coordinates.

The location, X,Y,Z, refers to the centroid of the spring. The location can be translated and rotated if the user populates these values in the dialog box.

If the **Use export grid** option is not selected, location, pressure, temperature and flow rate will be exported directly from each spring in the lattice.

The events or errors that occur during the generation of the export data will be printed in the **Results/Status** window. When the program finishes exporting the data, the total number of exported springs will be displayed. When exporting data using a grid file, the number of zones successfully and unsuccessfully mapped from the lattice into the grid file will be displayed in the **Results/Status** window.

Export Microcracks

When the **Export Microcracks** menu option is selected, the dialog in Figure 6 will be displayed.

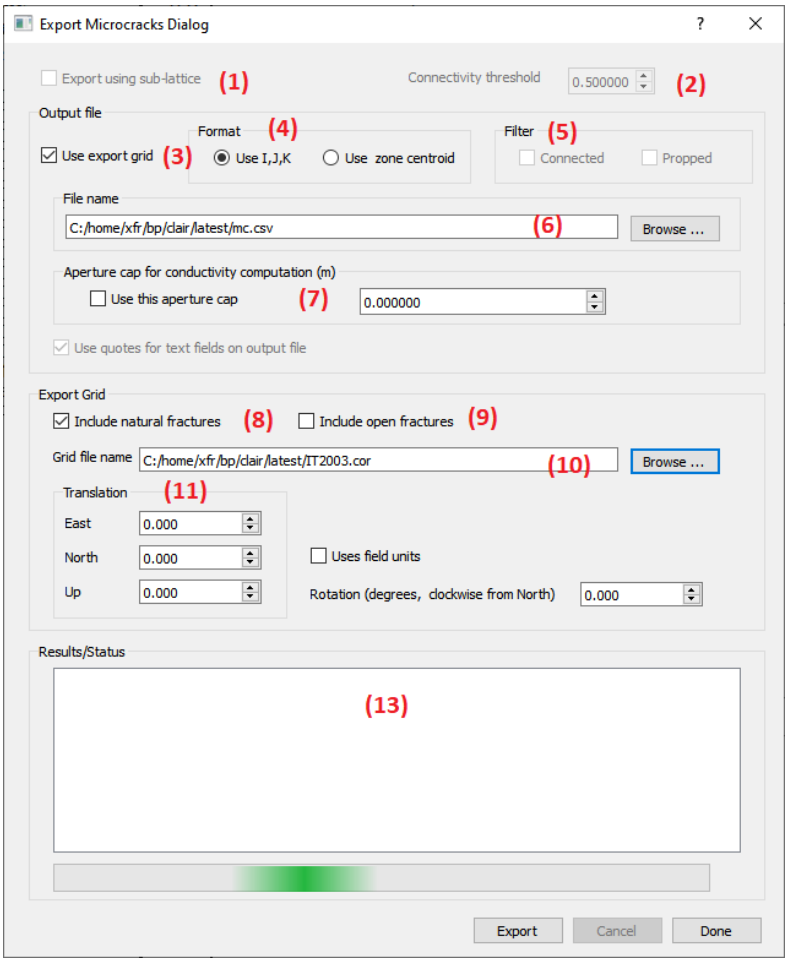


Figure 6 Export Microcracks Dialog.

Data is exported in comma delimited (CSV) format and the file name is specified on the edit box (denoted as (6) in Figure 6).

Sub-lattice data may be used to create the export data by selecting the checkbox (1). This checkbox is disabled if the sub-lattice mode was not used in the simulation. Connectivity threshold (default 0.5) is used to define sub-lattice microcracks that are connected to the injection cluster.

Microcrack data (e.g., location, radius, dip and dip direction, aperture, temperature, conductivity, name of the connected cluster, layer where the spring is located, etc.) are exported directly from the lattice. If the **Use export grid** checkbox (3) is selected, data are exported as computed upscaled quantities such as porosity and permeability.

The data exported directly from the model, which do not use a grid file, are: microcrack coordinates, the layer where the microcrack is located, the dip and dip direction, radius, counter that can be incremented by a user at different stages of the simulation, pressure, temperature, proppant concentration, aperture, capped aperture, conductivity, time of the crack creation and name of the cluster the crack is connected to.

You may filter the data selecting only connected and/or propped microcracks by checking the boxes (5).

Capped aperture (7) is used only in the calculation of the flow rates during the full flow simulations. It does not limit the apertures. Therefore, apertures in the model can exceed the aperture cap. The capped aperture is computed depending on how the model was setup on the Solution/Fluid/Aperture tab. If checked here, the capped aperture is used in upscaling the permeability (i.e., the apertures used in upscaling the permeability/conductivity cannot exceed the capped aperture).

If the user selects to export microcracks using a grid file, XSite uses two grids, one of which is internal and automatically generated by XSite (see Mechanical/Lattice grid edge), and the external grid provided by the user. The file grid must be specified in the field (10). See File/Create Export Grid for an explanation of the grid file. In this case, average microcrack data from the zones that belong to the internal XSite three-dimensional grid will be upscaled and mapped into the given grid and exported.

To export computed data using a grid file, select the checkbox (3). The grid is available in CSV format and there are two possible layouts for the data that can be selected by checking radio buttons (4) **Use IJK** or **Use Zone Centroid**.

The file layout for **Use IJK** has the following fields: I, J and K indexes of the grid, porosity, all conductivity tensor components, pressure, saturation, if the fluid node is located in a natural fracture and the label created during the model setup that identifies the fracture.

The file layout for the **Use Zone Centroid** has the following fields: *x*-, *y*- and *z*-coordinates of zone centroid, conductivity tensor diagonal components and porosity.

XSite computes the upscaled continuum fields using the same procedure used in *PFC3D* as described in the Measured Quantities section of the *PFC3D* documentation. However, instead of using spheres, XSite uses a regular grid.

If the user selects checkbox (8), data from natural fractures will be used to compute the export values. By default, XSite uses the conductivity curves defined in the **Resources** menu to calculate the values. Checkbox (9), **Include open fractures** must be selected to include the fractures that are not propped. In this case, the fracture permeability will be calculated using the formula:

$$\kappa = \frac{a^2}{12}$$

where a is the fracture aperture.

The fields in the group box (11) labeled **Translation** will apply translation and rotation to the gridpoints. In case the grid file uses field units, the gridpoints will be converted to coordinates in the metric system.

The events or errors that may occur during generation of the export data will be printed in the **Results/status** window (13). When the program finishes exporting the data, the total number of exported microcracks will be displayed. When exporting data to an external grid, the number of zones successfully and unsuccessfully mapped from the XSite grid into the external grid will be displayed in this window.

Export FRACMAN

The **Export FRACMAN** option exports fluid nodes to a file in FRACMAN format. The dialog in Figure 7 will be displayed when this item is selected.

Sub-lattice data may be used to create the export data by selecting the checkbox. This checkbox is disabled if the sub-lattice mode was not used in the simulation.

The user may filter the data selecting only connected and/or propped microcracks by checking the proper boxes.

Capped aperture is used only in the calculation of the flow rates during the full flow simulations. It does not limit the apertures. Therefore, apertures in the model can exceed the aperture cap. The capped aperture is computed depending on how the model was set up on the Solution/Fluid/Aperture tab. If checked here, the capped aperture is used in upscaling the permeability (i.e., the apertures used in upscaling the permeability/conductivity cannot exceed the capped aperture).

Fluid node locations and spring radii are used to create polygons and are formatted as fractures in the FRACMAN file. Natural fractures may be included if the corresponding checkbox is selected.

The **Results/Status** is updated during the export process and the total number of springs is displayed at the end of the run.

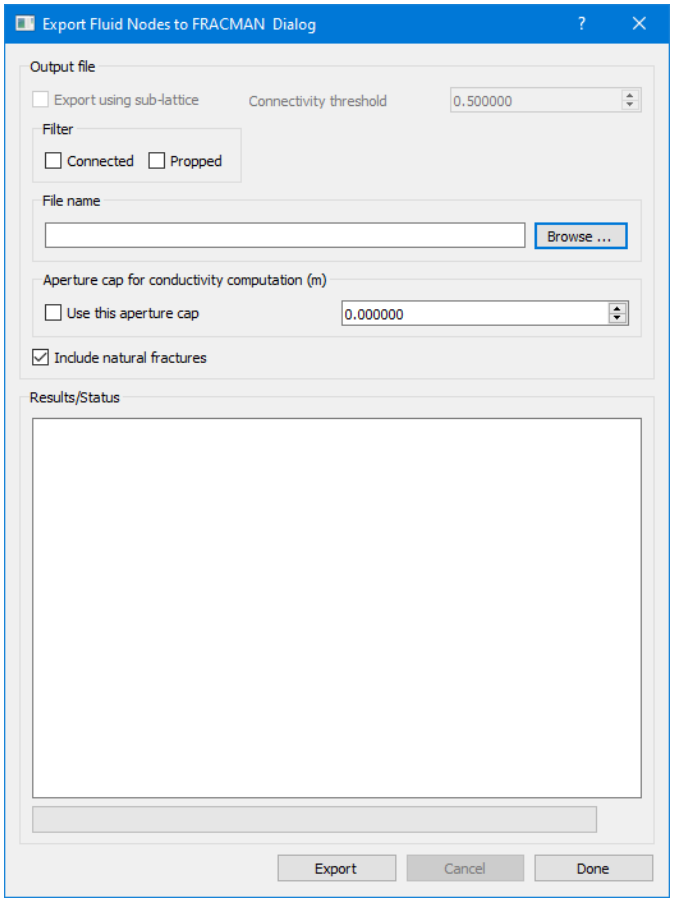


Figure 7 Export Fluid Nodes to FRACMAN Dialog.

Export Microseismicity

This feature exports (as *.csv) the location, type (slipped joint or broken spring), moment, energy and magnitude of the microseismic events generated during the simulation.

Export Nodes

This feature exports (as *.csv) the location, displacement and velocity of the lattice nodes in the model.

Export Pipes

This feature exports (as *.csv) the location, flow rate and aperture of the pipes in the model.

Export Sub-lattice Nodes

This feature exports (as *.csv) the location, displacement and velocity of the sub-lattice nodes in the model.

Export Sub-lattice Springs

This feature exports (as *.csv) the zone index, location, force, pressure, broken flag, joint flag and ID (valid for MPI version only) of all sub-lattice springs in the model.

Create Export Grid

The export grid is created using the **File/Create Export Grid…** menu option. This option will invoke the dialog shown in Figure 8. The edge length of the zones and the boundaries of the grid are specified by the user. The default boundary values are derived from the model extent, but the grid may be created within a certain region of the model by specifying a region extent within the model.

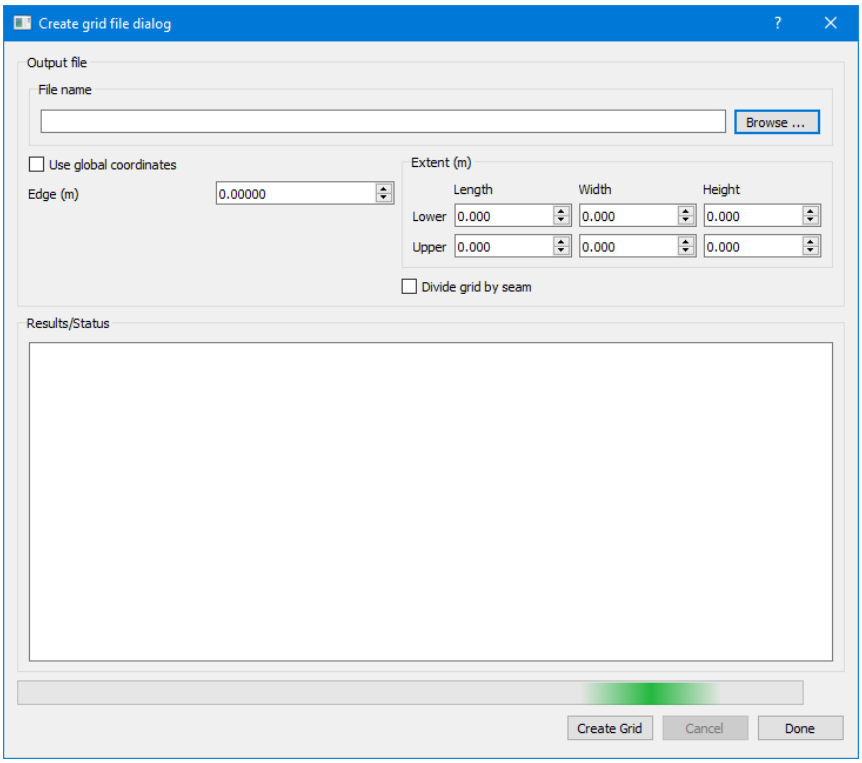


Figure 8 Create Grid File Dialog.

By selecting the **Divide grid by seam** checkbox, XSite uses the infinite planar seams defined in the model to adjust zone edges to fit zones within seams. Figure 9 shows a grid created by XSite. In this example, the grid edges are adjusted to accommodate a seam dipping at an angle of 13°. This option has the following limitations:

- The current implementation takes into consideration only infinite planar seams. A seam must cross the vertical boundaries of the model.
- Seams generated from DXF files are not considered.

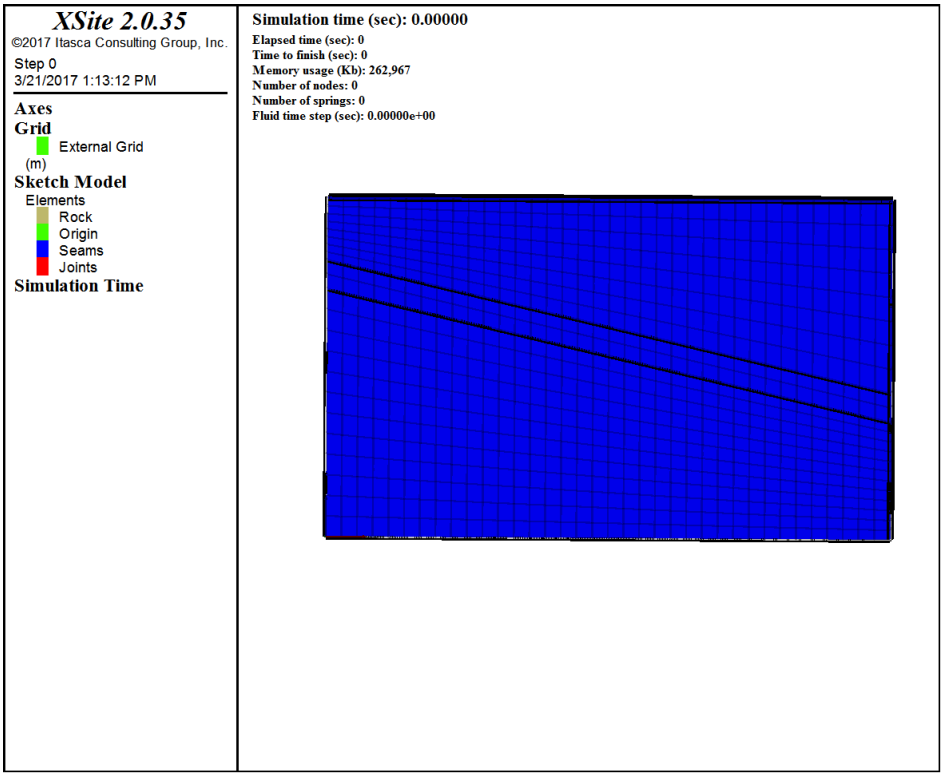


Figure 9 Model with a seam dipping at a 13° angle.

The grid file format is described as the following:

- Lines that start with the character "C" are comment lines.
- Each set of 24 numbers represents one cell. (The numbers are the *x*-, *y*- and *z*-coordinates of each corner of the cell.)
- *x* increases from west to east, *y* increases from south to north, and *z* increases downwards.
- The blocks are numbered I, J, K.

The grid file can be visualized using the **Monitor/Grid** plot item. The following attributes of this plot item shown in Figure 10 must be set by:

- Pressing the cross symbol denoted by (1) a **File browse** dialog will be displayed, allowing navigation to the directory where the grid file was created and selection of the desired file.
- Select **External Grid** denoted by (2).

The plot item will automatically adjust if global coordinates were used to create the grid.

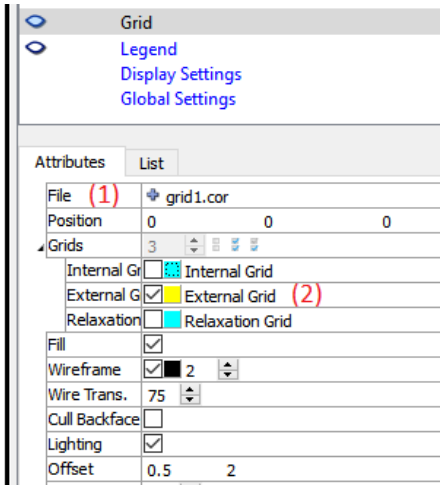


Figure 10 Attributes for "Grid" plot item.

Import data

Import DFN file in XML format

A Discrete Fracture Network (*.xml) may be imported. This allows fractures to be generated independently of the built-in joint generator. XSite supports two different file formats: JointStats and Itasca formats. Examples of these files and their schema definition can be found on the installation directory: "\Program Files\Itasca\ XSite400 \examples\DFN"[1]. Also, the format is described in the Joint/XML format section.

Import DFN file in CSV format

This feature imports a set of joints from a text file. The format of this file is described in: "\Program Files\Itasca\ XSite400 \examples\DFN\jointset_sample .dat"[1]. Also, the format is detailed in the Joint/Itasca FISHLab format section.

Import DFN file in FRACMAN format

This feature allows importing a set of joints from a file in FRACMAN (.FAB). The dialog in Figure 11 will be displayed when this menu option is selected.

[1] Assumes that the default installation directory is \Program Files\Itasca\XSite400

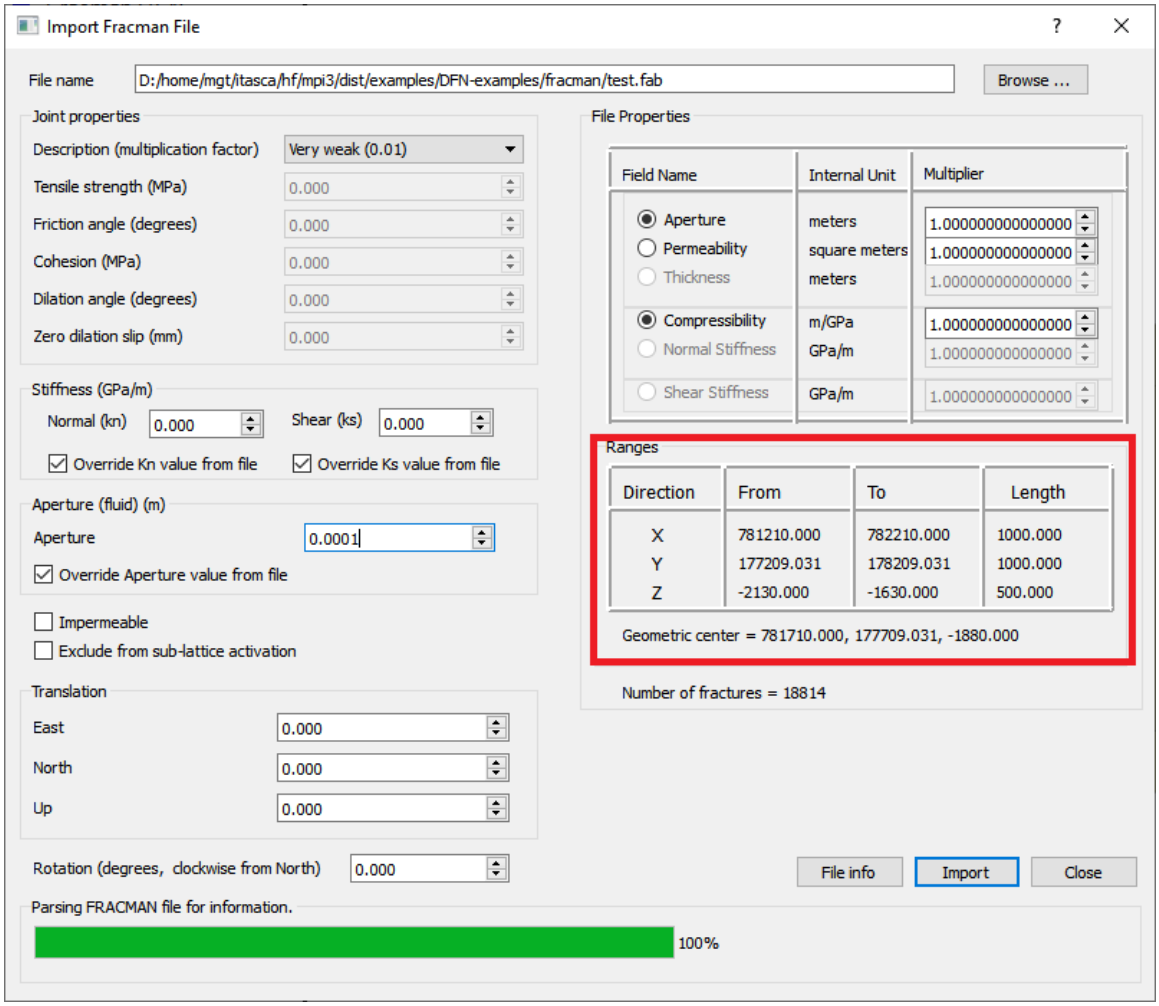


Figure 11 Import FRACMAN File Dialog.

After entering or browsing the FRACMAN file name, press the **File Info** button. The **File Properties** group box will be populated with information extracted from the file.

The geometric information displayed in the **Ranges** box can be used if the data in the FRACMAN file is given in global coordinates and needs translating to XSite's local coordinate system. This can be done either by populating the values of the vector in the **Translation** box in this dialog or populating the **Reference Point** vector in the Main Rock/Geometry tab. The FRACMAN example provided in file "C:\Program Files\Itasca\XSite400\examples\DFN\fracman\test.fab" [2] is a model with dimensions 1,000 m × 1,000 m × 500 m. The global coordinates of the reference point are (782,210 m; 177,709 m; -1630 m). The values for the East (*x*) and Up (*z*) points were obtained from the **To** column of the **Ranges** table. The value for the North (*y*) coordinate is the value of the **To** column minus half of the length of

[2] Assumes that the default installation directory is \Program Files\Itasca\ XSite400

the *y*-direction. Figure 12 shows the example file imported into the model, highlighting the coordinates of the reference point and the model dimensions.

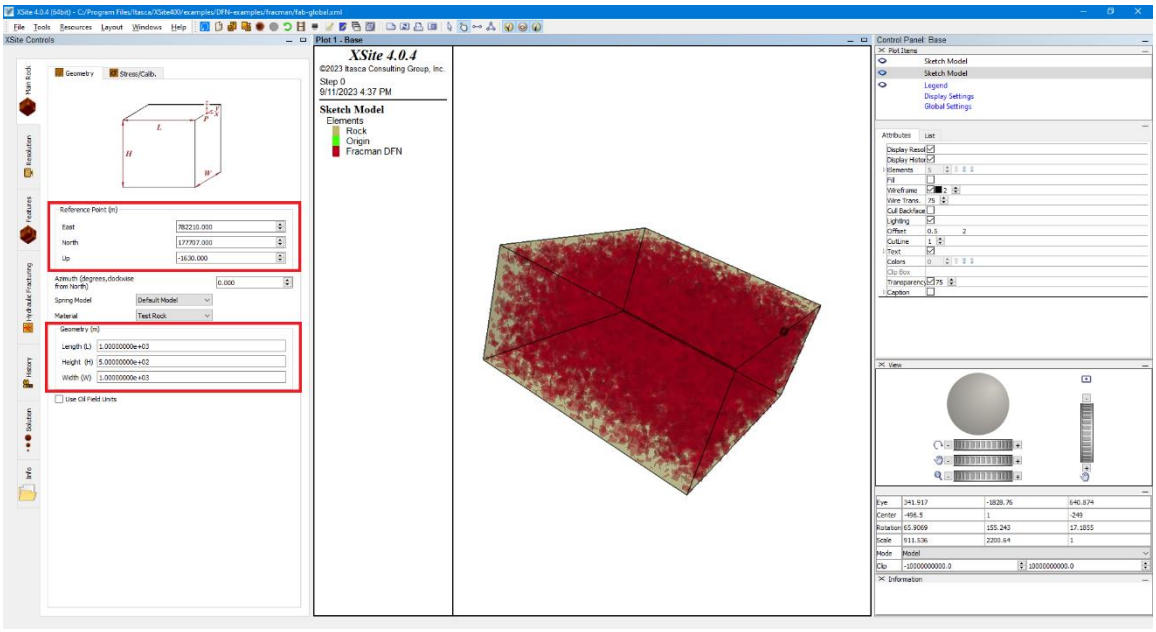


Figure 12 Plot of DFN Imported from FRACMAN.

The multipliers may be used to convert the units in the file into the XSite model internal units. The number of fractures is also displayed.

The joint properties may be specified, and they are discussed in the Features/Joint Sets section of this manual. Stiffness and the aperture found in the file may be overwritten. After entering the desired information, press the **Import** button. A dialog box will display the number of imported fractures. The imported FRACMAN DFN will be added to the Joints list in the **Features/Joint** tab, and it can be visualized in the Sketch Model plot item.

Import DFN and Seams from a DXF file

To include geology with complex geometry in XSite, the first step is to generate the 3D solid model of the geology (seams or geological layers) and the joints in the CAD application of your preference (e.g., Rhino, AutoCAD).

The 3D model must satisfy the following conditions.

1. Seams and joints must be created as meshes in the CAD application.
2. Seams must be defined as closed volumes and joints must be defined as surfaces.
3. A seam may be defined as an open mesh as long as it defines a closed volume.
4. Each seam and/or joint that will be imported into XSite must belong to a different DXF layer.

5. A set of seams or a set of joints (DFN) that share the same properties (e.g., seams that have the same rock type and stresses or the joints that have the same hydro-mechanical properties) may be mapped into the same layer.

Figure 13 shows a 3D solid model in a CAD application with four meshes, five closed volumes and two surfaces. Each mesh was mapped into a separate DXF layer. The highlighted seam (top seam, yellow) was assigned to a layer named Seam0. The **Properties** tab shows that the solid type is a "closed mesh."

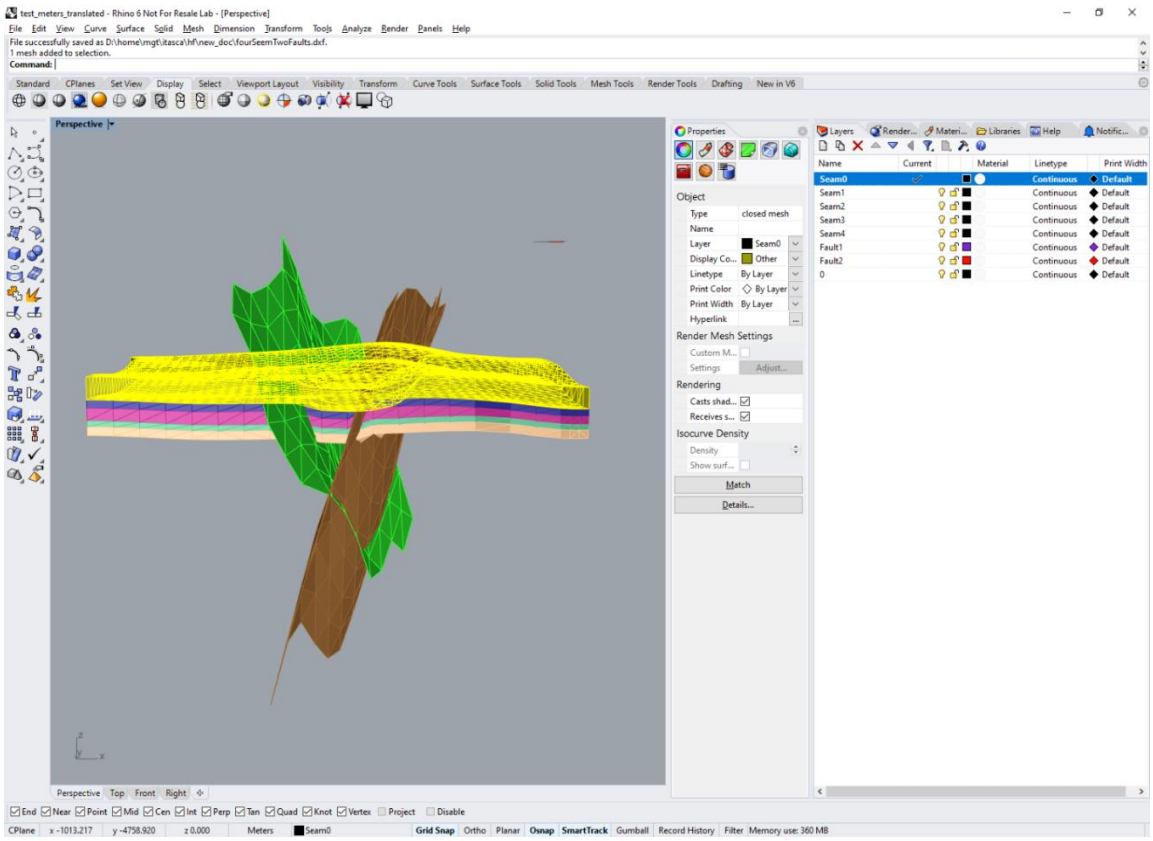


Figure 13 Example of 3D solid model.

The material(s) types to be applied to the seams must be defined in the XSite Material database.

The dialog box to import joints and seams is shown in Figure 14. The following steps are required during importing.

1. Enter the file name (1) or use the **Browse** button (2) to select the file.
2. Press the **Read** button (5). Columns **Layer Name** (3) and **Type** (4) will be populated.
3. In table **DXF Layers**, the column **Layer Name** will display the layers defined in the DXF file and the default value for type is **None**. All layers with type set to **None** will not be imported into XSite.

4. To assign type (i.e., seams and joint sets) in XSite to the CAD layers defined in the DXF file, double-click one of the rows of the table **DXF Layers**. The dialog shown on Figure 15 will be displayed.
5. Repeat steps above for all layers to be imported and leave the layer **Type** set to **None** for the features that you do not want to import.
6. You may want to translate and rotate the DXF data before importing or check **Uses global coordinate system**. In this case, the values of the reference point field defined in the Main Rock/Geometry tab are used to transform the points of the DXF file.
7. Press the **Import** (6) button and the features will be added to the model. The properties of the joints or seams will be added to the lists of joints or seams and may be later modified by double-clicking the desired item in these lists.
8. Use the Sketch Model plot item to verify the data were properly imported. You may also check the model by initializing it (i.e., executing it for zero time) using a coarse resolution and plot Materials/Uniform and Joints/Joint Traces.

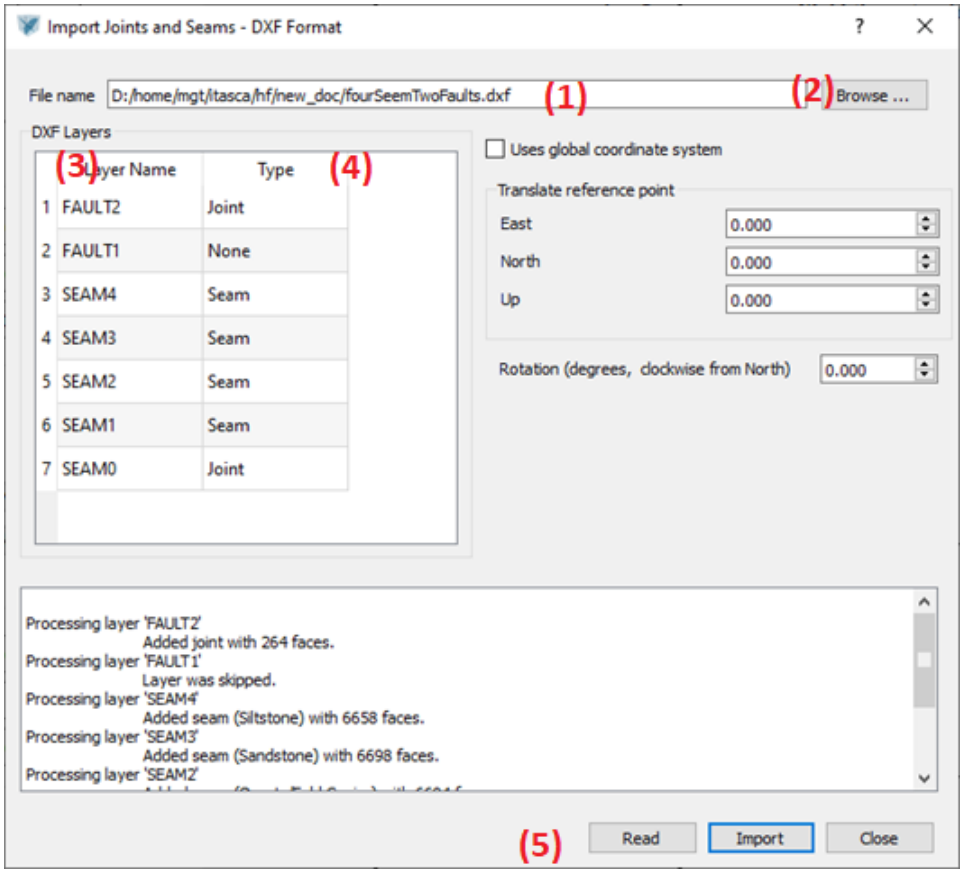


Figure 14 Import Joints and Seams – DXF Format Dialog.

In the dialog in Figure 15, the user can assign a DXF layer to either a fault (joint set) or a seam. The properties for joints or seams will be enabled depending on the selection of the **Layer Type** radio

buttons. For example, if the radio button **Joint** is selected, the joint aperture can be defined; if the radio button **Seam** is selected, the seam material, in-situ stress, etc., may be defined.

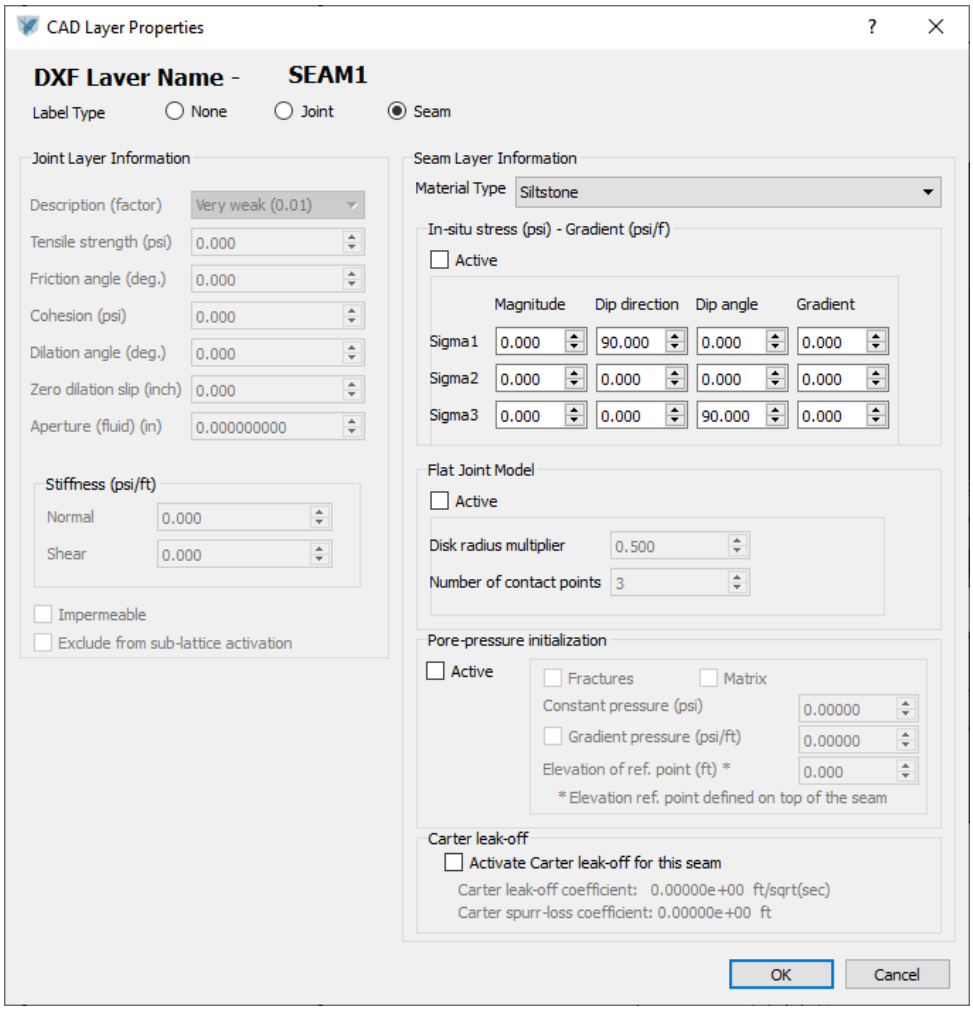


Figure 15 CAD Layer Properties Dialog

Import FLAC3D Block model (stresses and lithology)

This feature allows importing stresses and lithology from a *FLAC3D* block model defined in a comma delimited file. The format of the file is:

x, y, z , sxx, syy, szz, sxy, sxz, syz, material_type

where:

"x, y, z" are the names of the columns where the location of the centroid of the *FLAC3D* zone is defined.

"sxx, syy, szz, sxy, sxz, syz" are the names of the columns where the zone stress components are specified.

"material_type" is the name of the column where a material type name defined in XSite is specified.

Before importing the model, the material type of the imported "material_type" column must be specified, and the model must be initialized. The dialog in Figure 16 will be displayed when this menu option is selected.

In this dialog, the CSV file name must be specified. Coordinates, stress components and the reference stress may be specified in the Imperial or metric unit system. If the field **Use global coordinate system** is used, the data will be translated to the local coordinate system. The reference stress is subtracted from the imported stress values.

The **Set Geometry** button may be used to set the geometry of the model. To use this feature, the model should not be initialized, and the **File Info** button should be pressed. After the button is pressed, exit the dialog. Use the element **FLAC3D Block Model** of the **Sketch Model** plot to verify if the new geometry is correct. The element FLAC3D block model must match the element **Rock** in the plot item. After verifying that the geometry is correct, reinvoke the dialog to import the data.

The **Reset** button allows the user to import a different file without exiting the dialog. The **File info** button reads the information from the file allowing the user to verify if the data are being correctly read into the model. The **Import** button will read the data and apply it to the model. When the import operation is completed, a dialog box will be displayed informing the number of zones mapped into the XSite model.

The workflow for importing the data is the following:

1) Generate the CSV file as described above in *FLAC3D* (or other continuum software).
2) Create the XSite model, ensuring that the model has the correct geometry and that the materials specified in the CSV file are defined in XSite.
3) Ensure that the XSite grid length is properly defined. It is recommended for the grid length to be at least 4 times the background resolution length or the same length of the *FLAC3D* zone edge.
4) Initialize the model and run the model to equilibrium.
5) Import the CSV file.
6) Use the **Materials/Uniform** plot item to verify if the seams are correctly imported. Use the **Contour/Stress** plot item to verify if the stress was properly installed in the model. It is useful to add a cut-plane and move it to the location where you want to verify the stress.
7) Equilibrate the model again before running the simulation.

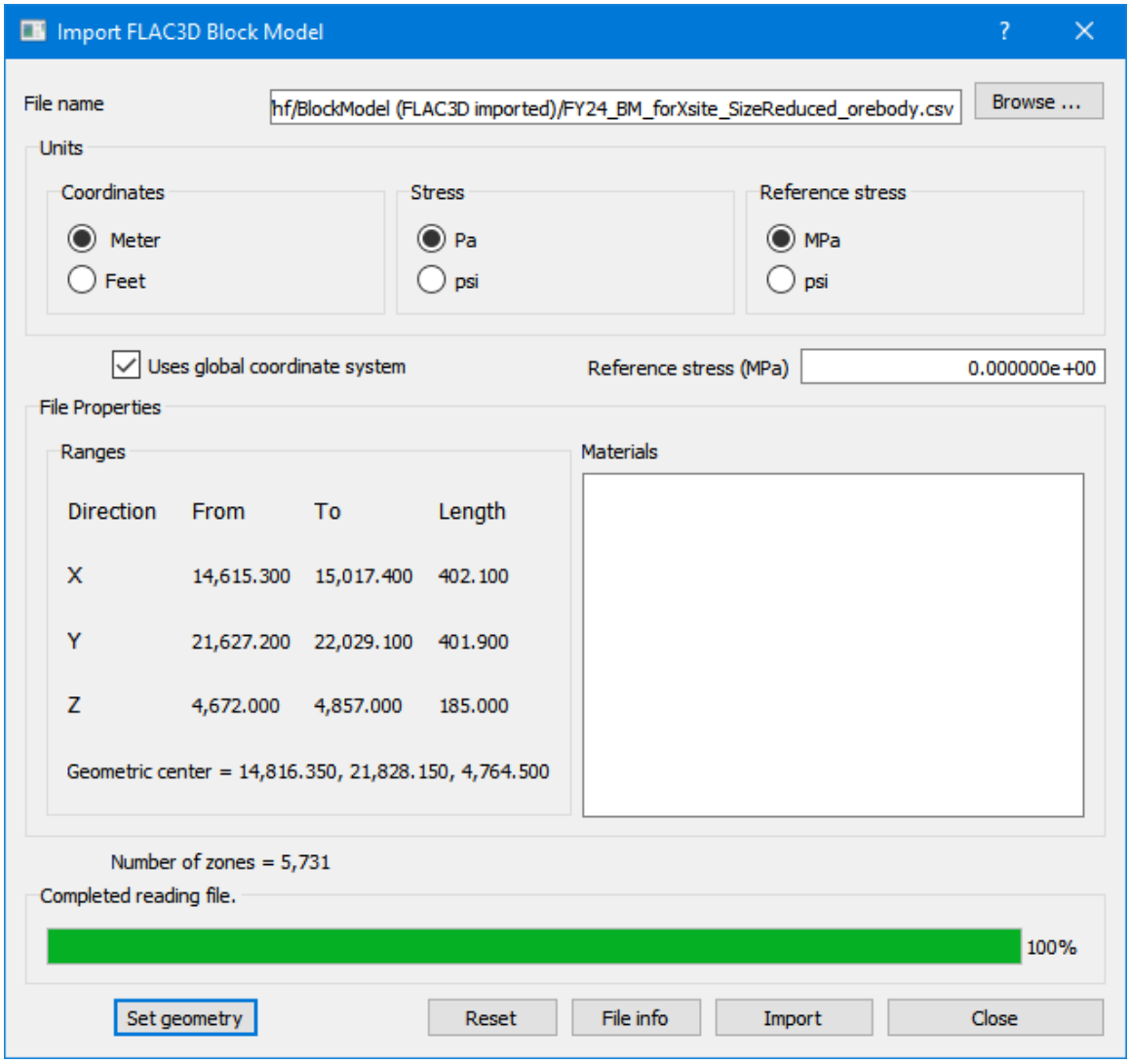


Figure 16 Import FLAC3D block model dialog.

Figure 17 below shows a model with imported stresses after running 5 seconds for equilibrium.

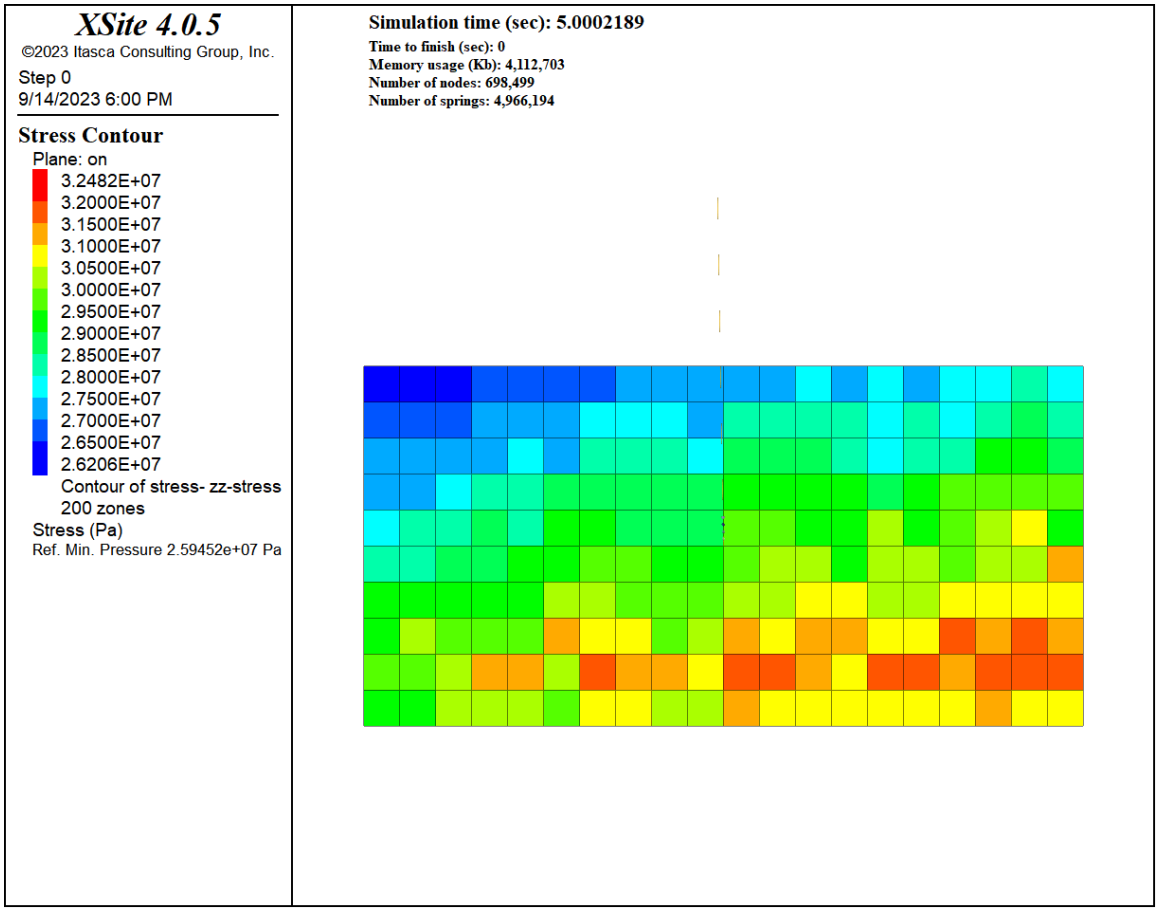


Figure 17 Stress contour imported from FLAC3D block model.

Import Microseismicity Data

This feature allows importing microseismicity data from a comma-delimited file. This data can be used to compare with the microseismicity events produced by XSite during model calibration. The dialog in Figure 18 will be displayed when this menu option is selected.

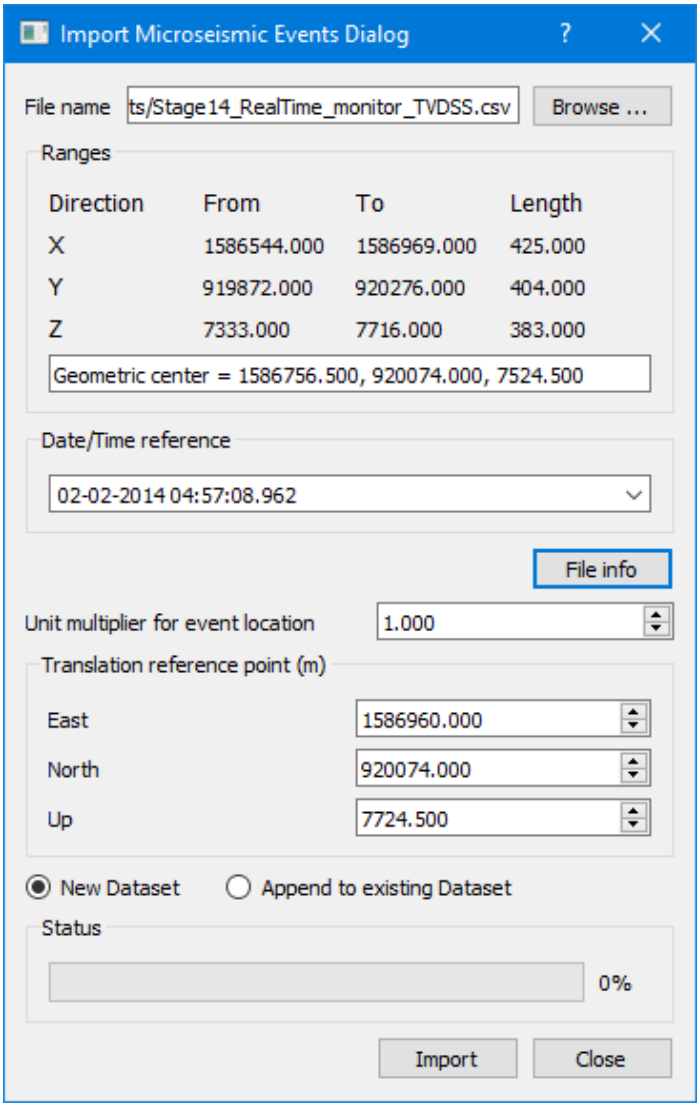


Figure 18 Import Microseismic Events Dialog.

The geometric information displayed in the **Ranges** box can be used if the data in the file is given in global coordinates and needs translating to the XSite local coordinate system. This can be done either by populating the values of the vector in the **Translation** box in this dialog or populating the **Reference Point** vector in the *Main Rock/Geometry* tab. The values for the **East** and **Up** points were obtained from the **To** column of the **Ranges** table. The value for the **North** coordinate is the value of the **To** column minus half of the length of the *y*-direction. Figure 19 shows the imported data into the 450 × 450 × 450 m model. The reference point and model dimensions are highlighted. The **Imported Microseismic Events** plot shows the locations and magnitudes of the events.

Data are imported from a comma-delimited file with the following fields:

- *x*, *y*, *z* coordinates
- Date in the format MM-DD-YYYY or MM/DD/YYYY

- Time in the format hh:mm:sec.milsec (hh in 24 hours)
- Amplitude
- Extra field, e.g., the stage that the event has occurred

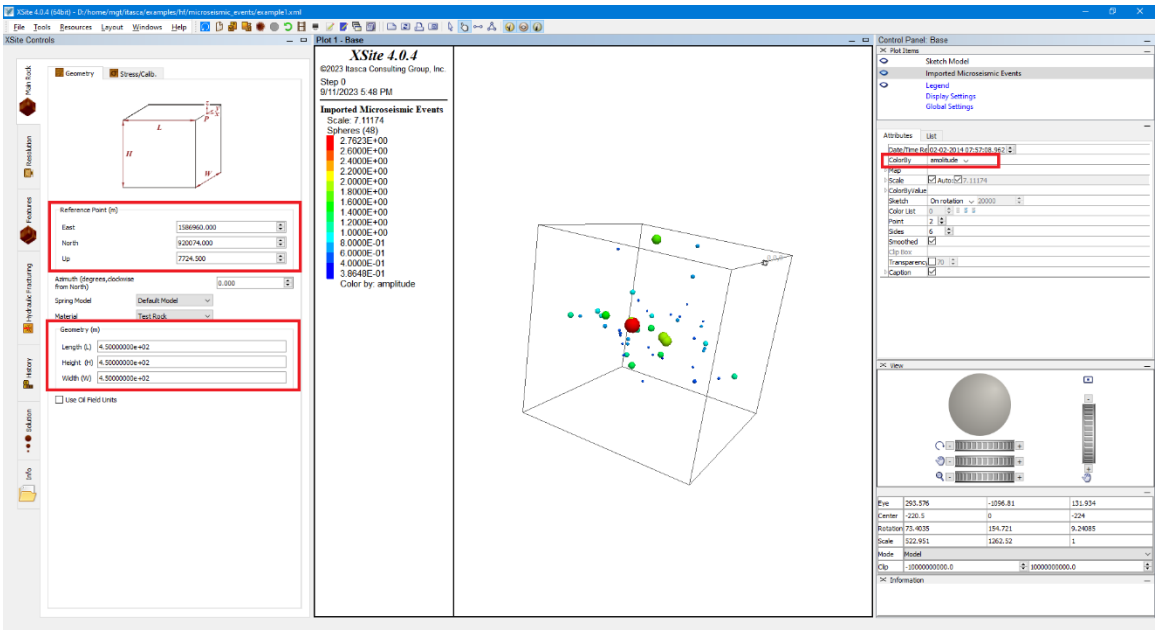


Figure 19 Plot of imported microseismic events.

Quit

Exits the application.

Tools

Tool items include: **Options**, which can be used to modify the start-up of XSite, display settings and configure movie settings; **Tooltips**, which enables or disables "hints" to be displayed when the mouse cursor hovers over icons in the tool bar; **Batch Simulation**, which allows a user to run an unattended set of simulations with different parameters; **Parametric Studies**, which allows a user to create different simulation scenarios based on the current models; and **Test sub-lattice activation** and **Test sub-lattice connectivity**, which are used to test or verify sub-lattice models.

Options

The **Options** dialog shown in Figure 20 is where the extensive range of user-specified settings in XSite are controlled. The dialog is divided into three areas of functionality. Each section and the controls within it are described in the following sections. They are: Startup Settings, Display and Movies.

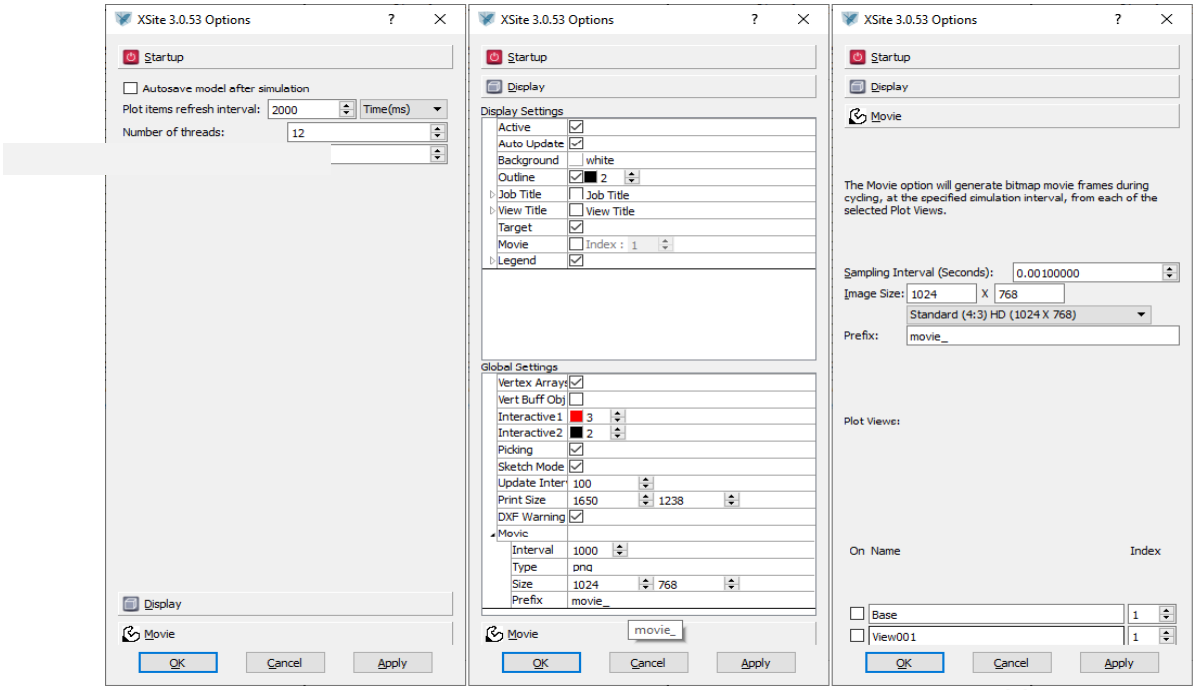


Figure 20 Options dialog boxes for controlling program (a) Startup, (b) Display and (c) Movie.

Startup Settings

Autosave Model After Simulation

Controls whether a .sav file is written after each simulation step is completed. For safety, this option may be selected (because the model can be restored after a crash, for example), but large models may take a long time to save.

Plot Items Refresh Interval

Allows the interval between plot refreshes to be specified in terms of clock time or number of simulation steps.

Number of Threads

The code automatically detects the number of threads (e.g., processors) available from the hardware. This option allows the user to specify manually the number of threads to be used. Using more than 16 threads does not increase simulation speed.

Display Settings

These controls are the defaults for newly created views. The controls available here are duplicated locally for each specific view as the **Display Settings** and **Global Settings** plot items that appear on the **Plot**

Items list for each view. Those local instances of the controls on a given view can be used to "override" the general preferences set here.

Active

When checked, each plot item will be active upon creation; otherwise, each plot item will be inactive upon creation.

Auto Update

When checked, the plot will automatically update at a given interval (specified using the **Update interval control** in the **Global Settings** section below). If not checked, the plot must be updated manually using the **Regenerate Current Plot** tool available on the toolbar.

Background

Indicates/sets the default background for plots using the pop-up color selector available from the indicator switch.

Outline

Indicates/sets the color and size of the displayed (if checked) plot area outline, where the plot area is the rectangular part of the view that excludes the legend.

Job Title

Indicates/sets the color, size, font-face and style of the displayed text (if checked) of the job title. When displayed, the job title appears in an outlined rectangle at the top of the plot area, where the plot area is the rectangular part of the view that excludes the legend. The job title text is set in the **General** section of the **Options** dialog.

View Title

Indicates/sets the color, size, font-face, style and text of the displayed view title (if checked). When displayed, the view title appears in an outlined rectangle at the top of the plot area, where the plot area is the rectangular part of the view that excludes the legend. It will appear below the **Job Title**, if that item is set to be displayed as well.

Target

Indicates/sets if the target view square will be displayed in the plot. All graphical objects in the target view square will be rendered regardless of the aspect ratio of the view window.

Movie

When checked, each new view will be a source for an output movie with specified initial index (used to name the successive movie output bitmaps).

Legend

Provides sub-controls that may be used to configure the default appearance and position of the legend and to specify what items (time, step, customer, title, view info, etc.) are or not included in the legend.

Vertex Array

Specifies/indicates whether vertex arrays will be used to draw objects in OpenGL. The default is on. Turning this off can sometimes improve images on older OpenGL display drivers. Note that turning this off implies vertex buffer objects are off as well.

Vert Buff Obj

Specifies/indicates whether the vertex buffer object OpenGL extension will be used if available. The default is on. Turning this off can improve images on drivers that report this extension as being present but do not properly support it.

Interactive1

Indicates/sets the color and thickness of the highlight outline that is used to delineate any interactive object (plot item, interactive range, legend, etc.) that appears in a plot and is selected while the manipulate mouse mode is active.

Interactive2

Indicates/sets the color and thickness of the handlebar points that appear on the highlight outline of any interactive object (plot item, interactive range, legend, etc.) that appears in a plot and is selected while the manipulate mouse mode is active.

Picking

Activates/deactivates picking, which allows for interactivity with the rendered node(s) on screen. As picking can slow plot rendering, turning picking off can be desirable in some cases.

Sketch Mode

Activates/deactivates sketch mode, which is a reduced rendering method that is faster than full rendering. Plots containing very large numbers of items to be rendered can be drawn and manipulated faster in sketch mode.

Update Interval

Indicates/sets the interval (number of calculation cycles or steps) at which a view will be regenerated. Note that views are always regenerated at the end of any simulation stage, regardless of the update interval.

Print Size

Specifies/indicates the default size (x and y dimensions, respectively, in pixels) of bitmap output sent to the printer.

DXF Warning

Specifies/indicates whether to display the DXF warning when exporting a view to a DXF file; the warning is a reminder to the user about the limitations of the DXF export functionality.

Movie

Specifies/indicates the interval for movie frame capture, the format type for the bitmap frames, the size of the bitmaps, and the file name prefix to be used (in conjunction with the index number) to name the file. Note these settings are global and will be used by any view that has been marked to generate movie bitmaps during cycling.

Movie Settings

Any open view in the program can be used to generate a sequence of .png files taken periodically over the duration of the simulation. These files can be combined into a movie using software tools like MovieMaker. This section provides a listing of all currently available views. Each individual view has a "movie" setting on its **Display Settings** plot item that can also be used to set the view to be the basis of a movie. Note that these controls are interchangeably live: setting the movie to "on" for an individual view will cause it to appear switched on in the **Options** dialog. Switching it "off" in the dialog will cause it to also become switched off on the view's **Display Settings**. Also note that the first five settings are global; changing the value provided for any of these for an individual view — using that view's **Global Settings** plot item will change the same setting for all views that are set to generate movie output.

Sampling Interval (seconds)

Sets the interval in seconds that should elapse before each frame capture.

Image Size X: (and Y:)

Specifies the dimensions of the captures, in pixels.

Prefix

Specifies a string that will be pre-pended to the file name of each capture file.

Plot Views

The Plot Views section lists all the available views. Individual views are selected to be made movies by selecting their checkboxes; the Index field is used to specify the starting point for numbering the captured frame files. All capture files are placed in the current project directory.

Batch Simulation

When the **Batch Simulation** menu option is selected, the dialog box in Figure 21 will be shown. The **Batch Simulation** can also be activated via the toolbar button  . The **Batch Simulation Steps** dialog allows the user to create a set of simulations steps and execute the entire set unattended, in a batch fashion without intervention. A simulation step is a set of instructions that will be sent to the XSite engine to be executed. Some of the instructions may be executed pre or post simulation command (cycle). The simulation steps will be executed in the sequence that they were created. Batch steps that have the columns *State*, *Start* and *Finish* filled (which means that they have been already executed) will not be executed. In the example below, step 1 is "Completed" and step 2 is currently "Running". The other steps are scheduled to be executed next. The **Batch Simulation** data is stored in the .xml file.

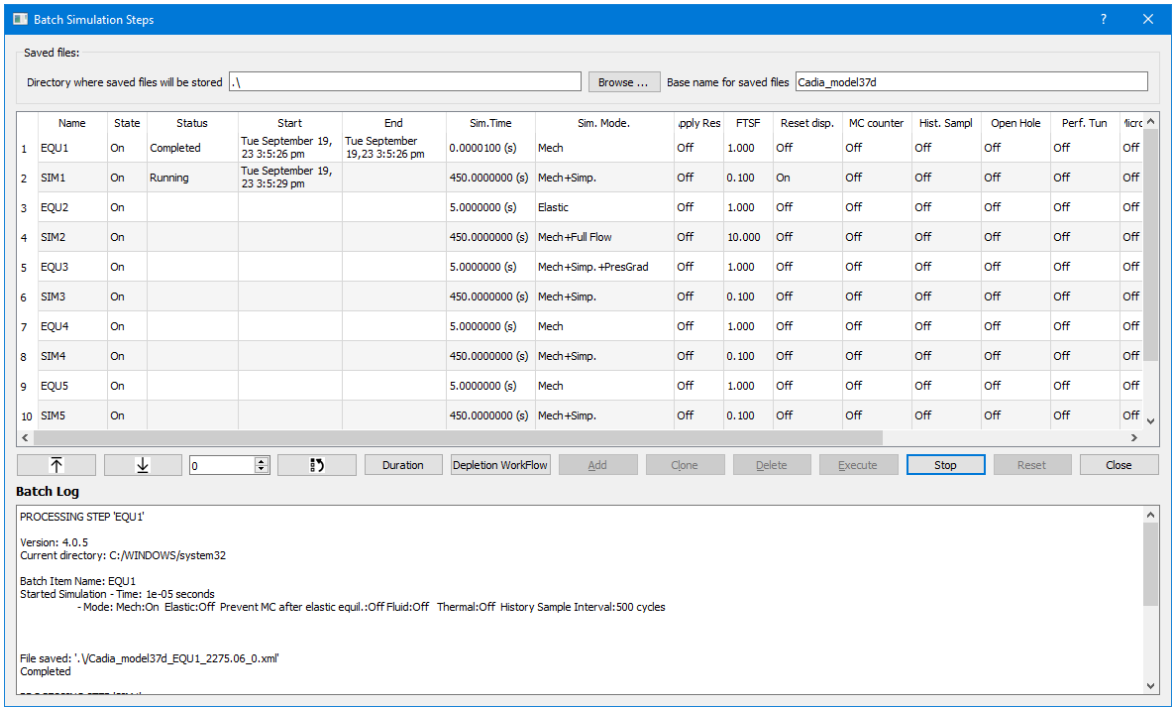


Figure 21 Batch Simulation Steps dialog.

The field **Sim. Time**, or simulation time, sets the simulation time for that simulation step. **Sim. Mode** describes the simulation mode, for instance, step 4 will be simulated as mechanical simulation coupled with fluid flow analysis. The column FTSF displays the fluid time step factor for that step. If FTSF is greater than 1.0, the program will use the implicit method to solve the fluid flow equations.

The field **Directory where the saved file will be stored** contains the file path. The path may be absolute (e.g., "c:\user\xsite\data") or relative (e.g., "/", which indicates that the file will be saved in the current directory). Either forward slash, /, or backslash, \, may be used to compose the path. For the Linux version it is recommended to use forward slash.

The field **Base name for saved files** is used to compose the name of the files that will be saved. The Post-process section describes how the saved file name is composed.

This dialog has the following operations available:

Buttons ⤓ and ⤒: Moves the currently selected batch entries up or down in the list.

Button : Moves the currently selected batch step to the line indicated in the box located to the left of this button.

Duration: Displays the execution time in minutes of the selected batch tasks.

Depletion workflow: Invokes the dialog box described in Appendix 1. This dialog creates batch steps necessary to perform simulation of parent-child well stimulations.

Add: Invokes the **Batch Simulation** dialog shown in Figure 22. In this dialog, the user will be able to add a new simulation step. All the simulation steps created by the user will be listed in **the Batch Simulation Steps** dialog (Figure 21). By double-clicking one of the items in the list, the **Batch Simulation dialog** may also be invoked, allowing editing of the selected item.

Clone: Creates a copy of the simulation step that has been previously selected (left-click selects an item in the list).

Delete: Deletes the simulation steps that have been previously selected (left-click an item in the list).

Execute: Executes the batch of simulation steps. When this button is pressed, the simulation steps will be executed in the order that they appear in the list. Only the blank items in the **Status/Start** and **End** columns will be executed. The item that is currently executed will show "**Running**" in the status column. The status will change to "**Completed**" when the item finishes execution. The **Start** and **End** columns display the start and finish times of a simulation step. The **Start Simulation** button on the toolbar remains inactive during the batch execution. If the **Stop Simulation** button on the tool bar is pressed, the current batch step will stop execution, the post-processing sub-steps will be executed and batch process will process the next batch steps.

Stop: Stops the batch execution. The status column of the last item being executed will show "canceled". This button is inactive until the **Execute** button is pressed and will remain active only during the batch execution.

Reset: Sets the **Status/Start/Complete** columns for all simulation steps to blank, allowing the user to re-run the batch mode from the beginning.

Close: Dismisses the dialog.

The fields located in the **Saved files** group box will be used in conjunction with the **Post-process/Save State** field of the **Batch Simulation** dialog. The **Directory where saved files will be stored** indicates where the saved files will be created. The field **Base name for saved files** is the prefix used in composing the name of saved files.

The **Batch Simulation** dialog (Figure 22) will create a simulation step and is invoked when the user presses the **Add** button. Editing an existing batch step is available by double-clicking an item in the batch step list.

The field **Name** in the **Batch Simulation** dialog is used to identify a specific batch step and to compose the name of the files that will be saved. The **Post-process** section describes how the saved file name is composed. A unique name is recommended in this field.

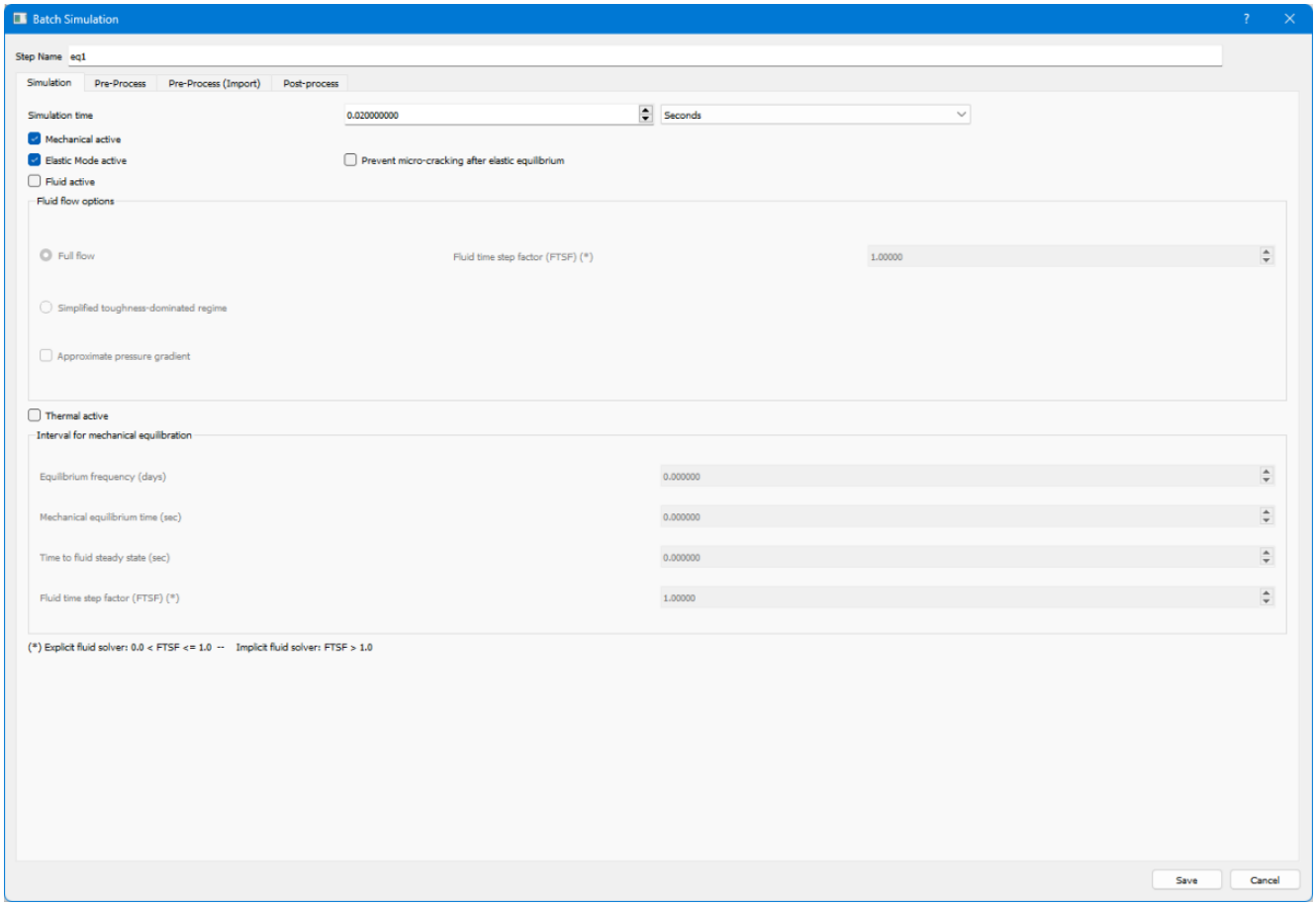


Figure 22 Batch Simulation Dialog.

The **Batch Simulation** dialog has four tabs: **Simulation**, **Pre-process (1)**, **Pre-process (import)** and **Post-process**, described below.

Simulation tab

The fields in the **Simulation** tab are shown in Figure 21.

The **Step Name** field is used to identify the batch step. This field is not mandatory, and it is used to compose the name of file that may be saved for this step, see Save state field on the **Post-process** tab.

The other fields shown in this tab are described in the File/Start Simulation section.

Figure 23 shows the **Pre-process** dialog. Description of the fields of this dialog follows.

Pre-process tab

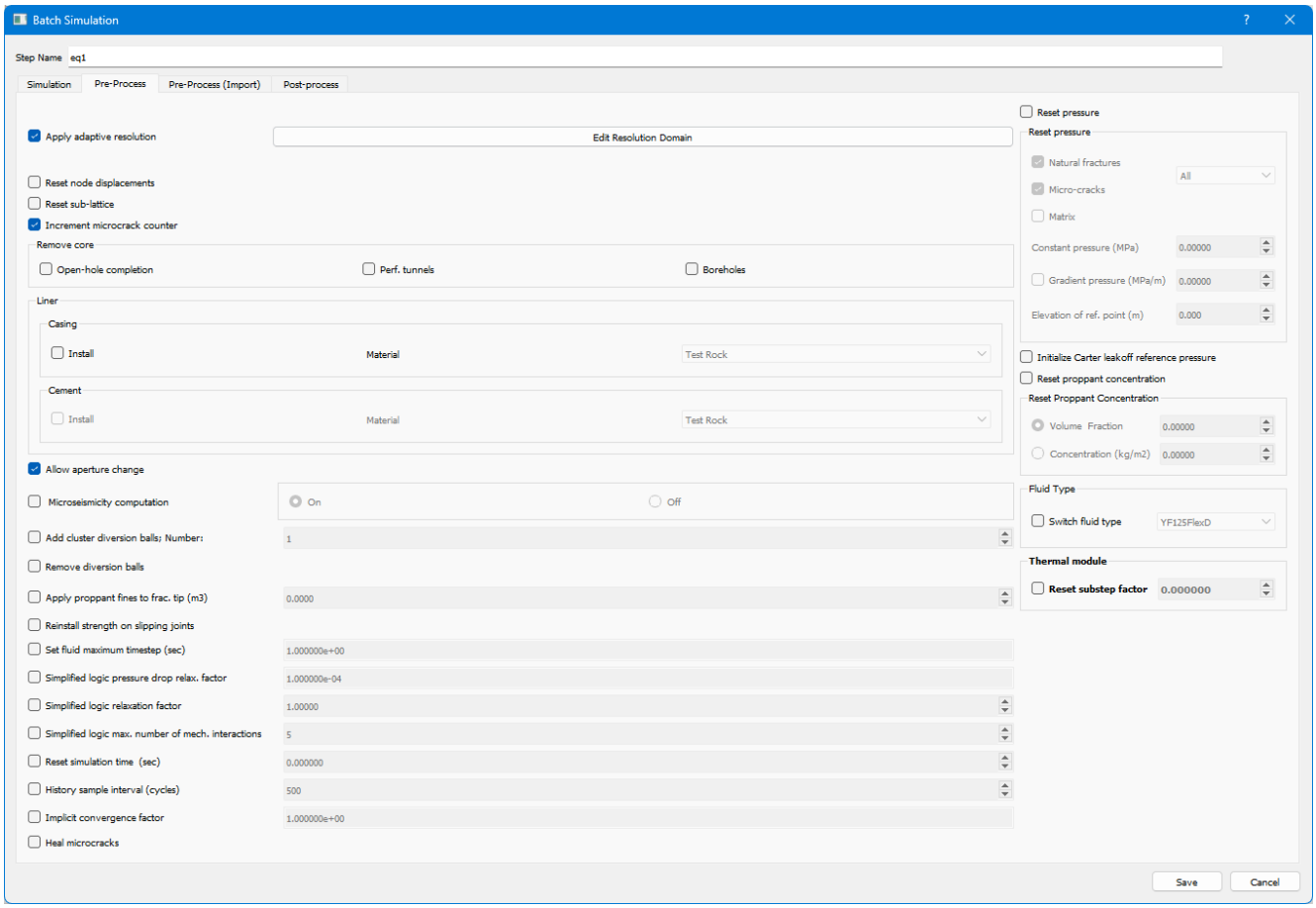


Figure 23 Pre-process (1) dialog.

Apply adaptive resolution

XSite allows users to change the lattice and distribution of different resolutions throughout the model domain (i.e., "remeshing") during the same simulation. This capability allows optimization of the model size when the location of the source of perturbation moves within a model (e.g., simulation of multiple stages along a well). With adaptive resolution, the finest resolution domain in the model is always only around the location of the source of perturbation in the model (e.g., currently active stage). All resolution domains must be defined during the model setup, before the start of execution. The resolution domains that are not currently active are assigned the background (coarse) resolution.

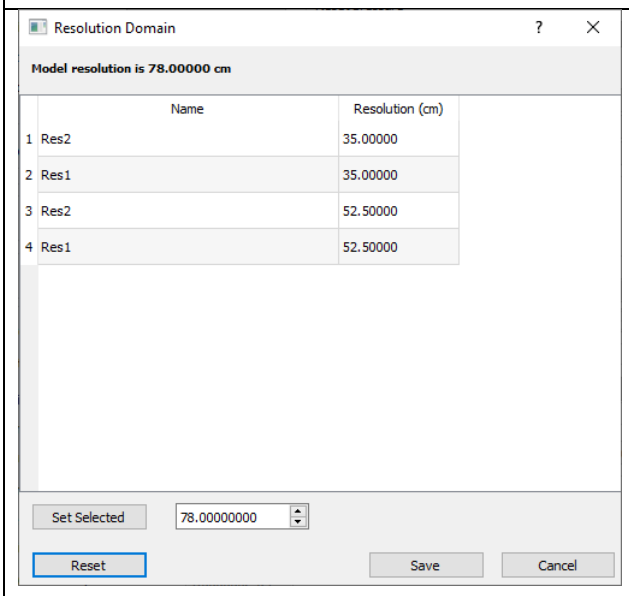
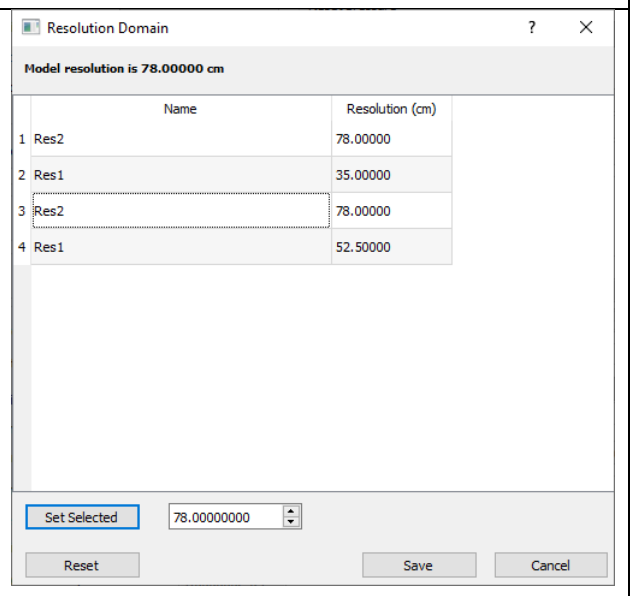
By checking the **Apply adaptive resolution** box, the dialogs shown in Table 1 will be displayed. In this dialog box, the user will be able to specify the adaptive resolutions that will be used for this batch step. The left column shows all resolution domains as defined in the Resolution tab. Initially each resolution domain is assigned the finest resolution, intended for that domain when it is active (i.e., injection occurs at the location within that resolution domain). The right column shows the state when the domains that will be activated are assigned fine resolution, and the remaining domains (inactive) are assigned the

background resolution. The following steps are recommended to apply the desired resolution to the model:

1. Press the **Reset** button to have the original resolution set.
2. Ensure the background resolution is correctly displayed in the edit box located on the left of the **Set Selected** button. Select the resolution domains that are associated with stages that **will not** be active during this simulation phase and press the **Set Selected** button.

In the example presented in Table 1, two resolution domain groups, Res1 and Res2, are defined. Each group has two resolution domains, with resolutions of 35 cm and 52.5 cm, as shown in the left column. The right column indicates that the group of resolution domains named Res1 (associated with stage 1) will be activated, and the resolution domains associated with group Res2 will be deactivated: the resolution in group Res2 will be set to the background resolution, 70 cm.

Table 1 Resolution Domains

Specified Resolution Domains	Resolution domains to be activated
	

Reset node displacements

Checking this box causes all accumulated displacements to be set to zero. Note that this does not affect the physics of the simulation — it simply allows the displacements that occur during a given stage of a simulation to be accumulated and displayed independently of previous stages. The deformation and stresses are not affected by resetting displacements. The code keeps track of the total and relative displacements. This action resets the relative displacements only. The total displacements cannot be

reset. By resetting the displacement, it is possible to differentiate displacements induced by each stage. Resetting node displacement also gets rid of noisy displacements after initial equilibration of the model.

Reset sub-lattice

Selecting this box will deactivate the active sub-lattice zones. Only active nodes and springs from active zones will be included in the mechanical computation, allowing for a more efficient simulation. This procedure should be executed at the beginning of a new stage on a well.

Increment microcrack counter

Selecting this box causes an identification number associated with microcracks to be incremented. If the counter is checked at the beginning of simulation of different stages, this allows microcracks to be displayed in different colors, according to the stage of the simulation in which they occurred. The microcrack counter for each new microcrack is initialized to zero. The microcrack counter of all existing microcracks is incremented if the microcrack counter is checked when a new simulation step is started. The **Microcrack** plot item can use the microcrack counter to display microcracks with different colors.

Remove openhole completion, perforation tunnels and boreholes core

Selection of these checkboxes removes springs and nodes from openhole completion, perforation tunnels and boreholes defined in the **Borehole/Segment Dialog**. This operation will add fluid elements to the borehole boundary springs, allowing injected fluid to initiate or propagate the fracture. This operation works as if material inside openhole completion, perforated tunnels and boreholes was excavated from the model. The **Materials/Uniform** plot item indicates items as excavated regions. This operation is typically used for near-borehole simulations in small-scale models.

Install Casing and Cement

In the **Borehole/Segment Dialog** a borehole segment can be specified having casing and cement. In this case, if these checkboxes were checked, the code will replace the model background material with the material specified in the drop-down lists located to the right of the checkboxes. The thickness of the casing and cement are specified in **Segment Dialog**. This operation is generally executed after the borehole is removed and the model achieved an equilibrium state.

Allow aperture changes

If this checkbox is left unchecked, hydraulic aperture of the joints and fractures will not change during the simulation. The value set in this checkbox will override the field **Allow aperture change** in the **Solution/Fluid** tab.

Microseismicity computation

Select this checkbox to activate or deactivate microseismicity computation. If the checkbox is checked, microseismicity events will be recorded. Microseismicity events are caused by bond breaking in the rock matrix and sliding of joints. It is possible to combine (cluster) shear events using the **Reset Lattice** dialog. When combining shear events, the largest N (the number specified by the user) events will be considered as outliers and removed from the computation.

Add diversion balls

If this checkbox is selected, the number of diversion balls specified in the edit box located to the right of the checkbox will be applied to the clusters of the active stage that take most of the injected fluid (i.e., have the greatest flow rate), effectively plugging them. For example, if the active stage has 4 clusters and each cluster has 12 perforations, the total number of diversion balls that can be added is 48, plugging the entire stage. If 24 balls are added, two clusters with greatest flow rate will be plugged.

Remove diversion balls

The selection of this checkbox will cause the code to remove all diversion balls applied to the model, regardless of the stage/cluster state (i.e., active, or inactive).

Apply proppant fines

If this option is checked, the fluid nodes with the smallest aperture will be plugged (starting with the smallest aperture in the ascending order) until the volume specified in the edit box located to the right of the checkbox is reached. No transport of fines is simulated. They are assumed to be instantaneously placed at the fracture tip.

Reinstall strength on slipping joints

The faults (or joints) in the model can be assigned initial strength and to be impermeable under initial conditions, meaning they become permeable only after they slip or open (i.e., fail in shear or tension) and lose cohesive and tensile strength. Forces are initialized and the full equilibrium in the model is achieved by running the model mechanically for a few seconds. During that equilibration phase, some of the contacts within joints can slip, which will change the desired initial condition that joints have cohesive strength and are impermeable. Selecting this checkbox reinstalls the initial joint strength and makes them impermeable.

Set maximum fluid timestep

Allows user to override the field **Maximum fluid timestep** defined in the **Solution/Fluid** tab.

Simplified logic pressure-drop relaxation factor

Allows user to override the field **Perforated pressure drop relaxation factor** defined in the **Solution/Fluid** tab. This parameter is the relaxation factor used to calculate the cluster perforation pressure drop for the simplified logic. The default value is 0.0001.

Simplified logic relaxation factor

Allows user to override the field **Relaxation factor** defined on **Solution/Fluid** tab. This parameter is the relaxation factor used in the relaxation scheme for calculation of the fracture pressure in the simplified logic. The default value is 1.0. The relaxation parameter of the relaxation scheme is proportional to the reservoir rock Young's modulus. The relaxation factor is a multiplier of the Young's modulus. Values greater than 1.0 result in stiffer response, with faster pressure changes that can result in faster convergence of the relaxation scheme, but also might result in instability. Values less than 1.0 result in smoother pressure changes and a generally slower but more stable numerical scheme.

Simplified logic max. number of mechanical interactions

This variable sets the maximum number of mechanical steps that can be executed in the relaxation scheme to bring the unbalanced volume (i.e., difference between fluid nodes volume and the injected fluid volume) to zero. The relaxation scheme is used in the simplified logic for simulation of fracture propagation in the toughness-dominated regime. The code will execute fewer mechanical steps than the maximum amount if the convergence criteria in terms of the relative unbalance volume has been reached. See the maximum number of mechanical interactions field in the **Solution/Fluid** tab.

Reset simulation time

This operation will allow the user to change the current simulation time. For example, it can be used in cases in which the mechanical time estimated to equilibrate the model is insufficient and additional mechanical simulation is needed that would advance the simulation time beyond the time when injection starts. The user can run the equilibrium phase longer and reset the simulation time without having to adjust the start and finish times in the pumping schedule. The history record will be out of sync if the simulation time is reset.

History sample interval (cycles)

The sampling interval for all histories defined in the model can be set for each simulation step independently. A short history interval in a very long simulation may cause the save file to be large.

Implicit convergence factor

The maximum relative error used in the convergence criterion of the implicit logic can be set for the simulation step. This value overrides the default set in Solution/Fluid.

Heal microcracks

This operation will close microcracks (make them elastic) that are not connected to the rest of the pipe network (i.e., other microcracks or pre-existing joints).

Reset pressure

Resets pressure in the fracture and/or matrix fluid. The pressures can be reset before any stage. For example, the pressures can be reset to the initial hydrostatic state to approximate flow back and/or long-term leakoff. If the **Reset pressure** checkbox is not checked, the existing pressures will be the initial pressures for the next simulation stage. The dialog allows application to different ranges within the model, including natural fractures, microcracks and matrix. The pressure can be reset to a constant value or with constant gradient in the *z*-direction. If gradient is used, the constant pressure is defined for the reference *z*-coordinate. After resetting the pressure, the model should be run to mechanical (no fluid flow) equilibrium to simulate fracture closure and associated stress change.

Initialize Carter leak-off reference pressure

If the Carter leak-off is used, the exposure time is measured from the state when the fracture pressure exceeds the reference pressure.

Reset proppant concentration

Resets proppant concentration in the entire model. The reset proppant concentration can be specified as a volume fraction or as a concentration (mass per volume, e.g., kg/m^3).

Fluid Type

Fluid type can be changed between simulation steps. Any fluid type from the **Fluid Item** list can be selected. It should be noted that the effect of a fluid type change in the model is non-physical, because it causes a change in the currently injected fluid but also all fluid in the model (fractures and matrix), and previously injected fluid is assigned properties of the new fluid type.

Thermal module/Reset sub-step factor

The default value of the thermal sub-step factor is set in the Solution/Thermal tab. The thermal sub-step factor is the number of the advection steps that are executed per one conduction step.

Pre-process (Import) tab

This dialog controls functionalities that manage parent-well model initialization in analyses of parent-child well interactions and the effects of reservoir depletion on hydraulic fracturing. See Appendix 1 – Workflow for Modeling of Effects of Reservoir Depletion (Parent-Child Wells) for details.

Post-process tab

This tab is shown in Figure 23. The fields are described in the following text.

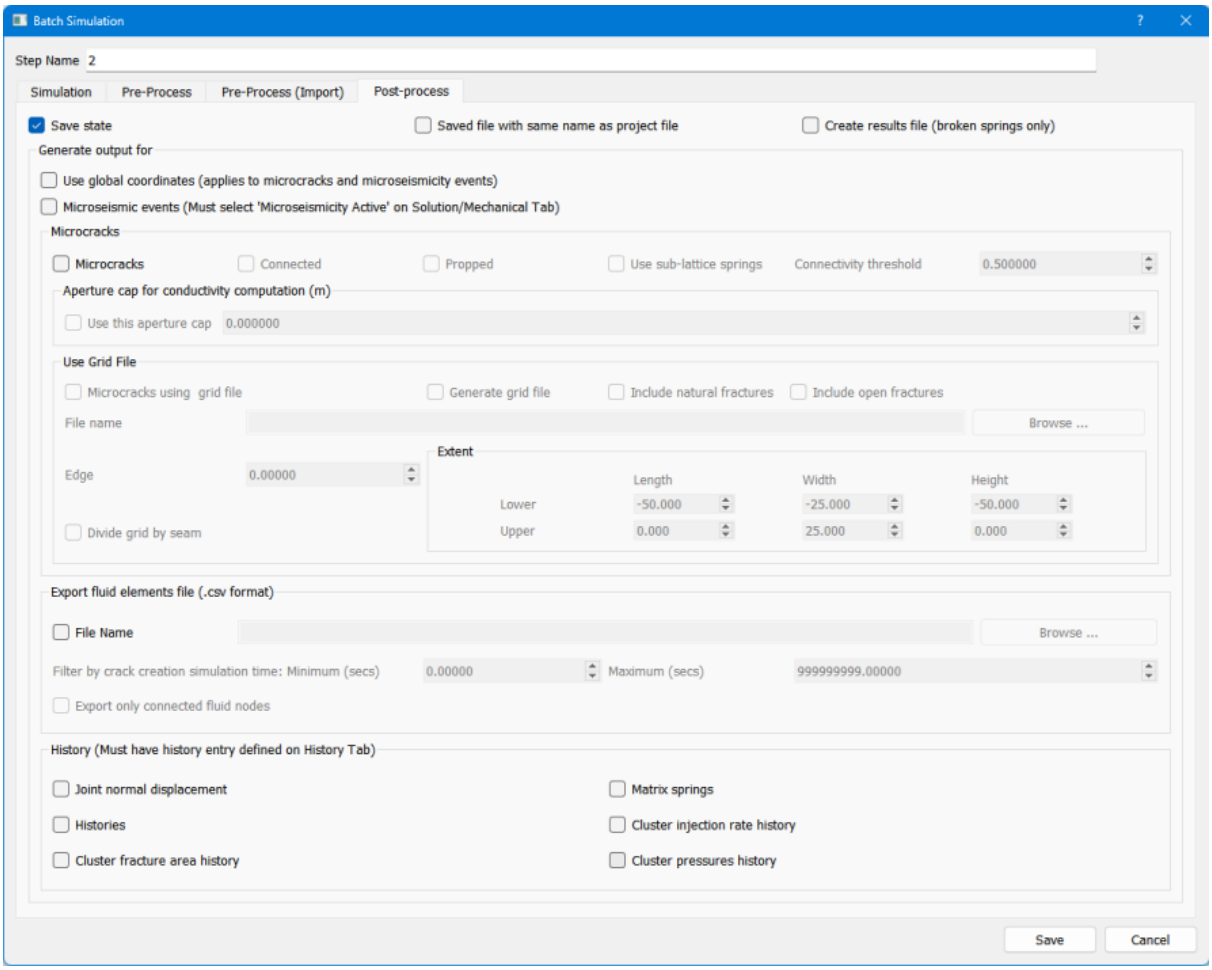


Figure 24 Batch post-process tab.

Save state

Saves the simulation state in new .xml and .sav files in the directory defined in the ***Batch Simulation*** dialog. A save file includes the entire model state with all filled variables. It allows inspection of the model state at the simulation step (time) when the save file was created. The saved file name has the following format: "Base file name" + "_" + "batch step name" + "-" + "_" + "current simulation time" _. Simulation can be continued from the save file.

Saved file with same name as project

The name of the created save files will be the same as the project (the original .xml file) name. If this option is used for multiple simulation steps, each stage will overwrite the previous one and there will be only one (final) save state at the end of the simulation.

Create results file (broken springs only)

Only the results file will be created. The advantage of the results file is that it is smaller than the save file. The model results can be inspected and post-processed. However, the results file cannot be used to restore the complete model and continue a simulation.

"Generate output for" group box

Use the appropriate checkboxes below to generate output in .csv formatted files for:

Use global coordinates

If this field is checked, the location of microcracks and/or microseismic events will be output in global coordinates. Otherwise, the output will be in local coordinates.

Microseismic events

A .csv file containing the microseismic events is included in the output.

Microcracks

If the **Microcracks** checkbox is selected, microcrack information (i.e., coordinates and orientation) is exported from the model at the end of this simulation step. The output can be limited to the connected cracks to the injection cluster and/or to the propped microcracks only. If sub-lattice is used in the simulation, used sub-lattice springs will export the information on microcracks as created in the sub-lattice (i.e., with finer resolution). In that case, **Connectivity threshold** (default 0.5) is used to define sub-lattice microcracks that are connected to the injection cluster. The sub-lattice properties, including connectivity, are interpolated from the main lattice properties. In the main lattice, microcracks can be connected or not.

If the **Use this aperture cap** field is checked and specified, the apertures used in the conductivity calculations will be limited to the specified cap.

Use Grid File checkbox

Microcrack data can be used to calculate upscaled equivalent properties such as porosity and permeability. The upscaling will be conducted and exported if **Microcracks using grid file** is checked. Upscaling can be done for an existing grid file (default) or for the grid that is generated internally by checking **Generate grid file**. If the grid is generated internally, it is defined by **Extent**, with coordinate ranges for **Length**, **Width** and **Height**, and **Edge** length. **Divide grid by seam** forces the grid to conform to the seam boundaries defined in the model. The microcracks used in upscaling can include natural fractures and the open (not propped) fractures.

Export fluid elements file (.csv format)

This group box specifies the file name where fluid elements data will be exported. Exported fluid elements can be limited to the specific time interval (**Filter crack creation simulation time**). Also, an option to export only connected fluid nodes option can be selected.

History

It allows selection of histories (including **Joint normal displacement**, **Cluster fracture area**, **Cluster pressure**, **Cluster injection rate**) to be automatically exported at the end of the simulation step. The selected histories checkbox must have a record defined in the **History** tab.

Parametric Studies

Parametric Studies allows for setup and automatic execution of parametric studies. The input parameters are defined as random variables with probability distributions. According to the defined parameters of the study (e.g., the number of simulations), XSite automatically creates sets of input parameters and simulation cases that are then executed on separate computers. After all simulations are complete, XSite automatically extracts and compiles results of interest (e.g., created fracture surface area).

The functionality of automatic execution of parametric studies can be used on multiple nodes in High Performance Computing (HPC) centers only. Running programs in an HPC center is different from running programs on a personal workstation. In an HPC center, to optimize resources for computational purposes, a program should run in console mode and the graphical interface should not be used. In fact, some of the nodes of the cluster may not even have X-Windows libraries installed, making it impossible to run graphical interface applications on these nodes. In addition, the recommended method to run simulations is by submitting jobs to a central queue. Hence, jobs are scheduled to run in an orderly fashion on the clusters, enabling computational resources to be properly used and their use maximized. Proper monitoring and accounting are also done by the central monitoring system. For example, some centers use Sun Grid Engine (SGE) to manage the scheduling queue.

XSite provides a graphical interface to setup, run and monitor simulations on HPC clusters. Although simulations can be executed on an HPC by running the model in the GUI version of the code, it is recommended that the simulations are executed using the console version of the code. The setup for execution of parametric studies launches multiple simulations using the console version of the code. Section 2.3.1 briefly describes how to set up and run XSite in a console version on an HPC.

Parametric Studies opens the dialog shown in Figure 26. This dialog has six tabs and allows the user to perform the following tasks:

- Create variations of a model for a parametric study: **Parameters for Completion design** (Section 2.3.2) and **Parameters for Field Conditions** (Section 2.3.3) tab.

- Select existing files to be simulated simultaneously: Select existing files (Section 2.3.4) tab.
- Select simulation post-process output data: Post-process Setup (Section 2.3.5) tab.
- Specify parameters to define a job that will be submitted for execution: Job Definition (Section 2.3.6) tab.
- Monitor execution: Job Status (Section 2.3.7) tab.

Each tab is discussed in more detail in the indicated sections.

Running Console Version on HPC

Setup of the Environment Variables

The Linux command **module** (assumes that the environment module package is installed) is used to set the environment variables (e.g., library and executable paths) to ensure that XSite is seen by all HPC nodes:

1. To see what versions of XSite are available in the system, type the command **module avail xsite**.
2. To load a specific version, type **module load xsite/<version>**. For example, **module load xsite/4.0.1**.
3. Typing **module load xsite** will load the default version.

This step can be automated by running the command below when you log in. On an HPC machine, edit file ".userrc" in the home directory and add the following line:

module load xsite

Logout from the account. The next time you login, XSite will be available to run on any HPC node.

Running XSite on Linux

To run XSite, the user needs to log in on an HPC machine with interactive graphics capabilities. Before running XSite, ensure that software for scheduling jobs at an HPC (e.g., Sun Grid Engine (SGE) or Altair PBS) module is loaded. The following command will load the SGE module:

module load sge

To run XSite type the command:

xsite_4.0

Running an Example on the HPC clusters

Copy the input xml file to a shared working directory, which is available to every execution host. Next, run XSite. Make sure the SGE module is already loaded before running XSite.

Load the example file in XSite. Open the **Batch Simulation**. XSite uses its console version to run a parametric study in the clusters. The console version of XSite will run the batch items defined in the XML file. Make sure that at least one batch file item is defined and that the last batch item will save the state of the run. In addition, make sure that the **Directory where saved filed will be stored** field is set properly (Figure 25, field (1)). For example, the path can be **/hpcdata/${USER}/projectname/batch**. Close the **Batch Simulation** dialog and save the dataset. Run a simulation for 0 seconds to ensure that the model is properly created.

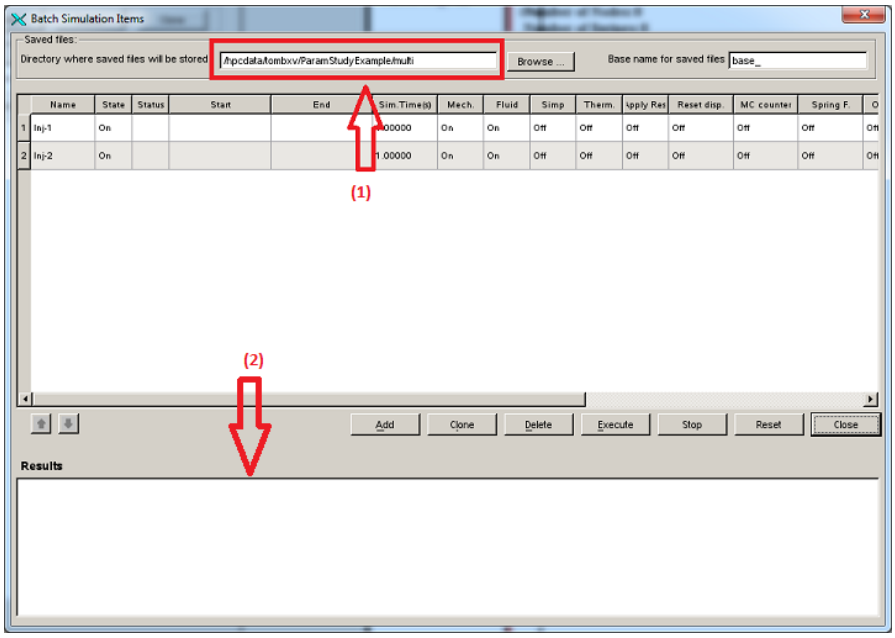


Figure 25 "Batch Simulation" dialog.

Parameters for Completion design tab.

XSite can automatically and simultaneously execute multiple models defined in the existing input files. That methodology is discussed in Section 2.3.4. Parametric study can be created automatically by XSite based on specified completion design variability and/or uncertainty in in-situ conditions (initial states or material properties).

In tabs shown in Figures 26 and 27, the user can define the variability of certain parameters to be randomly varied in a parametric study, which is conducted by creating multiple input files and their automatic execution on different cluster nodes using the console version of the code. The following notes apply to completion design, pumping schedule tables (Figure 26).

1. Double-clicking on one of the rows of the tables will open a dialog that will allow editing of the table entries (e.g., pumping schedule). The table entries can be manipulated using buttons **add**, **clone** and **delete**.

2. In the dialog that will be displayed for editing an item, the field **File ID** field will be used in the files created by the parametric study to help identify and differentiate files.
3. Application of the variable completion design requires that borehole(s) and stage(s) are defined in the model. The user may define different shapes, dimensions and parameters of the pressure loss calculation for different cluster designs to be randomly selected for stages of a borehole.
4. The **Pumping schedule** setup allows the user to define multiple simulation sequences to be used in the parametric study. The simulation sequence randomly selected from the provided list will be used for automatic generation of the pumping schedules for the wells and the batch simulation table. In the case that there are no pumping schedule items defined in this table, the user must have defined batch items and pumping schedules for each borehole.

Combining different completion designs and pumping schedules to create multiple model realizations in a parametric study is discussed in detail in Appendix 2.

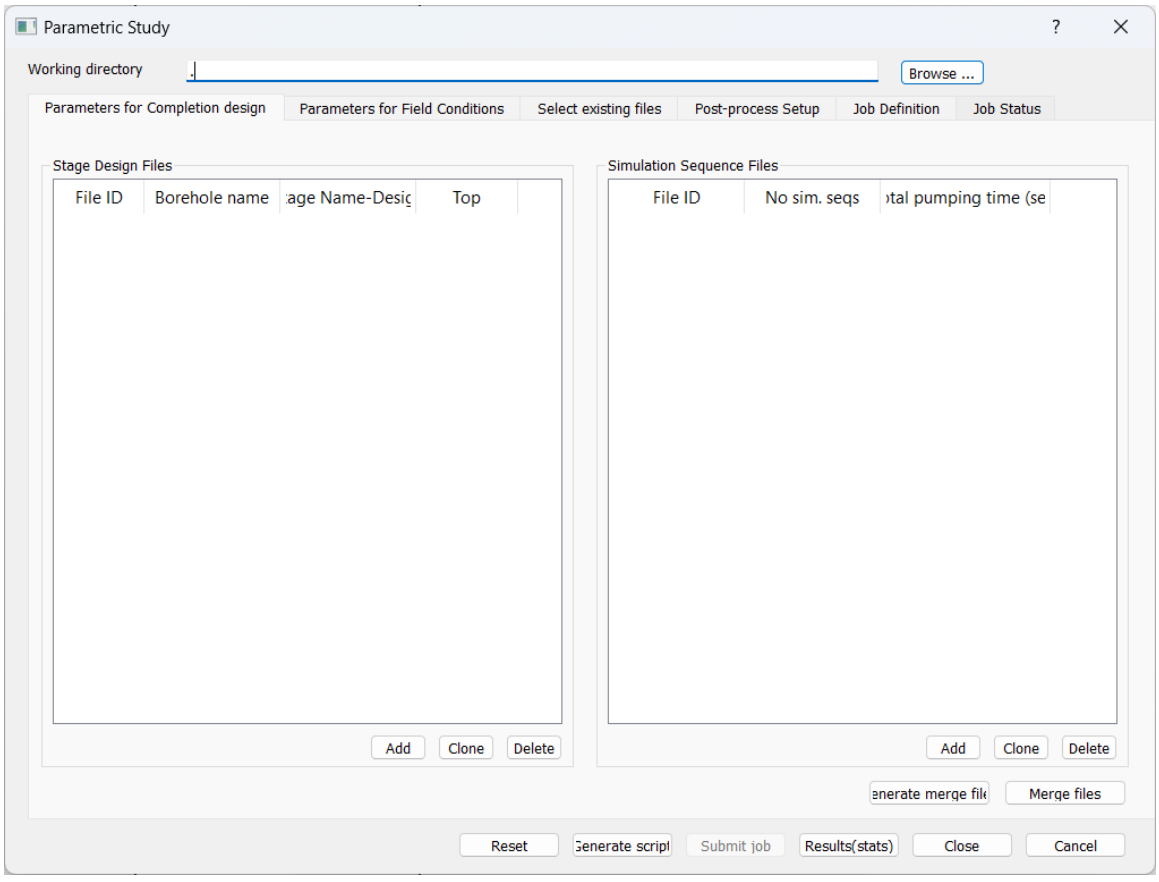


Figure 26 Parameters for Completion design tab.

Parameters for Field Conditions tab.

In Figure 27, the field Number of realizations indicates the number of runs that will be conducted as part of the parametric study with automatic selection of the input parameters and automatic generation of the input files.

To include a property or stress in the study, double-click the item and edit the standard deviation and distribution type. Then, select the checkbox to be active.

The DFN setup allows loading multiple DFNs from FRACMAN files, which must be in a directory accessible by the HPC nodes. For example, in a directory under "**/hpcdata/ ${USER}/projectname**". The user must define the joint properties and may override the joint aperture and stiffness defined in the file. In addition, the user may translate and rotate the DFN. Once DFNs are loaded and appear in the table, they are available to be used in the parametric study.When the **Generate scripts** button is pressed, one row per table category will be selected for each realization. The selection of the rows will be determined based on a random number generated using uniform distribution applied to the number of items defined for a table. Hence, if there are four DFN items defined, it is expected that each item will be used in approximately 25% of the total number of realizations.

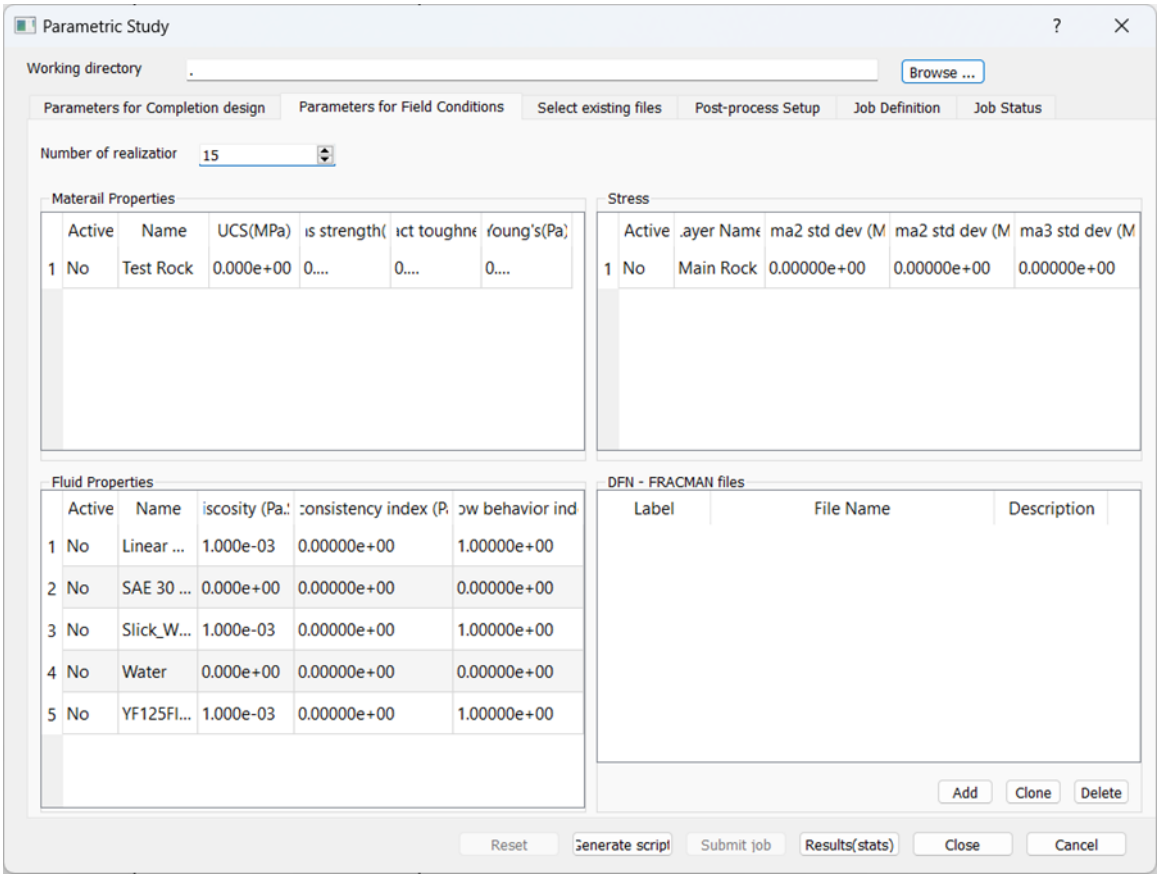


Figure 27 Parameters for Field Conditions tab.

Select existing files

To submit an execution of a parametric study with one or more existing files (manually prepared), use the **Select existing files** tab as shown in Figure 28 and carry out the following steps.

1. Prepare the files that will be used in the job. Make sure that batch items are defined and the results will be saved into an existing directory. This directory must be on a disk that has the appropriate space to store the state of the simulations and has read and write permissions for HPC center users.
2. The number of realizations is equal to the number of files that are selected for the job.
3. Enter the working directory.
4. Use the **Select files** button to browse and pick the files that will be used in the simulation. The selected files will be displayed in the **Selected files** window.
5. Use the **Copy files** button to copy the files from their original location to the working directory.
6. The **Remove files** button can be used if the user wants to remove files from the simulation.
7. When the files are selected, set the **Job Definition** parameters in the appropriate tab and press the **Generate Scripts** button. This step creates the script that will be submitted for execution.
8. Press the **Submit Job** button to start the simulation.

9. Job status can be checked on the **Job Status** tab.

Note that the simulation results will be saved in the working directories specified in the batch specification for each file.

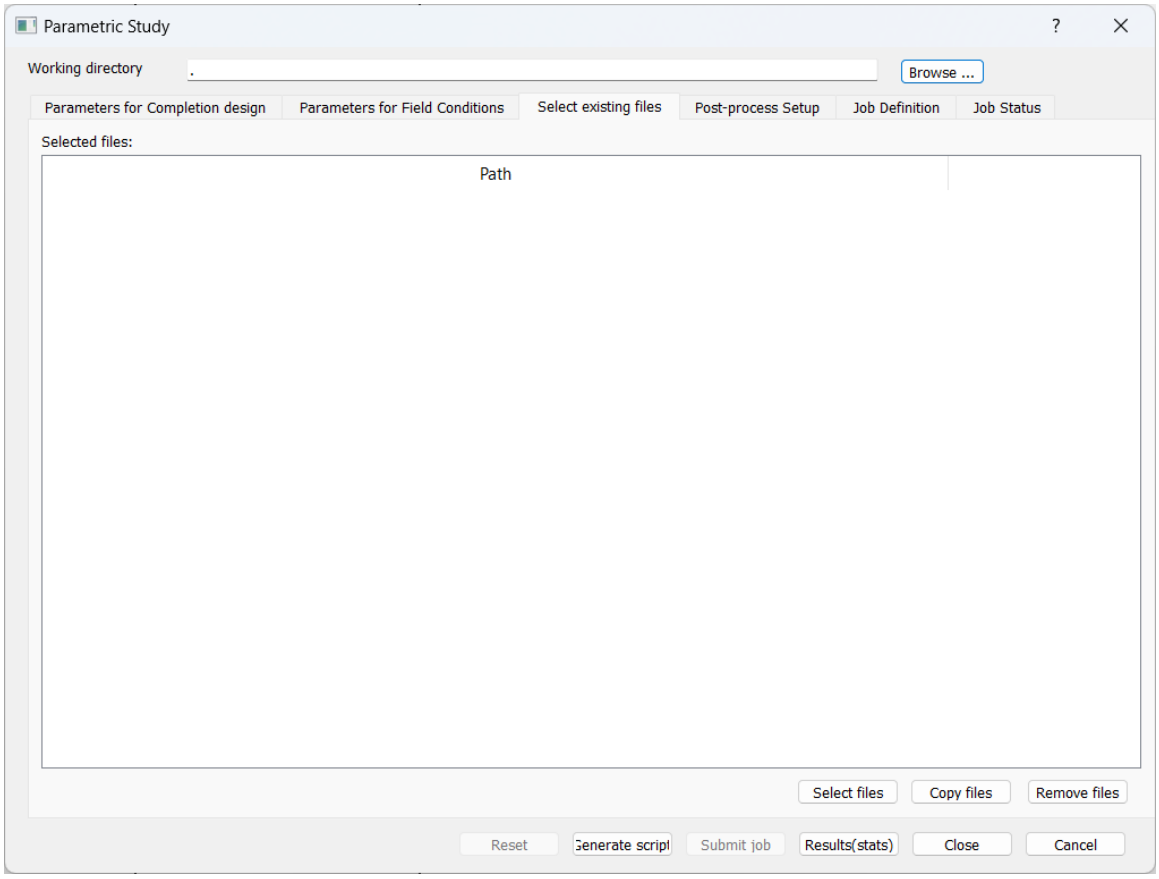


Figure 28 Select existing files tab.

Post-process Setup

All simulated models after their completion will be automatically post-processed. Details of information included in post-processing are specified in the **Post-process Setup** tab shown in Figure 29. Meanings of the fields in this tab are described in the Post-process tab section.

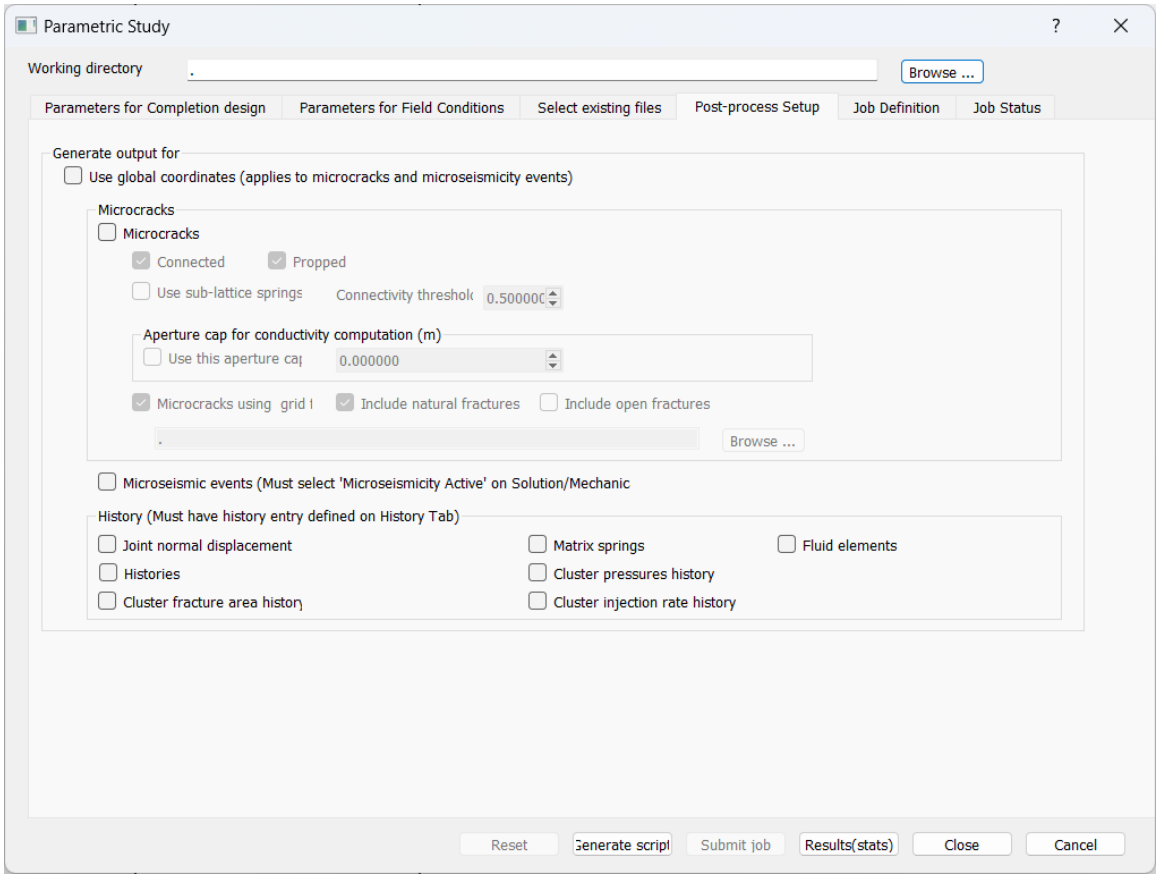


Figure 29 Post-process Setup tab.

Job Definition

The guidelines were developed for HPC centers that have the Sun Grid Engine installed. In this case, a script to execute a simulation, either a single node or an MPI, can be created and executed by XSite. For other HPC installations, the script and execution need to be created based on the HPC center installed software and guidelines.

For each file submitted for execution, a log file is created. The output displayed in the batch-results window is written to a file that uses the same name as configuration files, but with ".out" replacing the ".xml" extension. The user can check the progress of runs by looking at the contents of the file. This file can also be loaded in the "Log" window of the **Job Status** tab (Figure 31).

For each file submitted for execution, SGE writes the output of the console program (standard out and standard error). The file name starts with a value defined in the **Job Name** field followed by a "." and SGE job number. The user can check the run progress by looking at the contents of the file.

In the **Job definition** tab (Figure 30) the user will be able to set the SGE execution parameters. The fields are described below.

1. **Working directory** is the directory where files produced by XSite for the specific run are stored.
2. **Project Name** indicates how XSite jobs are identified by the HPC center, and it must always be specified.
3. **Job Name** must be a unique name that identifies a specific parametric study that will be submitted for execution. If the user submits a job with a name that is currently running in the clusters, SGE will refuse to schedule the new submission. Depending on the SGE implementation, the job name is part of the log file name.
4. **Cluster Queue** defines the queue that the job will be assigned to, and it must have the value **default** assigned to it.
5. **Parallel Environment** defines which parallel environment (e.g., MPI, shared memory) will be used to execute this job.
6. **Status refresh interval** defines how frequently SGE will be polled to inform about the job status. This information is displayed in the **Job Status** tab (Figure 31). The user should use the default value of 60 seconds (or greater) to avoid taxing the system unnecessarily.
7. **Maximum run-time** is used to help SGE better utilize resources. The user should try to make an educated guess of how long the simulation should run. In some HPC Centers, the job will be canceled after the specified maximum run-time has elapsed.
8. **Input base file name** is the base name of the configuration (.xml) files generated for the parametric study.
9. **File connector symbol** is part of the base name of the configuration (.xml) file generated for the parametric study.
10. **Number of threads (nt)** is the number of threads per process. This value depends on the hardware being utilized and it may be recommended by the HPC center guidelines.
11. **Creating scripts using** defines what files will be used in the script.
    If **Field condition files** is selected, files for the simulation will be created using the parameters specified on the **Parameters for Field Conditions** tab. Each file will have a separate script.
    If **Existing files** is selected, the script will utilize the files specified in the **Select existing files** tab. If several files were selected, a single node script will be created per file. MPI script only supports one selected file.
12. **Use MPI** must be selected if the user intends to split the simulation in separated nodes. When this checkbox is selected, the group box **MPI Parameters** is enabled. The model will be split in an orthogonal grid according to the specified XYZ entries. The **Number of nodes (np)** and the **Number of processes per node (ppn)** must be specified. Note that the product of **np** times **ppn** must be equal to the product of the **Domain Partitions**, XYZ. The number of threads must be adjusted to accommodate the number of processes per node. For example, a domain X=4 Y=5 and Z=2 will require a total of 40 processes. If the nodes of the specified queue support 36 threads, this domain may be executed specifying either:
    A) np = 40, ppn = 1 and nt = 36 or
    B) np = 20, ppn = 2 and nt = 18
13. **Simulation behavior** should always be set to non-deterministic.

The parametric study file names are composed of **Input base file name** followed by **File connector symbol** and a sequential number from 1 to the maximum number of realizations.

Once setup is completed, click on the **Generate scripts** button at the bottom of the dialog. Wait for a confirmation message box. At this point, all files are generated and ready to submit for execution. The script will be displayed on the edit box just above the **Generate scripts** button. Click on the **Submit job** button and wait for the confirmation message box. The job is submitted.

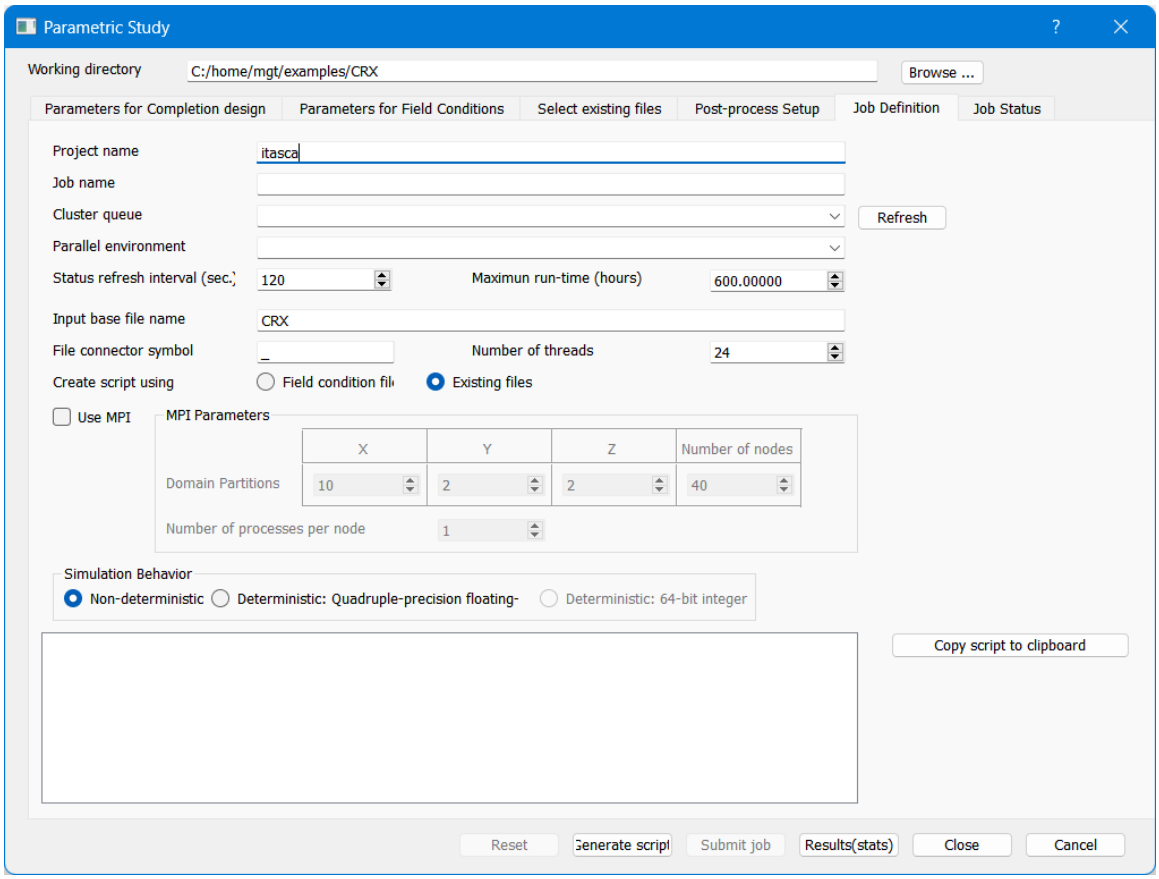


Figure 30 Job Definition tab.

Job Status

The **Job Status** tab opens the window shown in Figure 31. When the **Status** window is empty, all processes for the submitted jobs are finished. This screen is refreshed according to the value entered in field **Status refresh interval**.

Also, the user can check the status of specific simulations by selecting a file in the **Select a log file to show** combo box and pressing the **Show log file** button. The simulation status will be displayed in the **Log** window.

In the case the user closes an XSite session and wants to verify the status of a run, the user should first set the working directory to the directory where the parametric study was created and press **Load log files**. Then, use **Select a log file to show** to select the file to be checked.

To terminate a job, enter the **Job number** in **Terminate job** and press **Terminate**. This procedure may take a minute to execute.

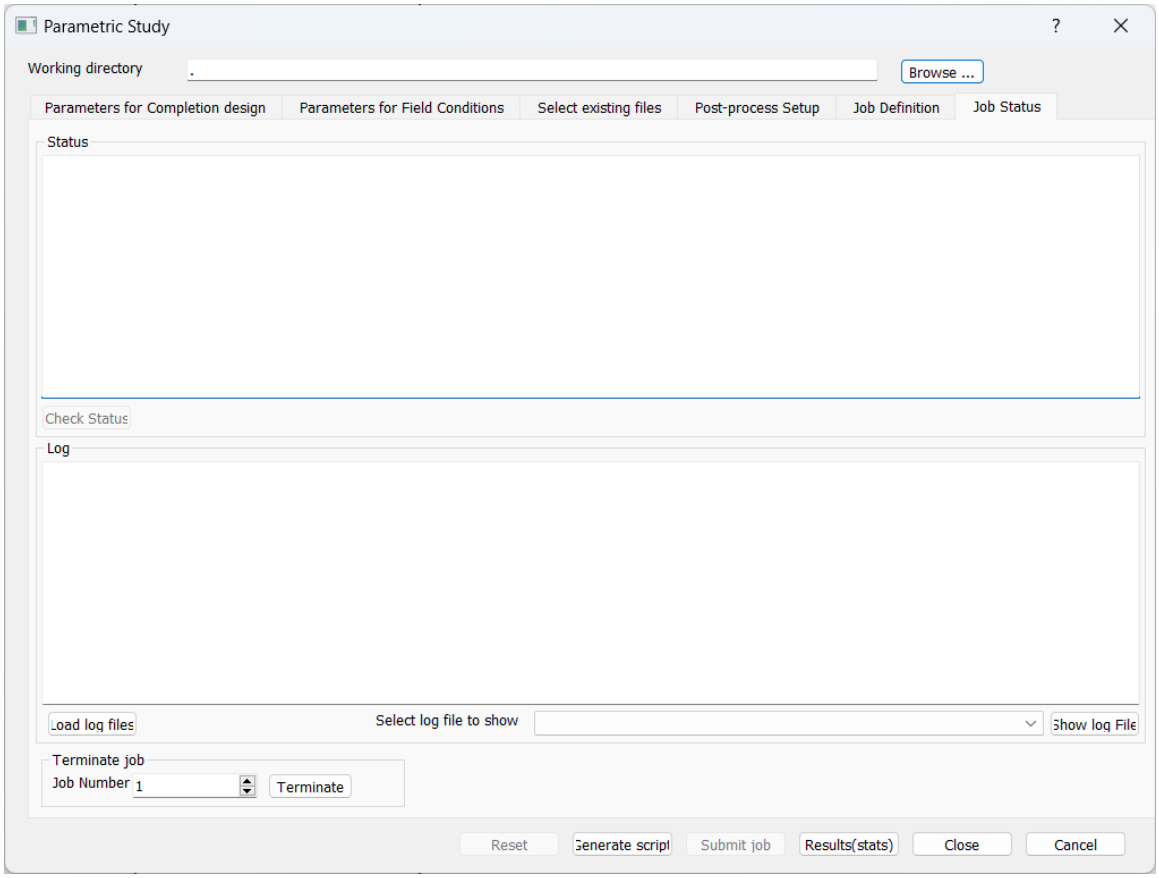


Figure 31 Job Status tab.

Resources

Resources contain data that are not necessarily used in the model but are available to be used in the models. The new information can be added to the resources during model creation. That information is available to be reused in future models.

Conductivity Curves

The conductivity curves for packed proppant provide dependence of the proppant permeability to proppant concentration and proppant stress due to fracture closure on the proppant. The proppant curves are used in the simulations to calculate permeability of the propped fractures. The proppant curves are defined for a model under **Features/Proppant/Permeability Calculation/Conductivity function of stress and concentration**. All curves available in the resources will appear in the drop-down menu, under **Curve name** and can be selected in the model. The example of the conductivity curves for 40-70 Ottawa Sand is shown in Figure 32.

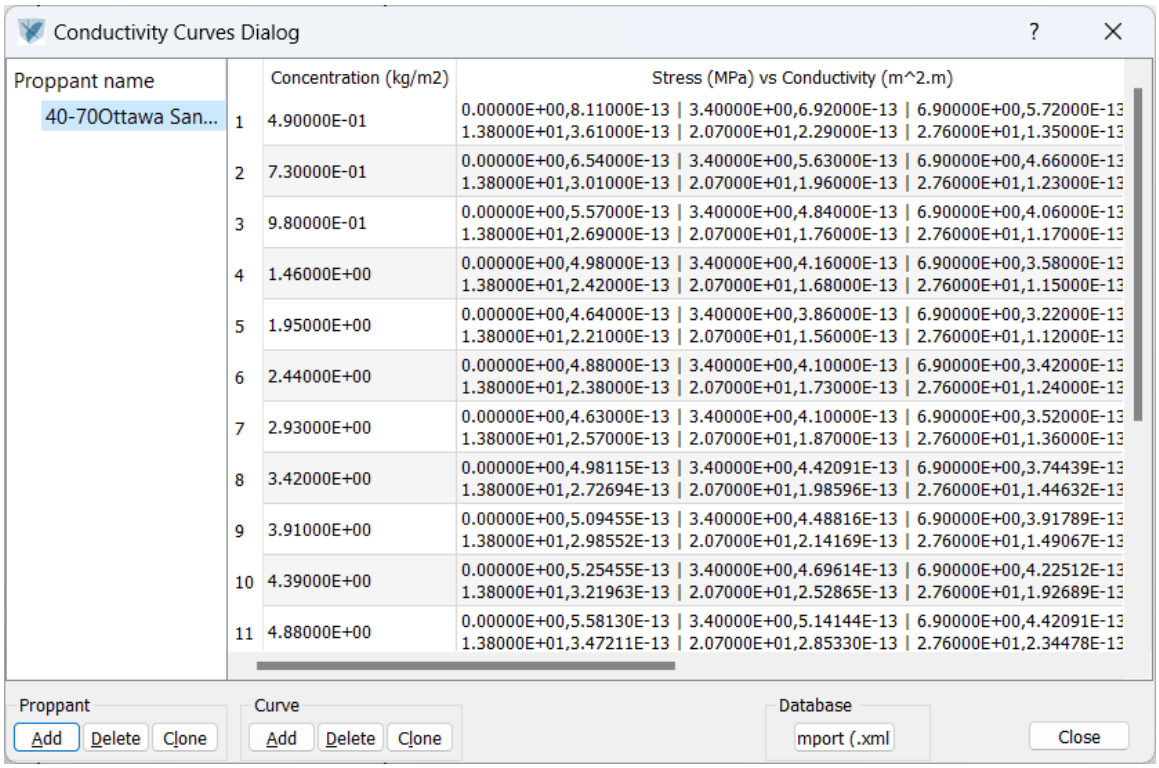


Figure 32 Conductivity Curves Dialog.

New curves can be imported from the Excel spreadsheet (i.e., multiple curves for different concentrations) by selecting a file under **Database** or manually entered by clicking the **Add** button under **Proppant** or **Curve**. **Curve** is used to add new curves for existing proppant. **Proppant** is used to add new proppant to the table. The dialog used for manual entry of the proppant conductivity curves is shown in Figure 33. The same dialog will appear if any of the entries in the **Conductivity Curves**

dialog (Figure 32) are double-clicked. Each curve is defined for a specific concentration as a function of the stress. The dialog allows for manual entry, importing a data file in .csv format and pasting columns copied from an Excel spreadsheet.

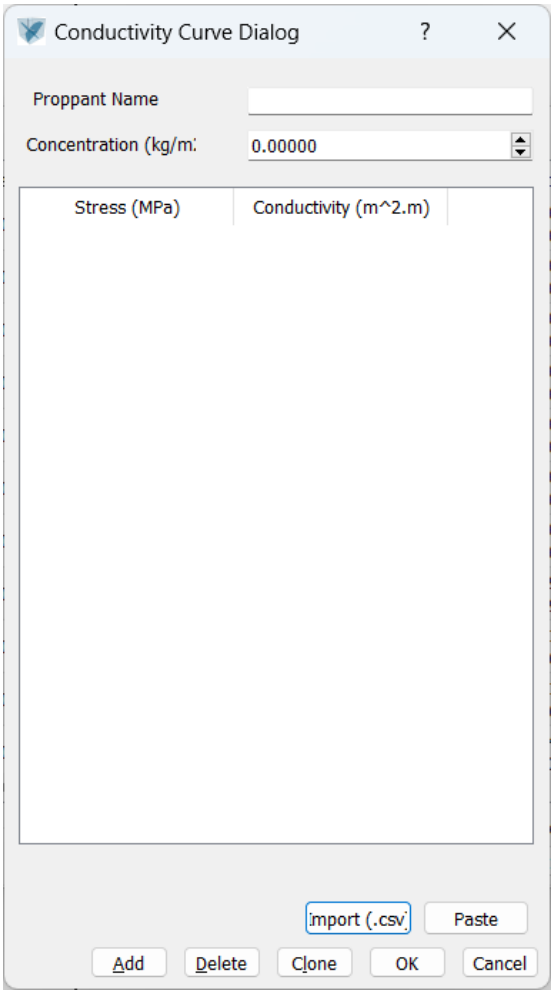


Figure 33 Dialog to add entries to conductivity curves.

Cluster Design

A list of available perforation cluster designs is in the **Cluster Design Items Dialog** shown in Figure 34. The cluster designs are used in **Resources/Stage Design**. New cluster design can be added by importing or manually by clicking the **Add** button, which will open the **Cluster Design Dialog** shown in Figure 35. The same dialog that will be shown by double-clicking **Cluster Design Items** will allow editing of the existing cluster designs. **Cluster Design** items can be associated with **Stages**. A **Cluster Design** item may be used multiple times in the same model by different **Stages** within different **Well Trajectory** items. The **Cluster Design Items** dialog presents a list of all **Cluster Design** records that are available for use in the model. Selecting the **Import** option will allow the user to read **Cluster Design** records from another XSite configuration file.

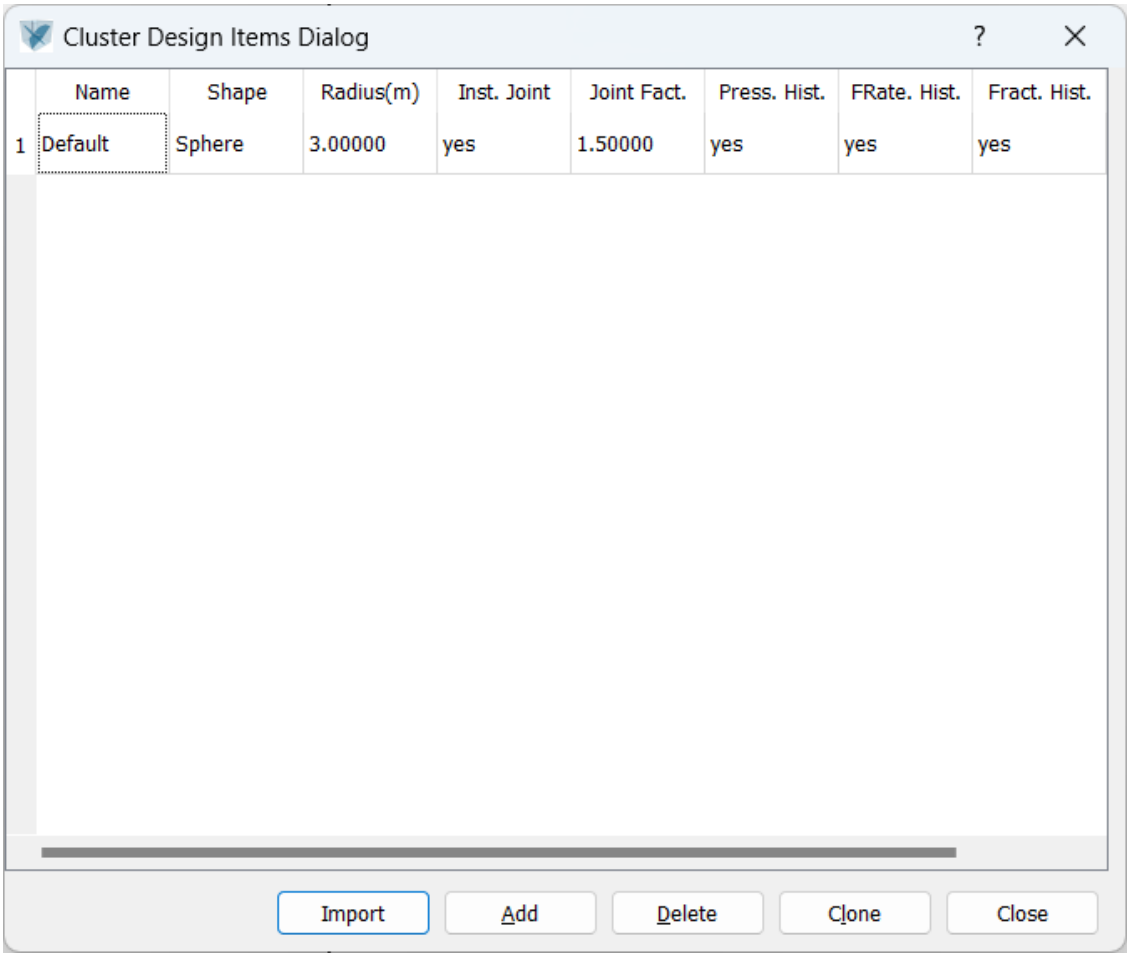


Figure 34 Cluster Design Items Dialog.

The cluster designs are identified by unique names. The clusters do not include explicit representation of the perforations. They are either spherical or cylindrical volumes (defined by **Radius** and **Cylinder length**) along wells through which fluid is injected into a reservoir. Although the perforations are not explicitly represented in the perforation cluster (in reservoir scale models), the perforation pressure drop can be accounted for using the empirical relation provided by Crump and Conway (1988). The input parameters for the perforation pressure drop calculation are **Number of Perforations**, **Perforation Diameter**, **Discharge Coefficient** and **Perforation factor**.

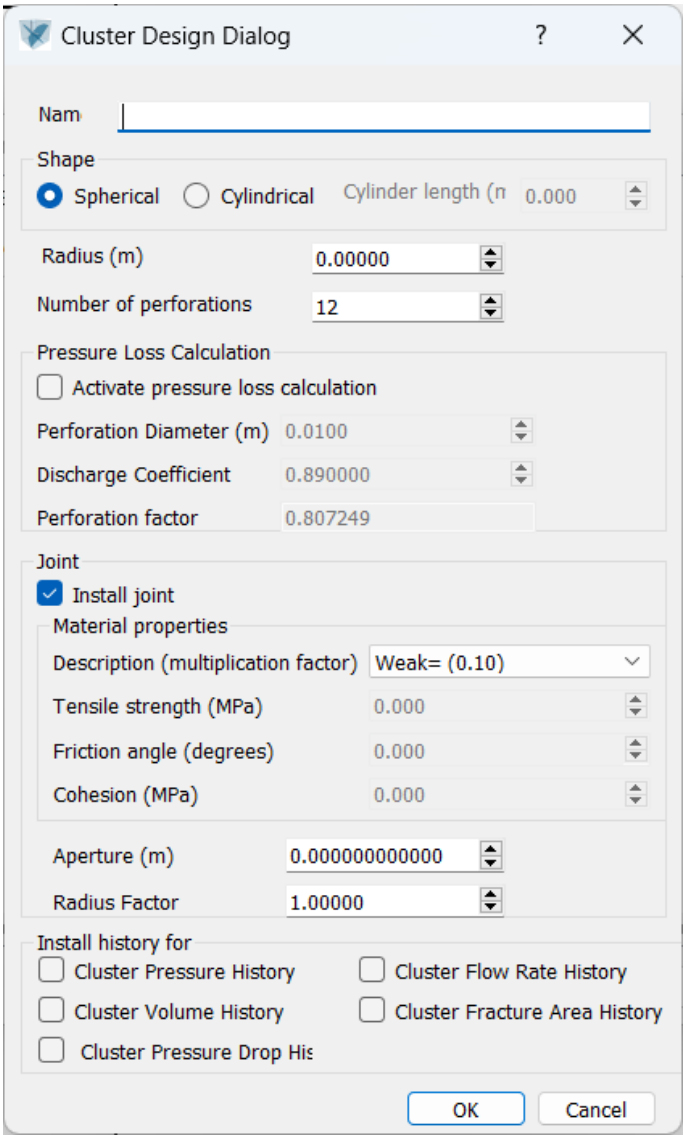


Figure 35 Cluster Design Dialog.

In the reservoir-scale models, the fracture initiation from the perforations is not simulated. Instead, it is assumed that the fractures are initiated perpendicular to the minimum principal stress. The fracture propagation models are typically set up with start-up circular fractures created though the center of the perforation cluster perpendicular to the minimum principal stress. The start-up joints can be defined manually using the **Features/Joints** tab. In that case, the user needs to specify the coordinates and orientation (unit normal) of potentially multiple start-up joints. Use of the cluster design will automatically generate the start-up joints with the proper orientation (perpendicular to the in-situ minimum principal stress). The joint radius is defined by the **Radius factor**, which is a multiplier of the cluster radius. The factor must be greater than one, and the joint diameter should be at least three times the lattice resolution at that location. The joint **Aperture** (typically ~10^{-4} m) should be relatively small to the apertures expected during fracture propagation. Also, the mechanical properties of the start-up

joint need to be specified. By default, it is assumed that the joint is frictional, with a specified friction coefficient.

The cluster setup also allows automatic definition of the cluster-related histories, like cluster pressure, area and volume of created fractures connected to the cluster, flow rate and pressure drop.

Fluid

Fluid types available for use in models are listed in the **Edit Fluid Items Dialog** shown in Figure 36. When selecting the fluid type used in a model (e.g., **Features/Fluid/Fluid Type**), the available fluid types can be selected from the drop-down menu. New fluid types can be created by adding them from scratch (**Add** button) or editing either an existing fluid type or cloned type.

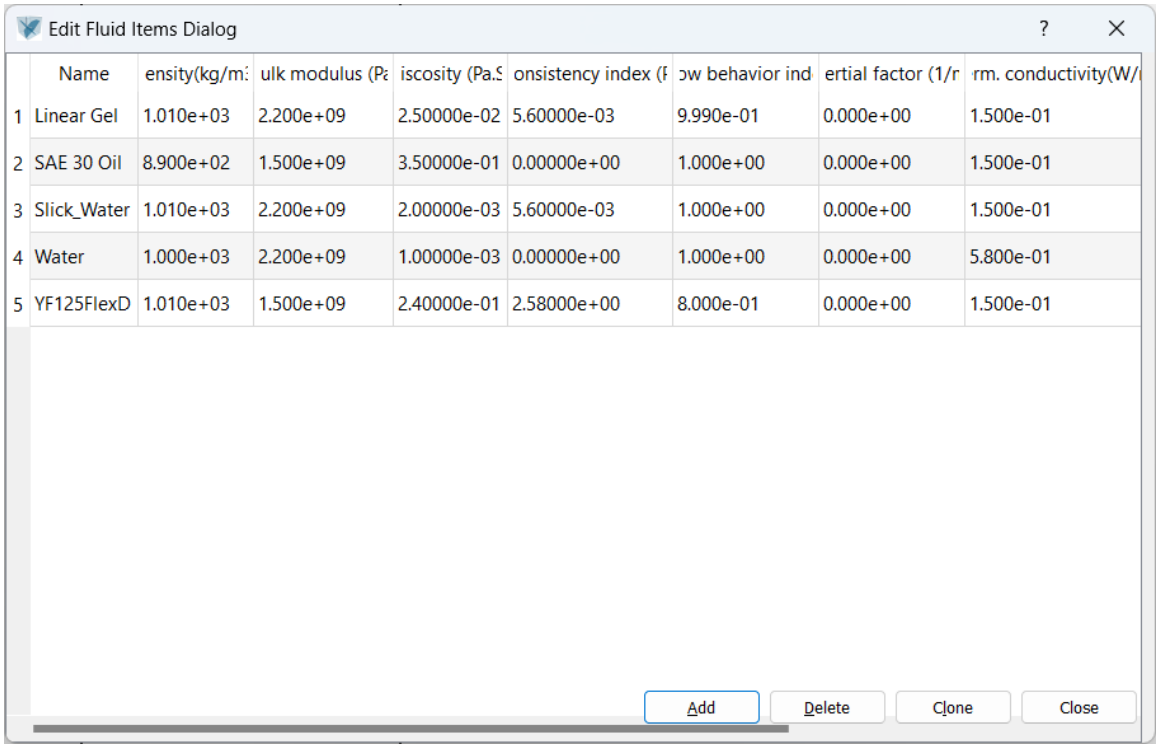


Figure 36 List of fluid types.

Double-clicking an entry in the **Edit Fluid Items Dialog** or clicking the **Add** button will open the **Fluid Properties** dialog shown in Figure 37. If an existing type is opened, the properties table will be populated by non-zero properties values. Each fluid type must have a unique name specified in the **Fluid name** field.

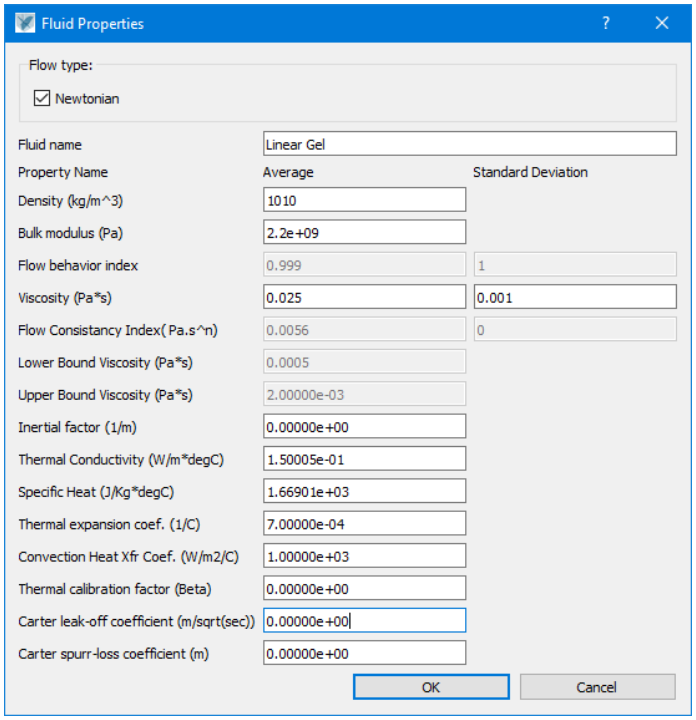


Figure 37 Fluid Properties dialog.

Materials

Rock material types available for use in models are listed in the **Edit Materials Dialog** shown in Figure 38. When selecting the rock material type used in a model (e.g., **Main Rock/Material**), the available rock material types can be selected from the drop-down menu. New rock material types can be created by adding them from scratch (**Add** button) or editing either an existing material type or cloned type.

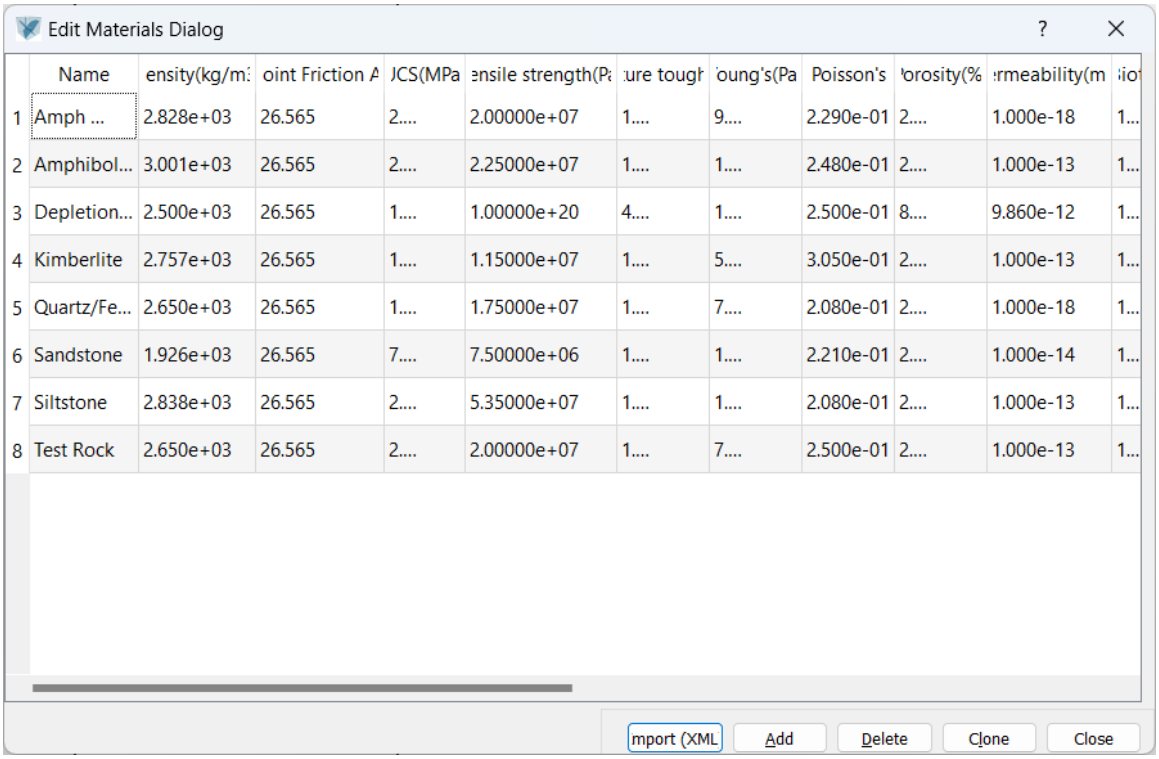


Figure 38 List of rock types.

Double-clicking an entry in the **Edit Materials Dialog** or clicking the **Add** button will open the **Materials Properties** dialog shown in Figure 39. If an existing type is opened, the properties table will be populated by non-zero properties values. Each rock material type must have a unique name specified if the **Material name** field.

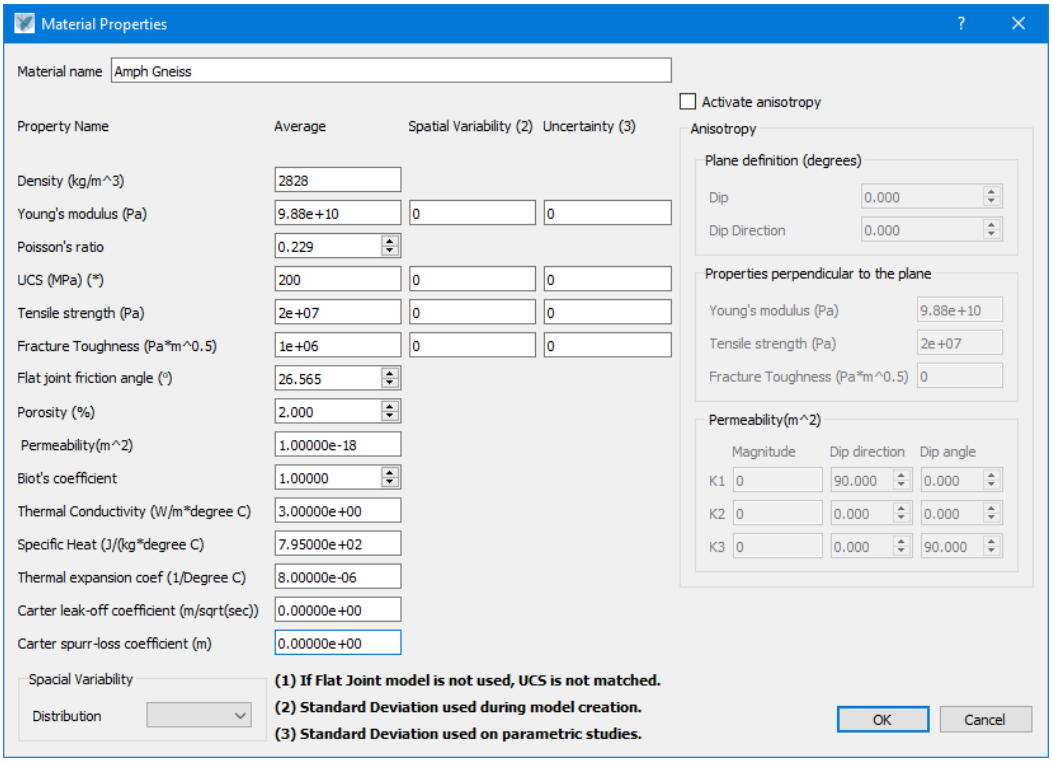


Figure 39 Rock Properties dialog.

Pumping Schedule

The pumping schedules available as a resource for model building are provided in the **Pumping Schedule Items** list shown in Figure 40. Those pumping schedules can be used as a component of simulation sequences for multistage modeling. A **Pumping Schedule** item may be used multiple times in the same model by different boreholes or for different stages on a borehole. The pumping schedule includes histories of pumping rate, proppant mass, fluid type, proppant type and proppant size.

The pumping schedules can be created in the resources by importing them from ASCII and .xml files or manually entering them. Clicking on **Import (text)** will open the **Pumping Schedule Import Text Dialog** shown in Figure 41. Each pumping schedule should have a unique name. This dialog provides flexibility to import the pumping schedule data, which can include different information appearing in columns in an arbitrary order. The tabs at the bottom of the dialog should be dragged into the window in the same order they appear in the data as provided in a table. If there are columns in a table that are not needed in the pumping schedule, they should be marked by the **Skip** tab. The import utility will do unit conversion if needed, with the conversion selected for different columns at the bottom of the dialog.

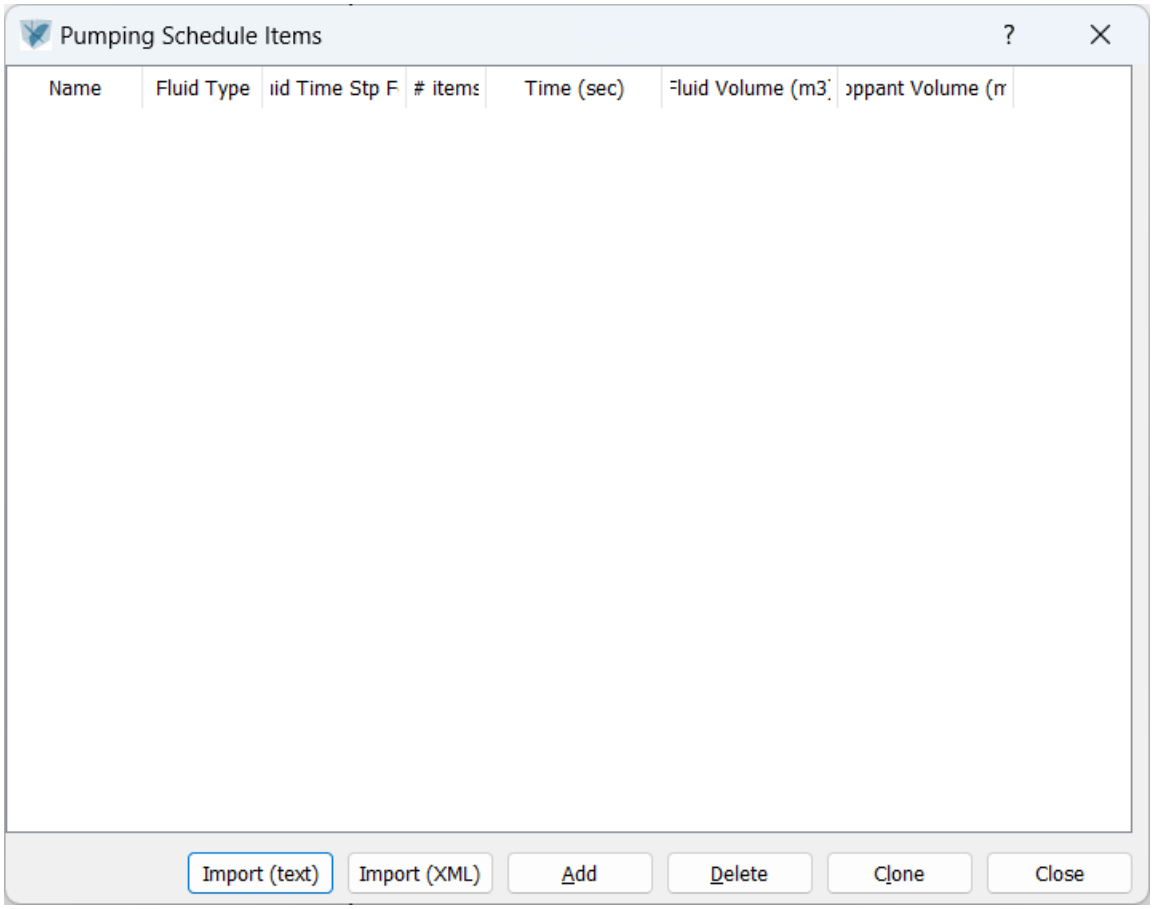


Figure 40 Pumping Schedule Items dialog.

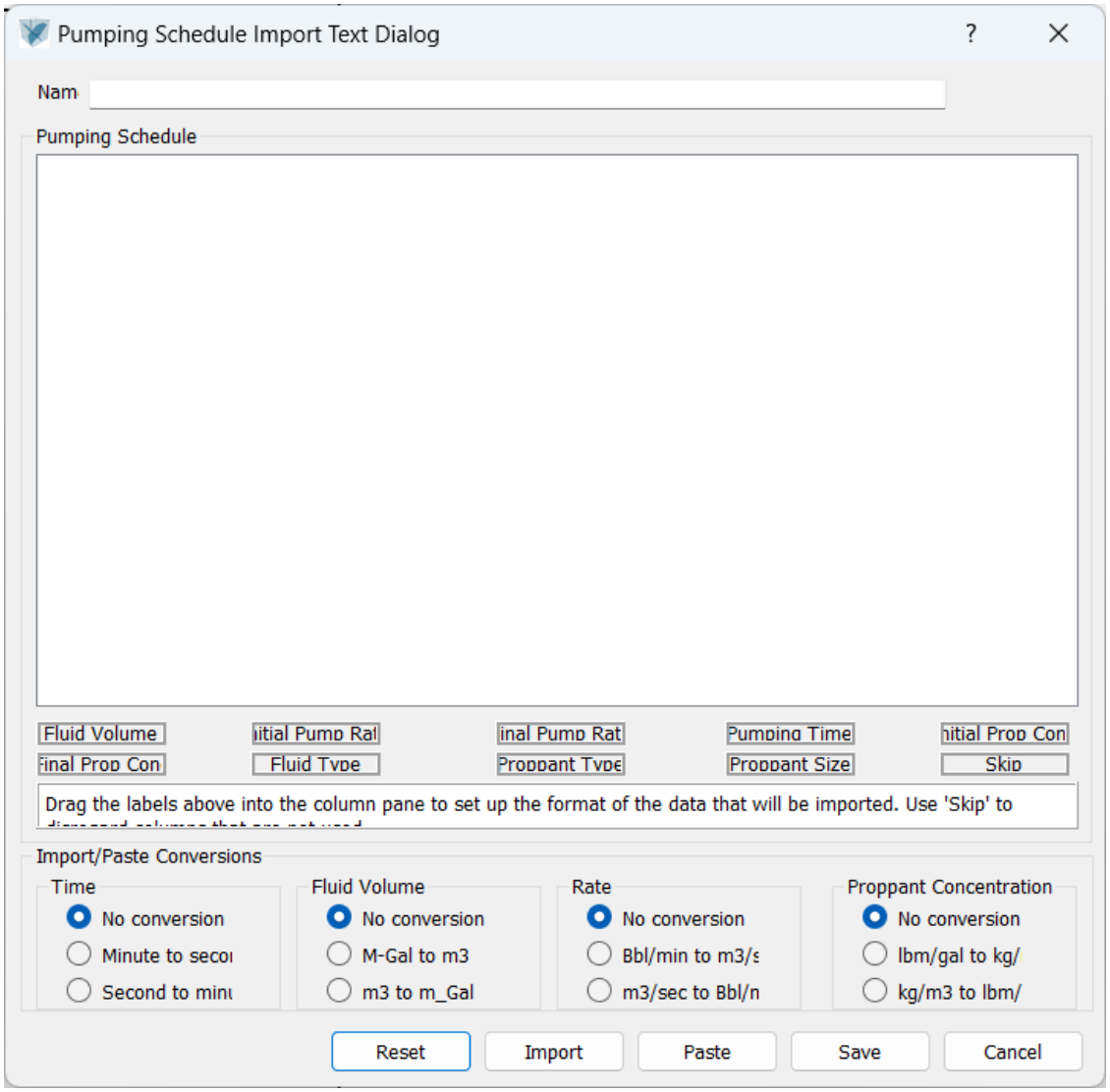


Figure 41 Pumping Schedule Import Dialog.

Figure 42 below shows an example pumping schedule that can be imported into XSite. Pressing the **Import (XML)** button in the **Pumping Schedule Items** dialog displayed in Figure 39 will allow the user to copy **Pumping Schedule** records from another XSite configuration file.

Slurry Volume (M-Gal)	Fluid Volume (M-Gal)	Proppant Conc (PPG) Start	Proppant Conc (PPG) End	Rate (BPM)	Fines Conc. (Vol Fraction)	Proppant (M-Lbs)	Pump Time (min)	Cum Time (min)	Fluid Type	Proppant Type
50.000	50.000	0.00	0.00	60.00	0.000000	0.0	19.841	19.8	Linear_HPG,_30#,_	Ottawa Snd 20-40
10.453	10.000	1.00	1.00	60.00	0.000000	10.0	4.148	24.0	Linear_HPG,_30#,_	Ottawa Snd 20-40
6.544	6.000	2.00	2.00	60.00	0.000000	12.0	2.597	26.6	Linear_HPG,_30#,_	Ottawa Snd 20-40
6.815	6.000	3.00	3.00	60.00	0.000000	18.0	2.705	29.3	Linear_HPG,_30#,_	Ottawa Snd 20-40
7.087	6.000	4.00	4.00	60.00	0.000000	24.0	2.812	32.1	Linear_HPG,_30#,_	Ottawa Snd 20-40
7.359	6.000	5.00	5.00	60.00	0.000000	30.0	2.920	35.0	Linear_HPG,_30#,_	Ottawa Snd 20-40
7.631	6.000	6.00	6.00	60.00	0.000000	36.0	3.028	38.1	Linear_HPG,_30#,_	Ottawa Snd 20-40
7.903	6.000	7.00	7.00	60.00	0.000000	42.0	3.136	41.2	Linear_HPG,_30#,_	Ottawa Snd 20-40
8.174	6.000	8.00	8.00	60.00	0.000000	48.0	3.244	44.4	Linear_HPG,_30#,_	Ottawa Snd 20-40

Figure 42 Example of pumping schedule data.

If the **Add** button is clicked (or an item in the list is double-clicked) in the **Pumping Schedule Items Dialog**, the **Pumping Schedule Dialog** shown in Figure 43 will be opened. This dialog allows manual definition of the pumping schedule, discretized in time increments within which the variables are changing linearly. Definition of a pumping schedule includes the unique schedule **Name** and **Fluid** type used. It is required to enter a unique name for the Pumping Schedule record.

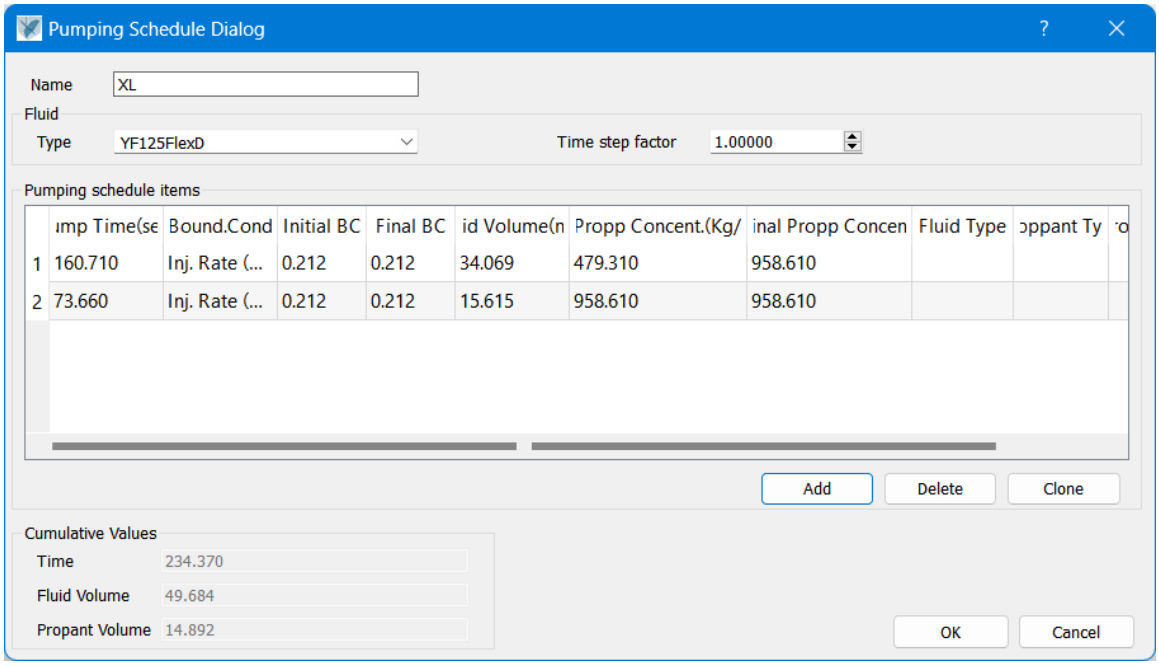


Figure 43 List of manual Pumping Schedule entries dialog.

Clicking the **Add** tab in this window opens the **Pumping Schedule dialog** shown in Figure 44. The pumping schedule controls rate or pressures during injection. The rate (pressure) history along with proppant concentration can be provided in a discretized manner. Fluid type, proppant type and size can change from one time increment to another.

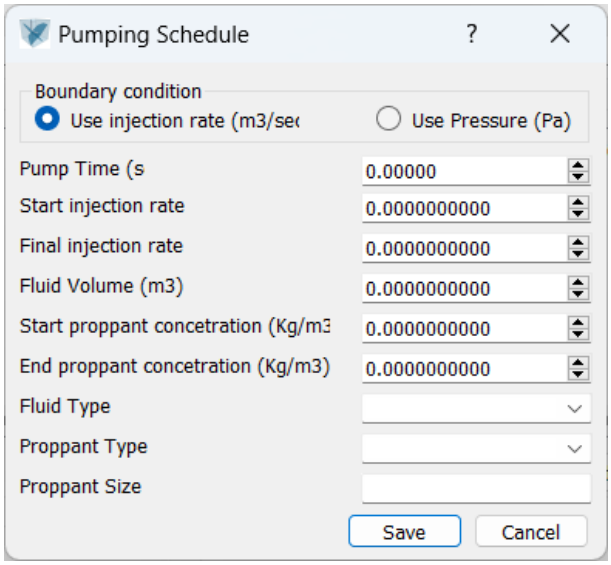


Figure 44 Manual Pumping Schedule entry dialog.

Simulation Sequence Design

Simulation of each stage of multistage stimulation potentially includes multiple simulation sub-steps, including definition of pumping schedules for stages (which can be the same but also vary between stages), application of adaptive resolution (i.e., recreating lattice with the finest resolution around the currently active stage), mechanical equilibration of the new lattice, mechanical equilibration of the lattice after completion of pumping and potentially resetting the fluid pressures to approximate the effect of flowback and leak-off. Definition of these steps requires multiple steps (e.g., **Batch Simulation** entries and **Injection Rate Schedules**) in the model setup that need to be consistent. Repeating all those steps for multiple stages could be a tedious and error-prone task. The **Simulation Sequence** available in **Hydraulic Fracturing/Boreholes** makes the process of multistage stimulation model setup easier, automatic and less error prone. Based on the provided information, the code will automatically generate pumping schedules and batch simulation entries. Different simulation sequences can be stored in resources to be used in future models. The list of simulation sequence items available in resources can be viewed in Figure 45.

The **Simulation Sequence Resource Dialog** (Figure 46) is opened by clicking the **Add** button in the **Simulation Sequence Resource Items Dialog**.

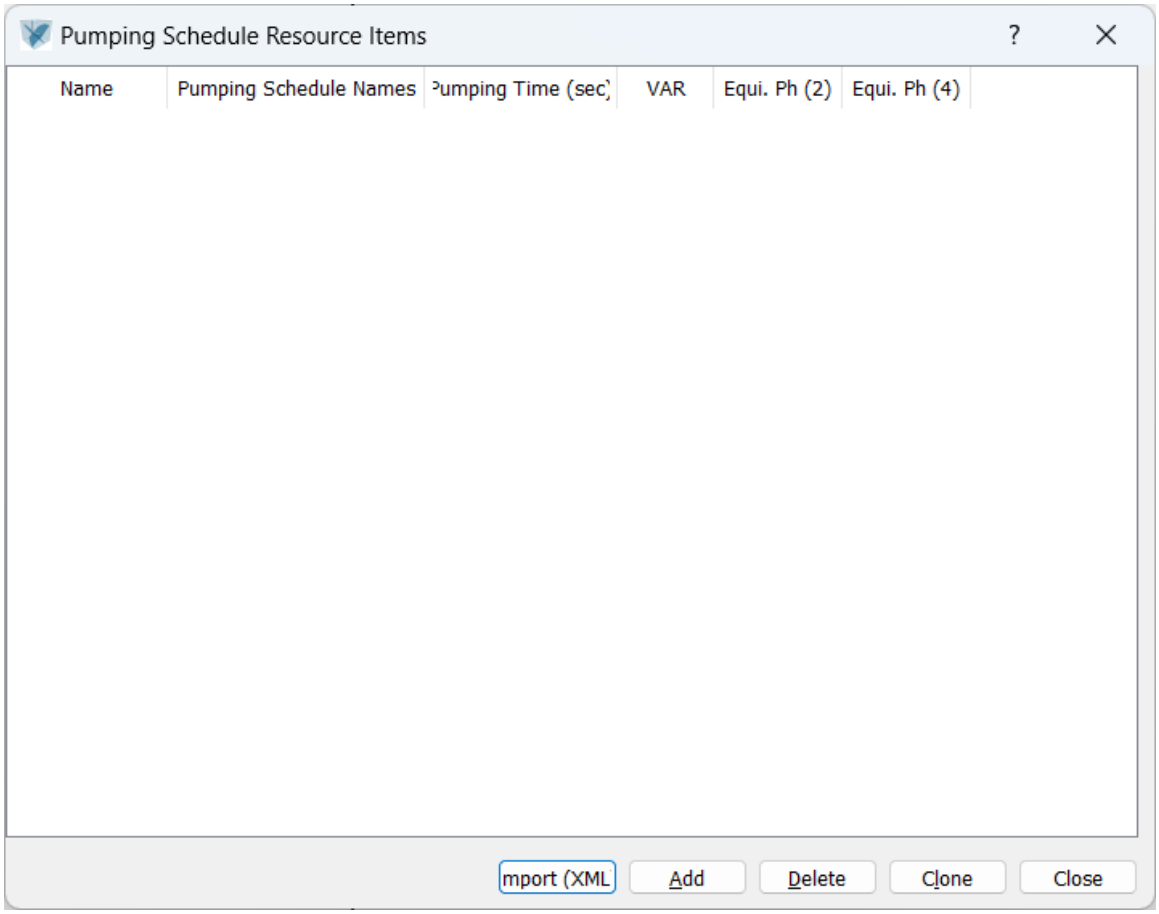


Figure 45 Simulation Sequence Resource Items.

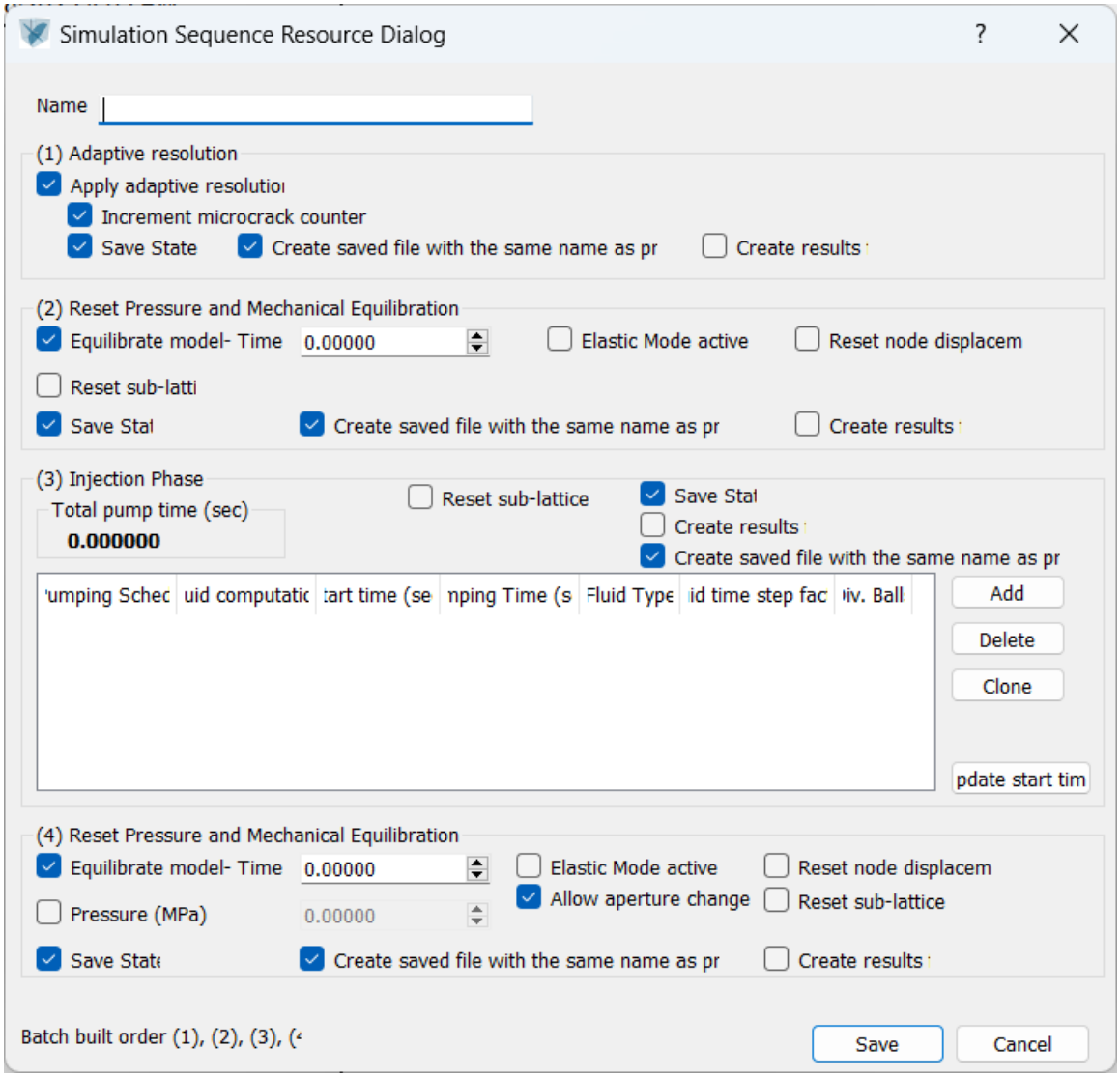


Figure 46 Simulation Sequence Resource Dialog.

Stage Design

The list of stage designs available in resources is shown in the **Stage Design Items Dialog** (Figure 47). The new stage designs can be added manually by clicking the **Add** button, which opens the dialog shown in Figure 48.

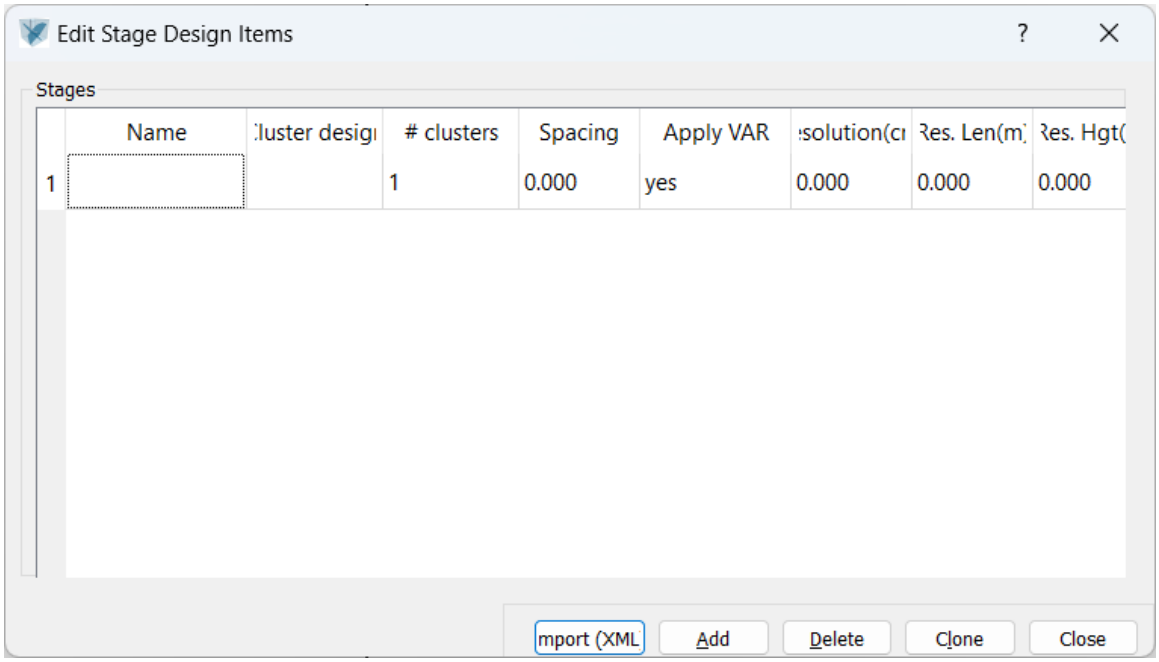


Figure 47 Dialog showing a list of Stage Design Items.

Each stage design has a unique **Name**. The existing cluster designs are selected with a definition of the **number of clusters** and **spacing between clusters**. The dialog can automatically generate the resolution domains for the stage. The height and length of the finest resolution domain must be specified by the user. The length can be automatically determined based on the stage geometry (number and spacing of the clusters) or specified by the user. Also, the orientation of the resolution domains can be determined automatically (aligned with the orientation of the minimum horizontal principal stresses or the borehole), as specified by the angle relative to the north in the clockwise direction or defined by the user. Gradual transition of the resolution can be achieved by using the multiple resolution domains by checking **Create layers** and specifying the layering parameters (see the Resolution/Nest feature).

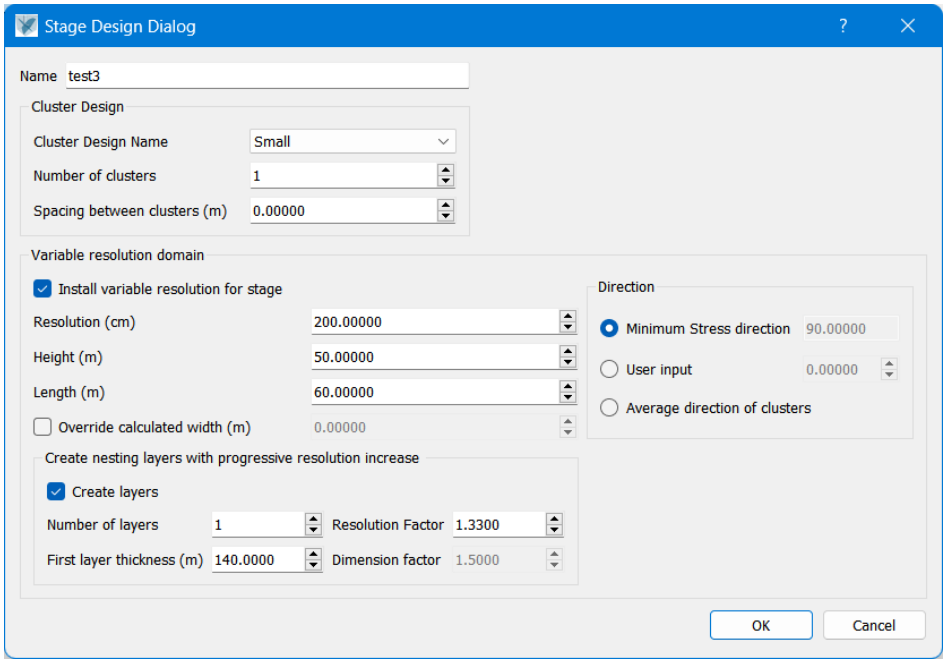


Figure 48 Stage Design Dialog.

Layout

The menu bar, tool bar and panels may be dragged to alternative positions or resized to form a custom GUI layout. The layout pull-down menu (Figure 49) allows custom GUI layouts to be saved and restored or reset to the default layout.

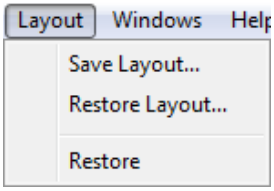


Figure 49 Layout drop-down menu.

Windows

The Windows pull-down menu (Figure 50) allows any of the three main panels to be switched on or off. When the setup process is complete, it is often convenient to eliminate the **Controls panel** in order to increase the size of the display window.

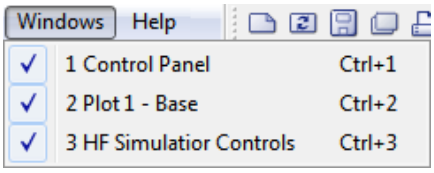


Figure 50 Windows drop-down menu showing panels and keyboard shortcuts.

Help

The **Help** drop-down menu (Figure 51) provides information about XSite (e.g., version and build numbers), ITASCA and Qt and Voro++ libraries in use (e.g., version number). Please note your software version number prior to contacting ITASCA for code support.

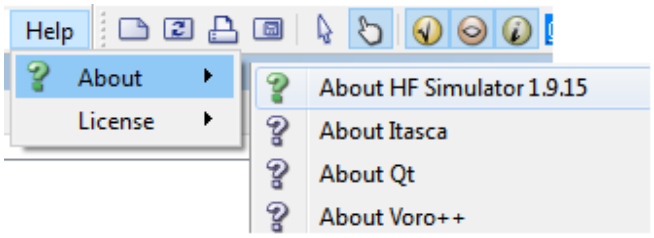


Figure 51 Help drop-down menu with the About item selected.

The **License** option allows one to specify the type of license that XSite will use by selecting the sub-menu **Specify License Location…**. See Figure 52 if the option is selected. Note that the **Help** button will invoke the help for this dialog.

The sub-menu option **Show License Diagnostic** will invoke a dialog box where you can run a procedure that will check the status of the licenses assigned to you. The diagnostics results are a valuable tool for the XSite team to diagnose if there are issues with the license system.

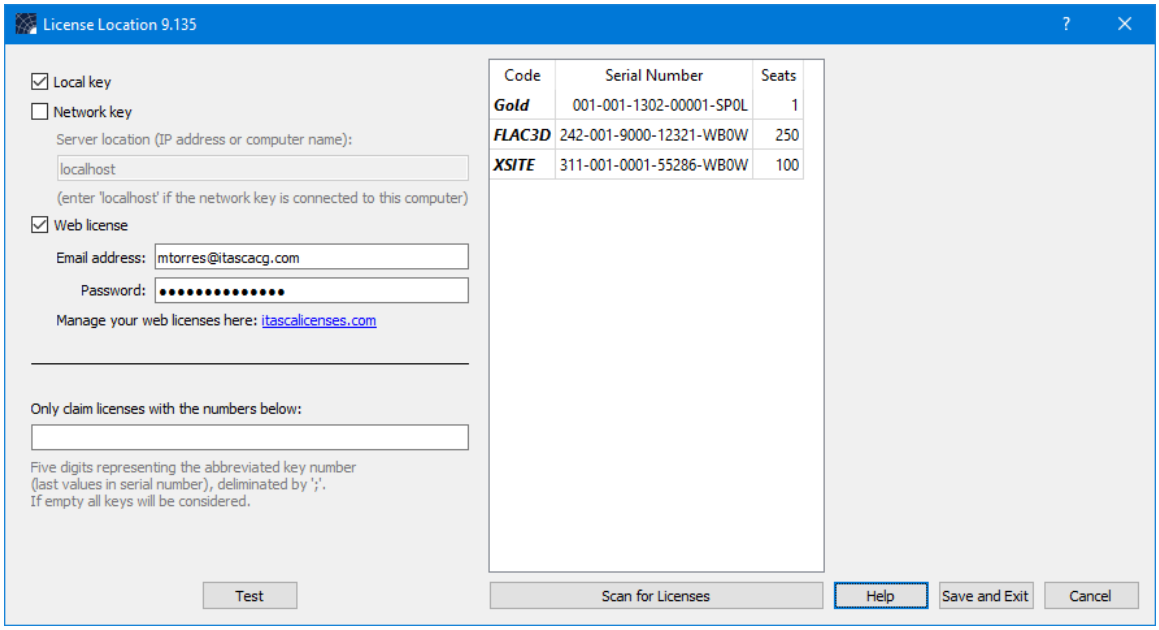


Figure 52 License Location dialog.

4.0 Toolbars

XSite has two toolbars (Figure 53): one for model manipulation and the other for plot item and plot view manipulation.

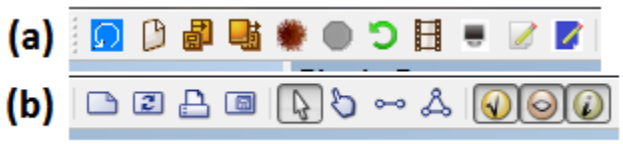


Figure 53 Toolbars (a) for model manipulation and (b) plot item and plot view manipulation.

Model Manipulation Toolbar

	**Reset model**	Clicking this icon causes the current model to be reset to the pre-simulation state, i.e., before generation of the lattice has occurred. Model data, such as dimension, joints, seams, options, etc., are retained. This allows the user to reset the model after a simulation is complete, modify some input parameters and restart a new simulation.
	**New dataset**	Clicking this icon causes all model information to be reset. All setup information is lost.
	**Load dataset**	This icon activates Load Dataset, as described in the File menu.
	**Save dataset**	This icon activates Save Dataset, as described in the File menu.
	**Start simulation**	This icon is equivalent to Start Simulation, as described in section File menu.
	**Stop simulation**	This icon changes from gray to red during an active simulation. Clicking on it when red causes the simulation to stop. If the simulation is running via batch, this action will stop the current batch step and start the next batch step.
	**Reset lattice**	This icon allows the user to reset lattice node displacements, the micro-crack counter, calibration velocity and other lattice variables

		after model initialization, as discussed in Reset Lattice Dialog. When clicked, the Reset Lattice window shown in Figure 54 will be displayed.
	**Record movie**	This icon toggles movie-capture on and off — the **on** state is indicated by a white background to the icon. Movie parameters (sample interval, frames per second, etc.) can be set from the **Tools** item in the menu bar and selecting **Options** — Movie Settings. Depending on the sampling frequency that the screenshots will be captured, this feature, when selected, may slow down the simulation because of the overhead of taking screenshots.
	**Capture screen**	Clicking this icon will create an image of the current plot in the current working directory.
	**Edit rock properties**	Clicking this icon causes the **Edit Materials Dialog** to be displayed. See menu option Resources/Materials for additional information.
	**Edit fluid properties**	Clicking this icon causes the **Edit Fluid Dialog** to be displayed. See menu option Resources/Fluid for additional information.

Reset Lattice Variables

When this toolbar option is selected, the dialog box displayed in Figure 54 is displayed. This dialog contains a list of tasks that can alter the state of an initialized model. There are two tabs in this dialog: **Mechanical, Fluid and Thermal Model Settings** and **Import Data**.

The functionality of the **Mechanical, Fluid and Thermal Model Settings** tab is a subset of the fields found in the tab Pre-process (1) of the Batch Simulation dialog. The task can be executed after the model is initialized. The tasks available in this tab will alter the state of the model. The selected task in this dialog will be executed when the user presses the **OK** button. When a task is selected, the corresponding value associated with the task is enabled, allowing the user to enter the new value.

The **Import Data** tab controls functionalities that manage parent-well model initialization in analyses of parent-child well interactions and the effects of reservoir depletion on hydraulic fracturing. See Appendix 1 – Workflow for Modeling of Effects of Reservoir Depletion (Parent-Child Wells) for details.

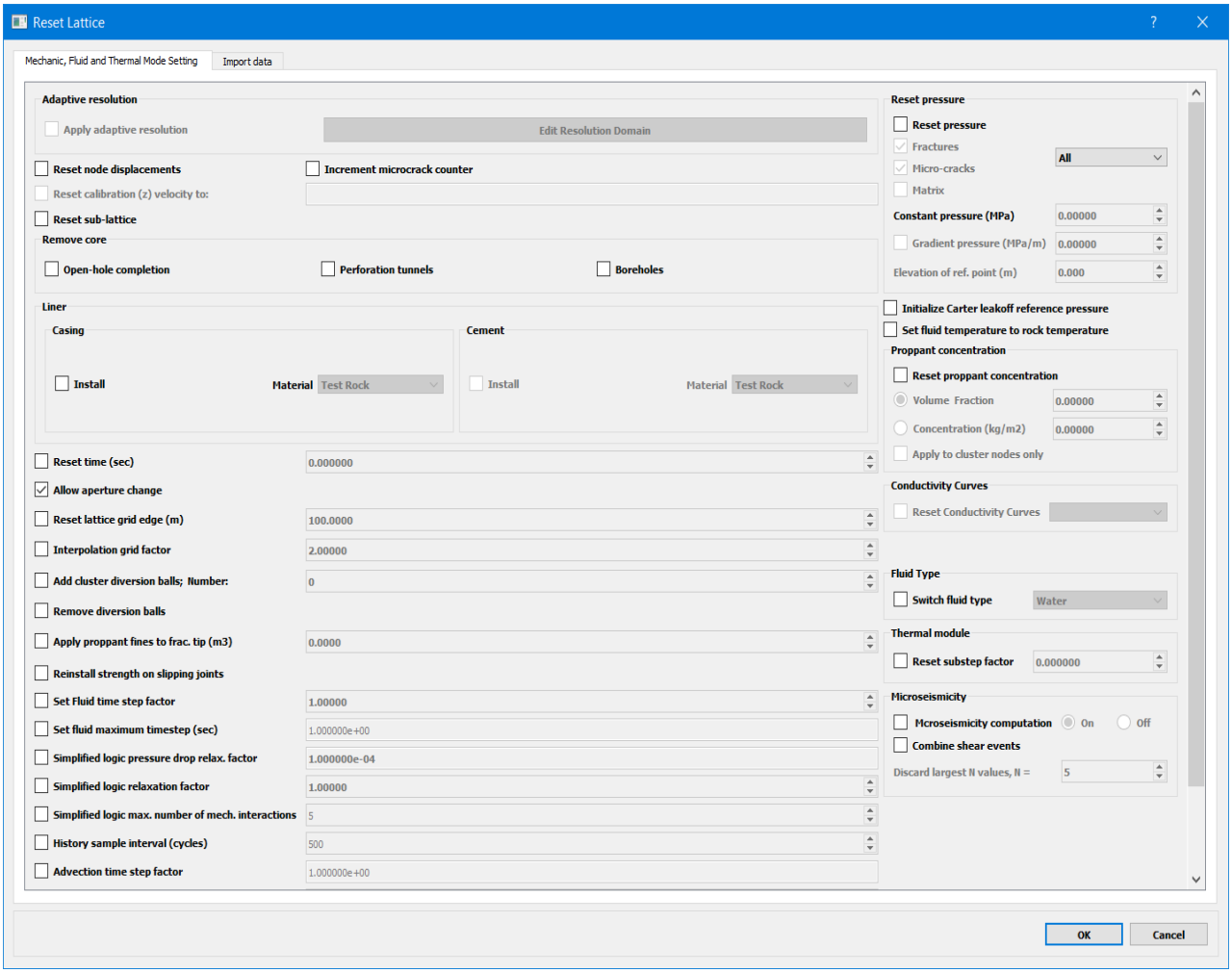


Figure 54 The Reset Lattice Dialog allows node displacements, the microcrack counter, vertical velocity and other lattice parameters to be reset after model initialization.

Plot View Toolbar

The following is the list of toolbar icons that can be used for manipulation of plots.

	**Create new plot view**	Clicking on this icon creates a new plot view. A plot view may contain several plot items. A plot view is activated by selecting one of the tabs found in the lower section of the **Plot View** panel.
	**Regenerate current plot view**	Clicking on this icon regenerates the current plot view, which is equivalent to pressing F5. During simulation the code regenerates all the plot items of the current plot view every certain number of cycles. This icon allows manual plot regeneration.
	**Print the current plot view**	Clicking on this icon prints the current plot view.
	**Export plot view as bitmap**	Clicking on this icon saves the current view as a bitmap file (*.bmp or *.png) that can be imported into other software (e.g., word processor, presentation, etc.).
	**View mode**	Puts the mouse in **View** mode. In this mode, interactive elements will not be affected.
	**Manipulate mode**	Puts the mouse in **Manipulate** mode, which is used to manipulate (resize, position, etc.) any interactive elements that appear in the plot (e.g., legend, distance measurements, cut planes, clip box, etc.)
	Distance measurement	Puts the mouse in **Distance measurement** mode. By clicking two points in the plot item, the distance between the two selected points will be displayed. This tool only works if this view is in **Manipulate mode** and the plot item under the mouse is opaque.
	**Definition plane**	Puts the mouse in **Definition plane** mode. By clicking on three points in the plot, the plane defined by the three selected points will be displayed. This tool only works if this view is in **Manipulate mode** and the plot item under the mouse is opaque.
	**Plot items**	Clicking on this icon opens and closes the **Plot Items** menu in the **Plot Control Panel**.
	**View**	Clicking on this icon opens and closes the **View** menu in the **Plot Control Panel**.
	**Information**	Clicking on this icon opens and closes the **Information** menu in the **Plot Control Panel**.

5.0 Plot Control Panel

As mentioned previously, the **Plot Control Panel** (Figure 55) consists of several panels: plot item list, plot item menu, view controls and view information, and an interactive mouse information display.

Plot Item List/Menu panel

The **Plot Item** list panel displays the list of plot items that have been added to the current plot view. All plot views, even empty ones, contain three permanent plot items: **Legend**, **Display Settings** (for the current view) and **Global Display Settings** (for all views). These are shown in blue in the list to indicate their special status. The visibility of a plot item can be toggled (i.e., shown/hidden) using the icon to its left. Plot items also may be deactivated/activated or deleted by right-clicking on the item and selecting the appropriate menu option.

The **Plot Items Menu** panel is comprised of two tabbed panes: the **List** tab and the **Attributes** tab. The items menu contains a tree that lists all available plot items. They are organized by category. Items are added to a plot by double-clicking on the selected item, after which the added item will appear in the **Plot Item** list. After the item is added, the **Plot Items** menu automatically shifts to display the **Attributes** tab for the added plot item. This tab contains all the available controls affecting display of the plot item. All attribute controls are summarized below in the **Plot Items** section.

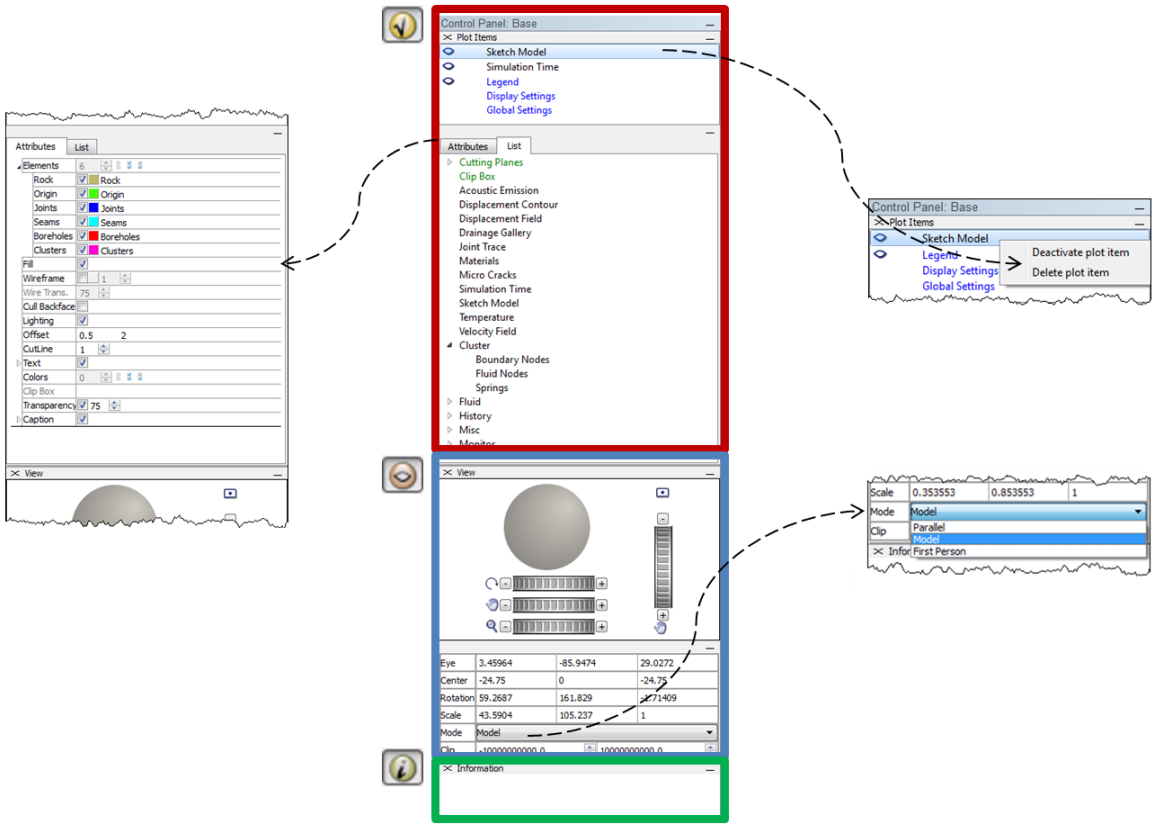


Figure 55 ***The Plot Control Panel consists of three main panels: Plot Items (list and menu tabs), View controls, and Information display. Each panel can be displayed or hidden using the icons shown to the left of the center image. Right-click on a plot item to deactivate/activate or delete it and click on the Mode drop-down list to modify the view perspective.***

View Controls Panel

The view controls panel allows manipulation of the model orientation, position and on-screen magnification. Much of the view manipulation capability provided by the **View** control set can also be obtained from the right mouse button and mouse wheel at any time — that is, regardless of which mouse mode is currently active. The available tools are summarized below.

	**Rollerball (3D)**	Rotates the model in 3D space.
	**Rotate (2D)**	Slide dial to the right (toward + sign) for continuous clockwise rotation, to the left (toward – sign) for continuous counterclockwise rotation. Rotation occurs in the view plane. Use the – and + buttons for incremental rotation.
	**Pan**	Use the horizontal dial for continuous horizontal panning to the left (toward – sign) or right (toward + sign); use the – and + signs for incremental panning. The vertical dial performs the same operations for the vertical direction.
	**Magnify view**	Slide the dial to the right (toward + sign) to continuously increase view magnification, to the left (toward – sign) to continuously decrease view magnification. Use the – and + signs for incremental changes to magnification.
	**Reset view**	Resets the plot to its initial default scale/orientation.

The plot view information panel provides continuously updated data on eye position, model center, model rotation, and scale. The text fields containing the data are editable and may be used to specify exact values for those view attributes. The fifth element of the view information is the view mode selector. There are three view modes available: **Parallel**, **Model** and **First Person**.

Eye	Describes the *x*-, *y*- and *z*-position of the eye in model coordinates.
Center	Describes the *x*-, *y*- and *z*-position of the view center in model coordinates.
Rotation	Describes the dip, dip direction and roll of the current viewing plane in model coordinates.
Scale	Describes the radius (extent from view center to view edge), eye distance, and magnification of the view. Radius and magnification both affect the apparent magnification of the view. However, radius uses a fixed angle for model perspective, so lowering the value will make the model appear closer, increasing it will make the model appear farther away and, in both cases, in order to satisfy the fixed angle of model perspective, the eye position is moved accordingly. Magnification increases or decreases the view without changing the eye position but correspondingly changes the radius value, with the result that increasing the value will enlarge the model while reducing the radius value and decreasing the value will reduce the model while enlarging the radius value.
Mode	The view is governed by the **Mode** setting; the three available modes are **Parallel**, **Model** and **First Person**. **Parallel** presents an orthogonal view of the model, while **Model** and **First Person** use a perspective view. In **Parallel** and **Model** modes, the view center is the default focal point of the view; in **First person** mode, the view center is the center of the screen.

Information Display Panel

The Information Display panel (Figure 56) provides non-interactive display of information regarding the current mouse position. When the mouse is over any linked part of the model, the information display will show attributes pertaining to that part of the plot item and the mouse position (in model coordinates). When the mouse is not over a part of the model, the information display is blank.

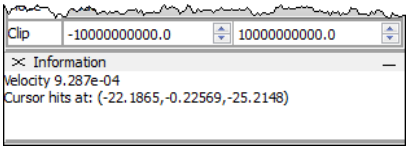
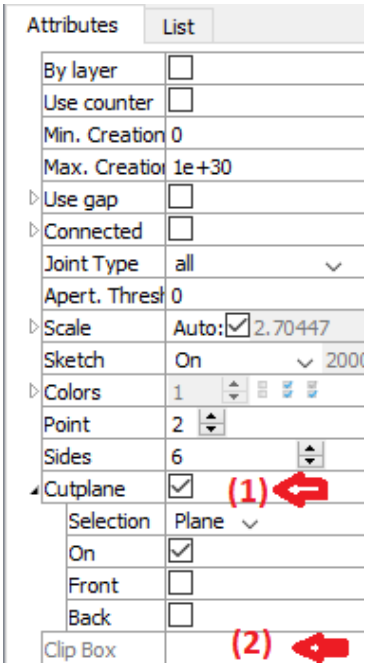


Figure 56 Information panel showing fluid flow rate for the vector under the mouse cursor in the plot window.

Plot Items

The following table details the plot items available in XSite.

Cutting planes	XSite supports cutting planes for a few selected plot items, including microcracks/disks, fluid temperature and matrix plot. When the plot item supports a cut plane, the **Cut plane** check box will be displayed in the plot attribute list. Once the cut plane is added, the check box is enabled. See item (1) in the figure below. Cutting planes can be controlled interactively when the mouse is in **Manipulation** mode. They may also be manipulated using the attributes provided on the plane's **Attributes** tab. The information below covers common controls that are applicable to all three plane types: planes, wedges and octants. A plane must be associated with a plot item first. To do this, first add a plane to the plot by double-clicking on the desired type on the **List** tab. Then select the plot item that will use the plane and check the **Plane** attribute from the item's **Attribute** tab (refer to adjacent image). If more than one plane is added to the plot, the select box is used to identify which cut plane is to be used (only one may be associated with a plot item). The **On**, **Front** and **Back** attributes provide further refinement of how the plane will "slice" the plot item. For instance, by selecting **On**, only the plane cutting the object will be drawn. By selecting **Back**, a section of the object from the plane where the object was cut towards the back of the object will be plotted. 

Clip box	A clip box defines a volume within which the plot items are visible. Plot items outside of the clip box are hidden. Clip boxes are not plot items in themselves, but modifiers applied to plot items. A clip box slices though plot item elements to display a true cut view. When in the manipulate mouse mode, clip-box interactive controls are displayed as a red axis. Multiple plot items may be associated with a single clip box, but any one plot item may be associated with only one clip box at a time. The clip box may be disabled by unselecting the **Clip box** control in the plot item attribute list. When a clip box is added to the plot item, the check box **Clip Box** is enabled. (Item 2 of the above figure).
Displacement contours	Plots lattice node displacements by the magnitude, or *x*-, *y*- or *z*-components as colored scalars. Users may select relative displacement or total displacement. Relative displacement is the node movement since the last reset node displacement command, whereas total displacement indicates the node displacement since the start of the simulation.
Displacement Field	Plots lattice node displacement as vectors. May be colored by magnitude. Users may select either relative or total displacement. Relative displacement is the node movement since the last reset node displacement command, whereas total displacement indicates the node displacement since the start of the simulation.
Rock Temperature	Plots lattice spring temperature as a scalar value. Points are located at the centroids of the springs.
Simulation time	Plots the accumulated *simulation* time, elapsed run time and estimated run time to finish in seconds. XSite memory usage (Kbytes) and the total number of lattice nodes and springs present are also displayed. The **Fluid time step** must be activated.
Sketch model	Plots a sketch, to scale, of the model elements, including Rock (i.e., matrix), Origin, Joints, Seam, Boreholes and Clusters.
User defined model	In case a user-defined model is used to replace the current spring model, this plot will display the property value associated with a spring.
Velocity field	Plots model velocity as vectors in lattice nodes. May be colored by magnitude. This plot can be used to check if the model is in equilibrium: a general guideline is that the node velocities should be less than 1e-6 m/s.

Cluster	**Boundary nodes**	Plots the location of the matrix nodes that are adjacent to the cluster-matrix boundary as a color value.
	Fluid node aperture	Plots the location and aperture of the fluid nodes associated with a spring that intersects and/or resides inside a cluster as a scalar value. The user has the option to plot all nodes or the nodes above or below the cluster average aperture.
	Fluid Nodes	Plots the location and pressure or injection rate of fluid nodes associates to a spring that intersect and/or resides inside a cluster as a scalar value.
	Pipes	Plots the connected fluid network path that intersects clusters. Pipes may be plotted using a uniform color, by flow rate or by aperture.
	Springs	Plots springs that are inside and/or intersect cluster boundaries.
	Unbreakable (elastic) springs	Plots, as a disk, the location of the springs that are unbreakable. A spring is unbreakable if the spring is located inside the cluster and the user sets the flag **Preserve inner cluster intact** in the Solution/Mechanical tab
Contours	To calculate the continuum fields (scalar or tensor), XSite uses an internal grid. The dimension of the zones of this grid are based on the edge length defined by the Lattice Grid Edge parameter located in the tab **Solution/Mechanical** or **Reset Lattice** toolbar shortcut. The spring property values located inside a grid zone are upscaled to provide the zone quantity. Any component of a tensor can be plotted. Before creating a contour plot, it is recommended to adjust the grid size to be at least five times the background resolution. The grid size can be adjusted on the "Reset Lattice" dialog. Also, adding a cut-plane will speed up the plot generation.	
	Fluid Nodes	Plots contours of the components of the permeability tensor, fracture porosity, density and fluid saturation in the zones of a rectangular three-dimensional grid.

	Stress	Plots contours of the components of the stress tensor in the zones of a rectangular three-dimensional grid. The stresses are calculated by averaging spring forces for specified cell (element) sizes. The cell size can be specified in the Reset Lattice dialog. The size of the cell used for averaging should be at least five times the resolution used. Otherwise, there will be excessive oscillations in the calculated stress values. One stress value is plotted for the entire cell (i.e., "block contours").

Fluid	**Cluster injection rate**	Plots the magnitude of fluid injection rate at each cluster as a scalar.
	Fines	Plots the location of proppant fines. Ideally the fines should be located at the tip of the fractures.
	Flow rate	Plots the fluid flow rate on the fracture and joints as a 3D vector.
	Fluid node groups	Plots a group of fluid nodes with the same color. A group is defined by a set of adjacent fluid elements connected by high aperture pipes. Node groups are formed when both fluid nodes of the pipe are fully saturated, and the Solution/Fluid tab parameters Aperture Cap and Aperture cap effect are set.
	Fracture fluid pressure	Plots the fracture fluid pressure at the fluid nodes as a scalar. A pressure threshold parameter can be set so that only fluid nodes with the pressure greater than the threshold are plotted.
	Injection rate	Plots the magnitude of the pipe flow rate as a scalar.
	Pipes	Plots the connected fluid network paths as a series of linear pipes. Pipes can be colored uniformly or by aperture.
	Proppant concentration	Plots the magnitude of proppant concentration as a scalar.
	Proppant stress	Plots the magnitude of propped fluid stress at the fluid nodes as a scalar.
	Propped fracture conductivity	Plots the magnitude of propped fluid conductivity at the fluid nodes as a scalar.

History	Plots any of the histories selected (Figure 91) prior to model simulation. Histories are sampled and stored during the run. History variables then can be plotted versus simulation time. The available histories include acceleration, calibration force, cluster pressure, number of cracks, displacement, flow rate, fluid pressure, matrix pore pressure, microcrack area per cluster, sheared volume, simulated area (in tension, in shear or in both), temperature, total inflow, total outflow and velocity. In order to record acceleration, select the velocity history (Figure 92). All histories are sampled at a single sampling interval. By default, the sampling interval for the history is every 500 steps. The sampling interval can be changed in the **Reset Lattice** window (shown in Figure 54) using **History sample interval (cycle)**. Different sampling interval values *cannot* be assigned for different history variables.	
Joints	The joint plot item allows the user to plot pre-existing joints and some of their properties and associated field variables and states. For example, *XSite* can color joints that have slipped and joints that are open differently.	
	Contour forces	Plots the normal or shear stresses applied to joints. The forces are the summation of the spring forces that are part of a joint and are located within a given radius from the joint location (spring center or joint-spring intersection) divided by the summation of the area of the selected joints. Note that this plot may take some time to plot and should not be used when a simulation is running.
	Properties	Plots a contour of the joint properties: friction angle, tensile strength or cohesion.
	Traces	Plots the position of springs that intersect associated joints present in the model. User may select to display all joints, only open joints, or joints that have slipped. The user also may select to display only joints associated with springs connected to a cluster and/or only joints that are permeable. If the check box **Use color** is selected, the joints will be colored according to the associated label defined in the **Features/Joints** dialog.
	Node displacement magnitude	Displays the magnitude of either the normal or shear displacement of the nodes of a spring intersected by a joint as a contour.

	Node shear displacement field	Displays shear displacement field of the nodes of a spring intersected by a joint.
Materials	**Node scalar factor**	Plots as color the node scaling factor of the nodes of a resolution domain with respect to the background resolution (background resolution scale factor is 1.0)
	Properties	Displays contours of the node properties including Young's modulus, fracture toughness, tensile strength, UCS and density.
	Spatial Variability	Plots a contour of the node spatial variability of the properties including Young's modulus, fracture toughness, tensile strength, UCS or density. The space variability distribution can be specified as uniform, log-normal or gauss.
	Uniform	Plots by color the position of the lattice nodes for rock matrix material(s) and seams. Core removed from open-hole completion, perforated tunnels and boreholes are indicated as "excavations".
Matrix	**Contours**	Plots contours of the flow rate or pressure of the lattice springs averaged in zones of the lattice rectangular three-dimensional grid. Cut planes may be applied to this plot item.
	Flow rate	Plots the fluid flow rate as a 3D vector field of the lattice springs averaged in zones of the lattice rectangular three-dimensional grid. Cut planes may be applied to this plot item.
	Fluid pressure	Plots the magnitude of fracture fluid pressure at the fluid nodes as a scalar.
Microcracks	**Average Orientation**	Plots the disk contours of the density of microcracks for zones of the lattice rectangular three-dimensional grid. The disks are drawn in a plane defined by the average dip and dip direction and centered in the average location of the microcracks in the zone. Cut planes may be applied to this plot item.

	Disks	Plots the location and orientation of any rock damage (i.e., broken matrix spring) as disks. The microcracks may be colored by layer or by microcrack counter. There is also an option to plot only the connected microcracks and color them by cluster. Cut planes may be applied to this plot item.
	Permeability tensor	Plots the permeability tensor in 3D as three vectors in directions of principal values. The location and magnitude of the vectors are the permeability of the microcracks averaged for the lattice rectangular three-dimensional grid. Cut planes may be applied to this plot item. The permeability tensor is calculated under the assumption that all cracks are continuous within the zones.
Micro-seismicity	**Imported**	Plots amplitude, time or stage of imported microseismicity data. See the File/Import Data/Microseismicity section for a detailed explanation.
	Predicted	Plots position and magnitude or start time of acoustic emission events associated with broken springs or joint slippage as spheres. May be colored by magnitude or start time.
Miscellan-eous	**Axes**	Plots three-dimensional orthogonal axes in the *x*-, *y*-, *z*-directions or two-dimensional axes in the North-South, East-West directions. If the plot is set to **fixed**, the axis will be plotted in the left lower corner of the model. If the axes are not fixed, they will be plotted at the location specified by the user. The plot will rotate as the model rotates (hold the left mouse button and spin the mouse).
	Global coordinates axes	Plots three-dimensional orthogonal axes in the *x*-, *y*-, *z*-directions or two-dimensional axes in the North-South, East-West directions. The axes will be plotted at the location specified in the fields that define the **Reference Point** in the **Main Rock/Geometry** tab. The plot will rotate based on the value of the field **Azimuth** defined in the **Main Rock/Geometry** tab.

	Remove FRACMAN fractures	This plot removes specific fractures from FRACMAN joint sets as follows: a) Import the FRACMAN joint set. b) In this plot item, expand the **Fractures** drop box and unselect the fractures that will not be included in the model. c) Press the **Save removed fractures** button. d) Initialize the model and use the **Joint Traces** plot items to verify if the fractures were removed
	ScaleBox	Draws a set of three orthogonal planes around the model. The planes are oriented in the *x*-, *y*-, and *z*-directions of the model. The planes are divided in a two-dimensional grid, showing a user-defined scale, allowing the user to better visualize the scale and extent of the model. It also helps to measure elements inside the model.
Monitor	**Carter leak-off offsets**	Displays the Carter leak-off offsets (for fluid arrival time in calculation of leak-off volume) in the fluid nodes.
	Connected pipes	Displays the aperture of the pipes that are connected to a cluster.
	Fixity	Plots the location of the lattice nodes colored based on the node degrees freedom.
	Fluid joint upstream	For each fluid node, this displays the vectors pointing to the closest upstream node. The fluid joint upstream pointer is used to calculate the pressure and proppant distribution when running the simulation using simplified logic with pressure gradient.
	Gap	Displays the gap in open cracks or joints.
	Grid	Plots the zones of the lattice rectangular three-dimensional grid, which is used for averaging different variables to plot them as continuum fields (e.g., the permeability tensor). The edge of this grid is defined in the **Main Rock** tab.

	MPI domain decomposition	Displays a Cartesian domain decomposition. The division in the *x*-, *y*-, and *z*-directions are defined in the plot by the user.
	Path	Display the incremental distance of the fluid node from the cluster. Fluid nodes immediately connected to a cluster are assigned the path equal to one. A higher path value will be assigned to nodes as they are located farther from the cluster.
	Spring Forces	Plots the forces acting at the springs between lattice nodes using a color scale.
	Spring model	Displays the spring model types used by different colors.
	Spring properties	Displays either the spring area or length.
	Spring state	Displays the type of spring (e.g., regular or joint) in the model as color.
Stress	**Tensor**	For each zone of the lattice rectangular three-dimensional grid, this plot item displays the upscaled stress components of the set of springs located inside the zone.
Sub-lattice	Group of plot items that uses springs and nodes of the sub-lattice. The descriptions for the main lattice apply to these plot items.	

6.0 Model Setup Panel — Main Tabs

The five tabs on the left side of the XSite **Model Controls** panel are reviewed here.

Main Rock Tab

Geometry Tab

The **Geometry** tab of the **Main Rock** panel (Figure 57) enables the geometry of the model (box shape) to be set up. Note that rectangular coordinates, **X**, **Y** and **Z**, are used: these correspond to **East**, **North** and **Up** when other instances of geological information are needed. (For example, the dip direction (used by the J**oint sets** dialog) is relative to **North,** or **Y.**) The origin of the model (box) is located by default at the most easterly center-top position of the model block (refer to the point ***Origin*** in Figure 1). The model local origin is defined relative to a reference point by specifying a relative East, North and/or Up distance.

The fields in the **Reference Point** group box and **Azimuth** are used to allow generalization of the model positioning, allowing transforming the model to the global coordinate system. The **Reference Point** fields indicate the origin of the model in the local coordinates. The **Azimuth** field is the angle in degrees that the model is rotated in clockwise direction in relation to the north. The lattice engine builds the model and plots items are displayed in the local coordinate system. The built-in DFN, seams and borehole generators allow the user to enter data in global coordinates. When importing DFN data, an option to transform the data from global coordinates will be given. An option to export data in global coordinates is also available.

The rock type **Material** may be selected from a drop-down list of pre-defined materials, or new materials may be created by using the Edit Materials Dialog icon as described previously.

The standard spring model may be replaced with a different spring model. The model is developed as a DLL with respect to an interface specified by Itasca. The DLL is loaded during run-time and its name is included as one of the options in the **Spring Model** drop-box.

The model **Geometry** is set by entering the block **Length (L)**, **Height (H)** and **Width (W)** into the corresponding fields.

The values provided by the user must be entered according to the units displayed in the field labels. By default, the SI unit system is used. If the field **Use Oil Field Units** is selected, the oil-field unit system is used. The fluid and rock materials data will be automatically converted when the unit system is changed. All the plot items and export features will provide values in the corresponding field units. The imported values must also be provided in the correct unit system.

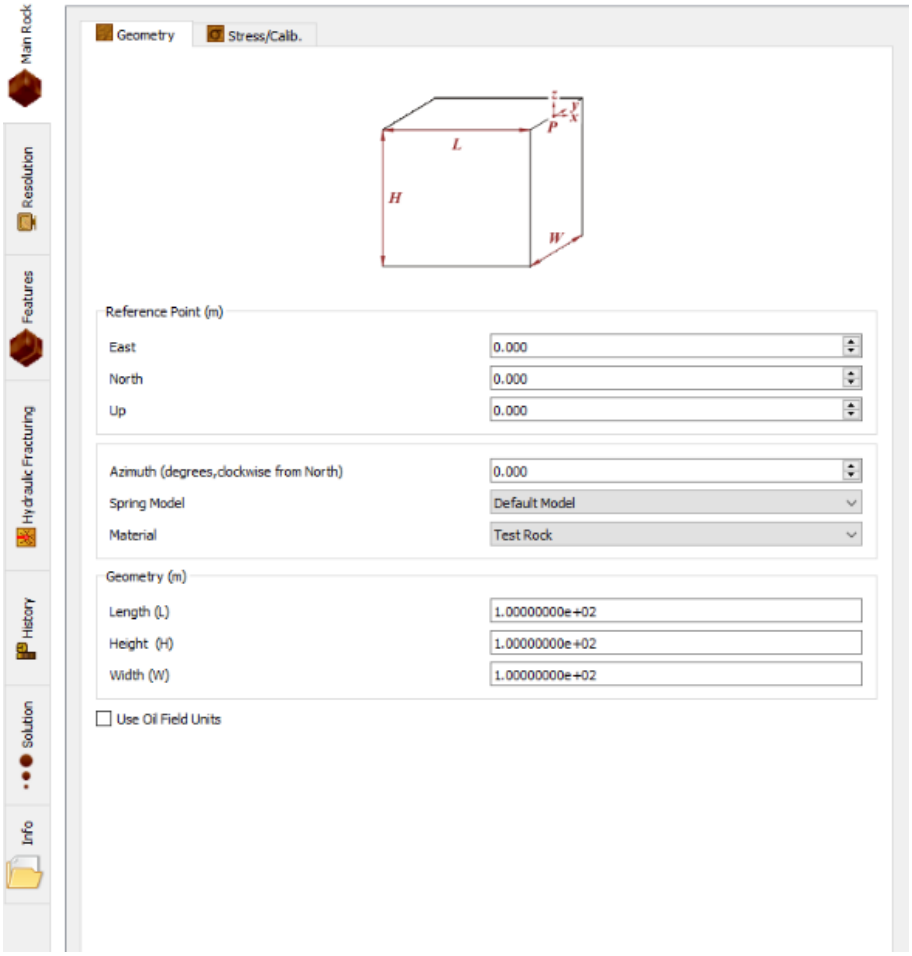


Figure 57 The Main Rock Geometry tab can be used to specify the model size, reference position, rock type and the different (oil-field) unit system.

Stress/Calibration Tab

The **Stress/Calibration** tab (Figure 58) of the **Main Rock** panel allows the mode of simulation to be set and initial and boundary conditions for the simulation to be entered. With **Calibration** mode not checked (default mode), this tab is used to set the model initial stress state. Arbitrary orientation of the principal stresses can be specified (i.e., oblique relative to the coordinate axes and the model boundaries). However, the three principal stresses must be orthogonal; if not, the code will display an error message and not execute the **Simulate** command.

If the minimum principal stress is not zero, specifying a **Reference minimum stress** value is recommended. During initialization of the model, this value will be subtracted from all components of the stress tensor and from fractures and matrix fluid pressure. This value is recommended to be equal to the smallest value of the minimum principal stress in the model. If the minimum principal stress is manually set to zero and all other stress and pressure variables are consistently corrected, the pressures

in the model will be net pressures. However, in the plot items that display stresses or pressures, the reference minimum stress may be added back, hence the results will be shown as absolute values.

Model far-field boundary conditions (on the vertical boundaries) can be assigned as either fixed (default setting) or as rollers by checking the **Roller far-field boundary** check box. It is recommended to use fixed boundaries if the principal stresses are oblique relative to the boundaries to prevent reorientation of stresses. The top and base of the model are always fixed.

If **Calibration** mode is checked, a special mode is activated that allows material (intact or fractured) strength and stiffness to be tested and calibrated; note that this is currently intended for advanced users only. Calibration mode can be activated only before starting a model simulation. Boundary conditions for a calibration test can be set with **z-velocity (m/s)** (velocity applied on the top boundary) and **Confinement pressure** (MPa), which applies load on the all four model sides. If **Confinement Pressure Symmetry** is checked, the calibration simulation is run assuming quarter-symmetry along the horizontal X-Y plane, which will reduce the simulation run-time.

XSite also provides the ability to perform a **Bending test**. The **Platen area** is defined by multiplying the background resolution to the input entered.

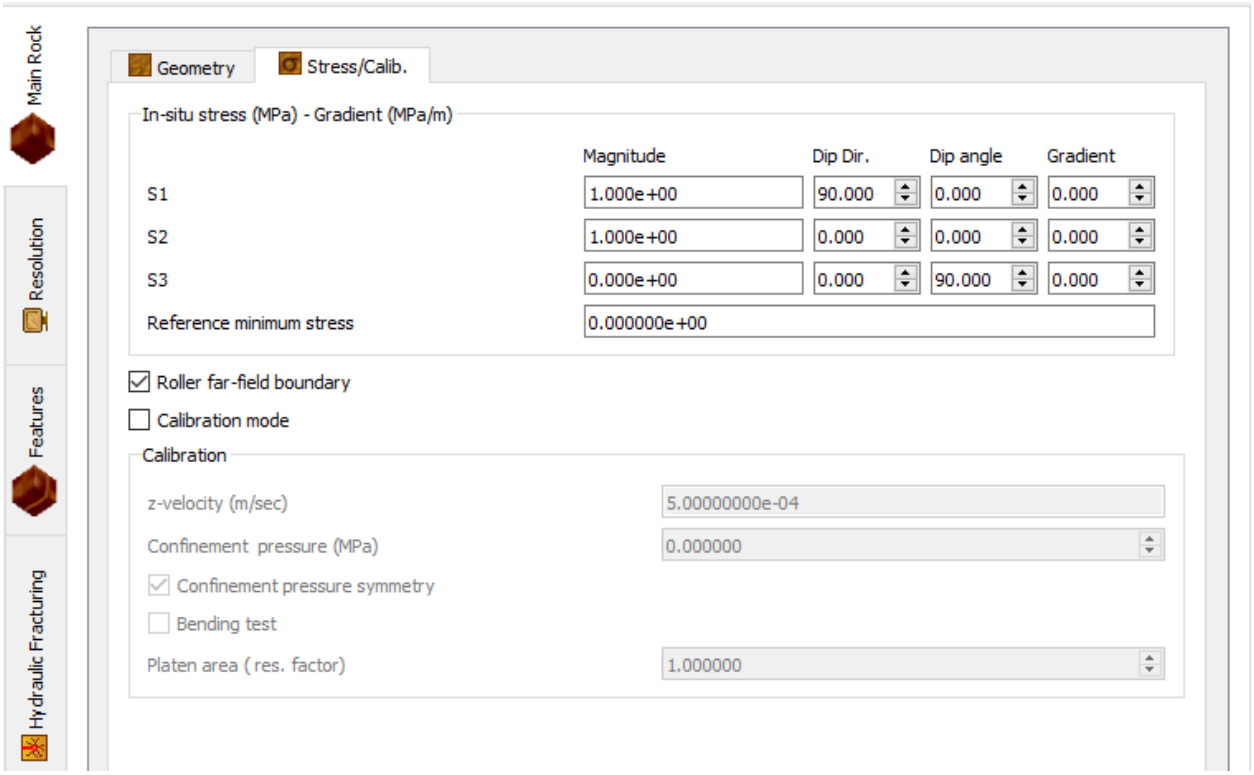


Figure 58 Main Rock Stress/Calibration tab can be used to specify the in-situ rock stress or calibrate the lattice model to actual rock strength properties.

Resolution Tab

It is recommended to have ~4 resolutions per fracture spacing/diameter in the area of interest (i.e., close to the injection point). Maintaining the same ratio everywhere in the model would be too costly computationally. Domain resolutions allow to grade the lattice resolution toward the far-field boundaries, which will optimize the number of nodes and springs in the model. Unfortunately, the critical time step will be controlled by the minimum model resolution. (The time step of the flow model is proportional to the minimum resolution squared.)

Add, edit, clone or delete a domain resolution

Resolution domains allow use of finer resolution in regions that need more detail and coarser resolution in the far-field regions. Consequently, the model can be discretized with fewer springs and nodes, and, consequently, faster execution times.

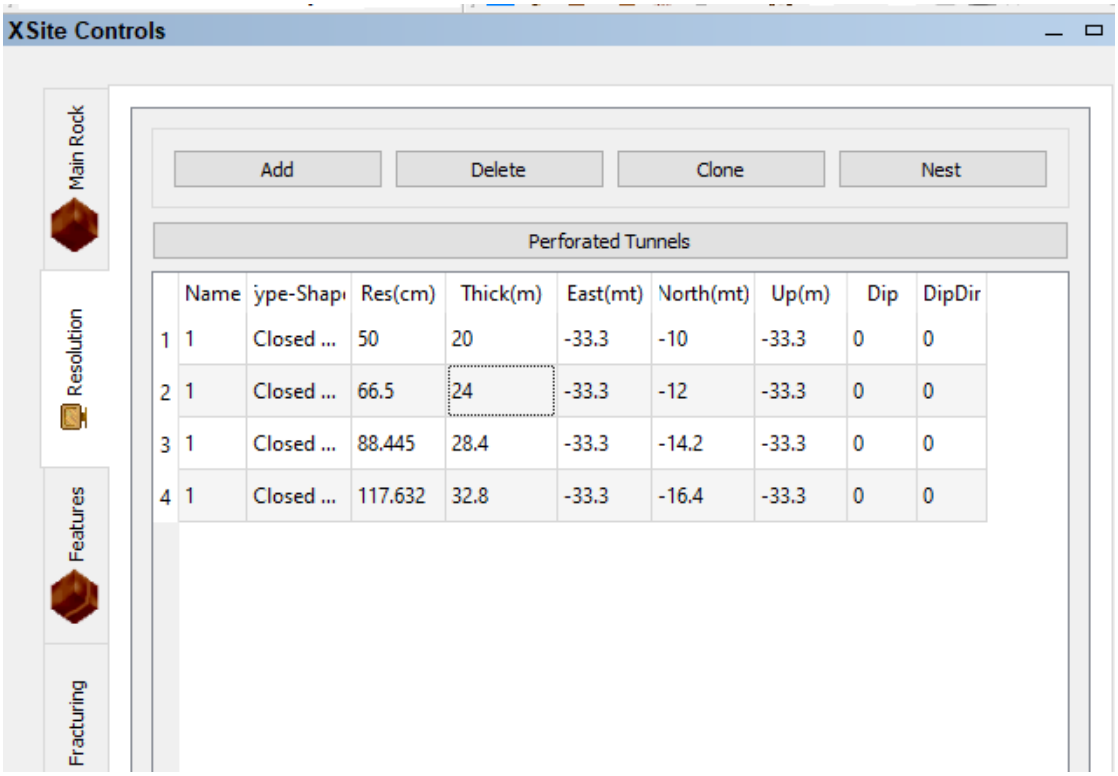


Figure 59 Domain Resolution tab.

Existing domain resolutions are shown in Figure 59. New domain resolution entries can be added to the simulation by clicking on the **Add** button, which will bring up the **Resolution Domain** generator dialog box (as shown in Figure 60). Clicking on the **Clone** button while one of existing joint sets is selected will copy that resolution domain entry. This new entry can be modified by double-clicking the cloned entry.

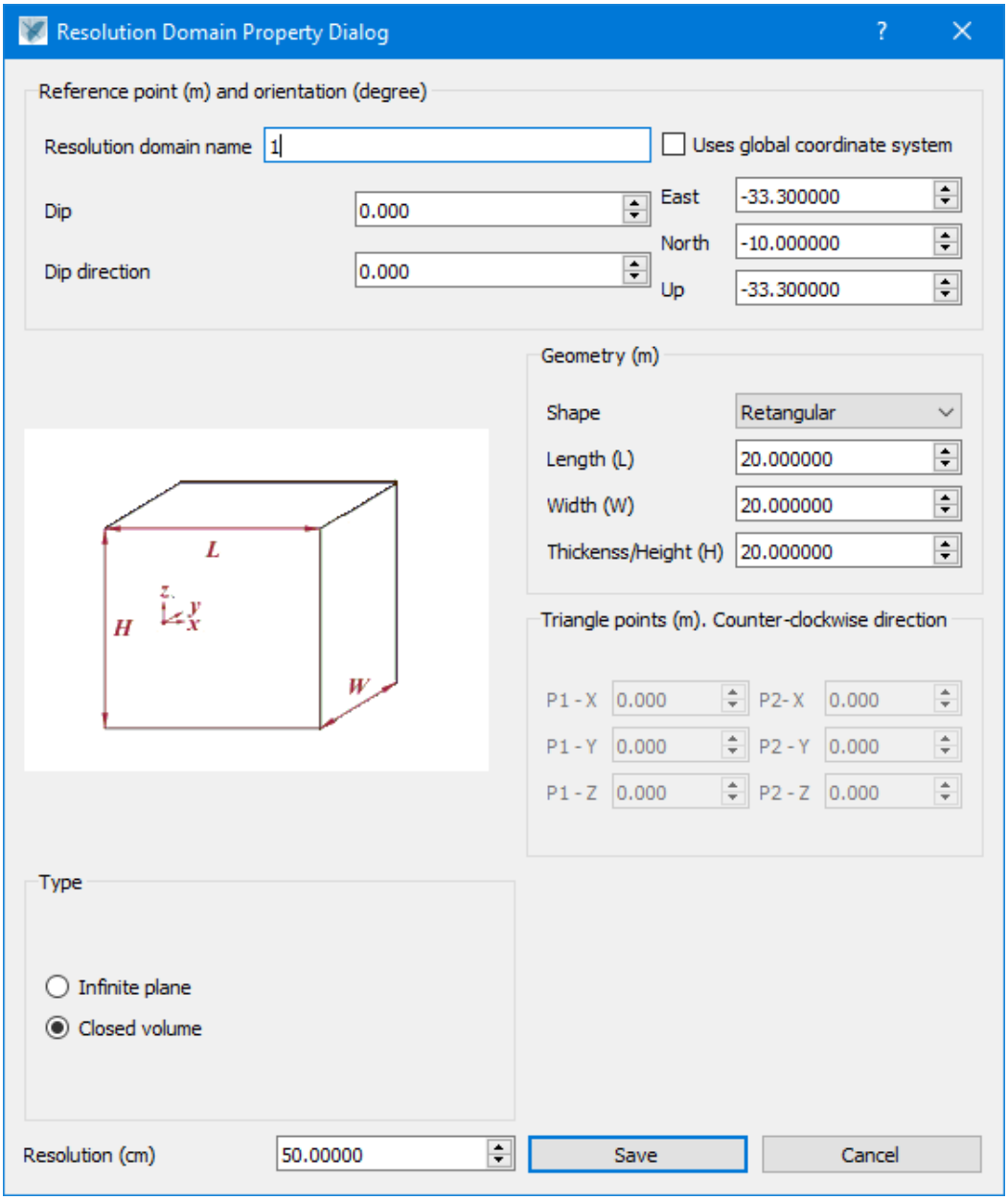


Figure 60 Resolution Domain dialog.

The table below describes the fields found in this dialog.

Domain resolution name	An arbitrary number given by the user. It helps identify the resolution domain in the **Sketch Model** plot item and when specifying a simulation with adaptive resolution.
Dip angle	The dip angle (downward from horizontal) of the resolution domain is specified.
Dip direction	The dip direction (the angle from North toward East) of the domain resolution is specified.
Uses global coordinate system	If this field is marked in the sketch plot and during the model initialization, the field **Reference** Point specified in the **Main Rock/Geometry** tab will be subtracted from the values given in the reference fields described below. If an azimuth value is given, it will be applied to the translated resolution domain reference point. After the model is created, the **Materials** plot items should be used to verify based on the nodal density if the resolution domain was created in the proper location.
Reference point	The origin of the resolution domain.
Geometry: Type, Shape, Length (L), Width (W) and Thickness/Height (H)	Two types of geometry are available: infinite plane and closed volume. An infinite plane domain resolution is defined by two parallel planes. The distance between these planes is given by the **Thickness/Height** parameter. The parameters **Length** and **Width** are disabled when the infinite plane option is selected. A closed volume can be either rectangular or cylindrical. Length is measured along the East-West or X-axis. Width is measured along the North-South or Y-axis. Thickness is measured along the Z-axis.
Resolution	This field defined the lattice resolution that will be applied to the 3D region defined by the resolution domain parameters. Note that this parameter cannot exceed the value defined in the background resolution lattice. It is recommended that there must be at least 4 to 5 nodes across any resolution dimension. The jump between resolutions should be between the interval of 25% to 50%.

Add nested domain resolutions

When you select a resolution domain from the list in Figure 59 and press the **Nest** button, the dialog **Add Nested Resolution Domains** dialog displayed in Figure 61 is shown. This feature allows building a set of nested resolution domains with a gradual transition from an inner finer resolution domain (the resolution selected by the user) towards the coarser background model resolution. This gradual transition is recommended to achieve an optimized model discretization. This feature can be applied only to closed-volume domain resolutions. A domain resolution must be selected before pressing the **Nest** button. The **Number of layers** field indicates how many layers will be added around the selected domain with the finest resolution. The dimension (thickness) of the first layer around the selected domain can be specified in the **First layer thickness** field. If more than one layer is specified, the **Dimension Factor** will be applied as a multiplier (relative to the thickness of the previous domain) to calculate thickness subsequently to the following domains. The **Resolution Factor** field is used to define the relative increase in the resolution between the nested domain, starting with the domain in the first added domain towards the outside domain. The program will stop adding layers when either of these conditions occurs: the **Number of layers** has been reached or the calculated resolution of the next layer is greater than the model resolution After building the model, the **Materials** plot item can be used to verify the resolution of the various domains were properly calculated.

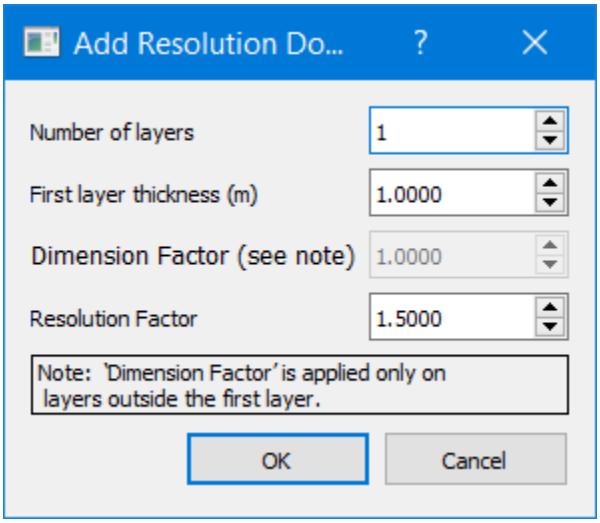


Figure 61 Add nested domain resolution dialog.

Add domain resolutions around perforated tunnels

When you press the **Perforated Tunnels** button, the dialog **Add Perforated Tunnels Resolution** dialog displayed in Figure 62 is shown. This feature allows building a set of nested cylindrical resolution domains around perforated tunnels. The domains are created with a gradual transition from an inner finer resolution domain (the resolution that immediately envelops the perforated tunnel) towards the coarser background model resolution. This gradual transition is recommended to achieve an optimum model discretization. This feature can be applied only to perforated tunnels currently present in the

model. The field **Resolution** defines the lattice resolution to be applied to the domain immediately adjacent to the perforated tunnel. The **Number of resolution domains** field indicates how many domains will be added around the adjacent domain, which will have the finest resolution. The **Radius factor** field is a multiplier that will be applied to the radius of the perforation tunnel to calculate the radius of the domain adjacent to the perforated tunnel. If more than one domain is specified, the **Outer radius factor** defines the radius increase of the subsequent cylindrical domains relative to adjacent inner domains. The **Resolution Factor** field is used to define the relative increase in the resolution between the nested domain, starting with the domain in the first added domain towards the outside domain. The program will stop adding domains when either of these conditions occurs: the **Number of Layers** has been reached or the calculated resolution of the next layer is greater than the model resolution. After building the model, the **Materials** plot item can be used to verify, based on nodal density, that the resolution of the various domains were assigned proper resolutions. Make sure that the ratio of the resolutions of the adjacent domains is not greater than 1.5. The Spring Flat Joint Model may be applied to the domains created by this operation.

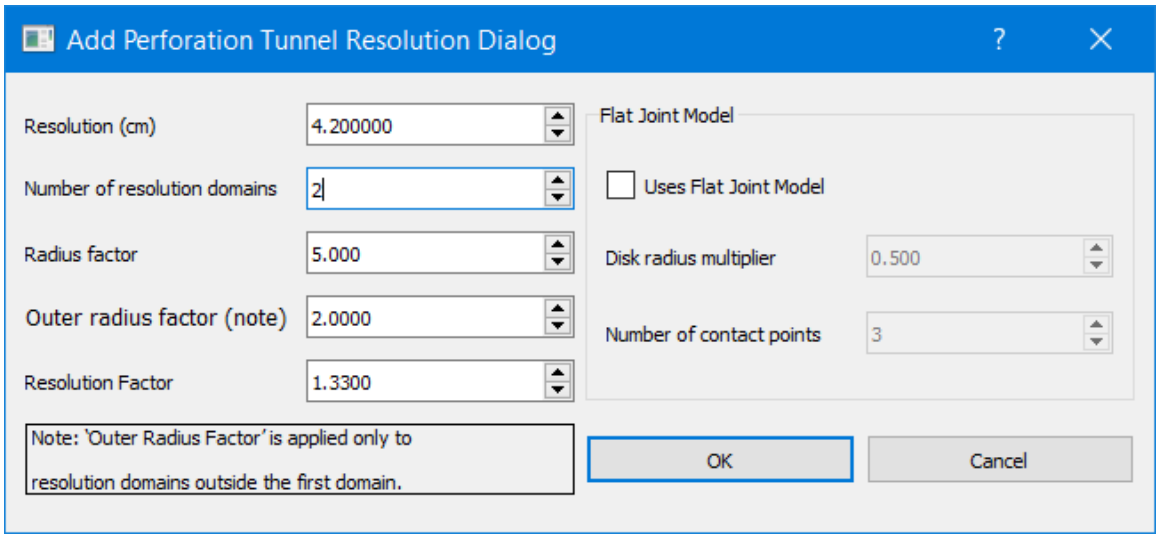


Figure 62 Dialog to add resolution domains around perforated tunnels.

Features Tab

When the Features tab is selected, six tabs will be available: Joint Sets, Seams, Fluid, Proppant, Thermal and Sink. The set of horizontal arrows on the right corner of the page can be used to access the desired tab. These tabs are described in the following.

Joint Sets

Joint sets are groups of fractures in the rock with similar orientation and spacing. Joints can fail both in shear and tension. Existing model joint sets are summarized as shown in Figure 63. New joint sets can be added to the simulation by clicking on the **Add** button, which will bring up the **Joint Set generator**

dialog box (as shown in Figure 64). Clicking on the **Clone** button while one of the existing joint sets is selected will create a copy of that joint set, which then can be edited to create a new set as a variation of the existing one.

The code will check for springs that are intersected by two or more predefined fractures. If the angle between two intersected predefined joints is greater than the field **Angle tolerance for joint intersection**, highlighted in Figure 63, the springs are marked. The marked springs will be skipped in the mechanical calculation to prevent locking on one of the intersected planes because the smooth joint contact is oriented perpendicular to the plane of other joint(s). The marked springs will not be included in the pipe network.

Note that the DFN may introduce stress perturbation resulting in spurious cracking. Make sure to run the model to mechanical equilibrium, starting with elastic mechanical equilibrium, before starting injection.

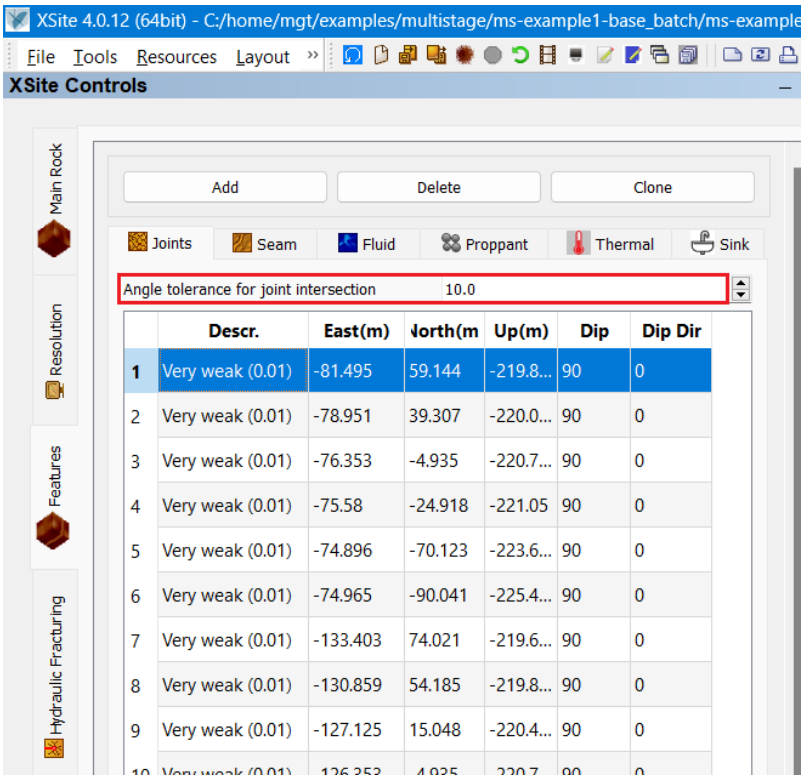


Figure 63 Tab displaying the joint sets that are currently defined in the model.

By pressing the **Add** button or double-clicking an item in the joints list under the **Joints** panel of the **Features** tab, the dialog shown in Figure 64 will be displayed.

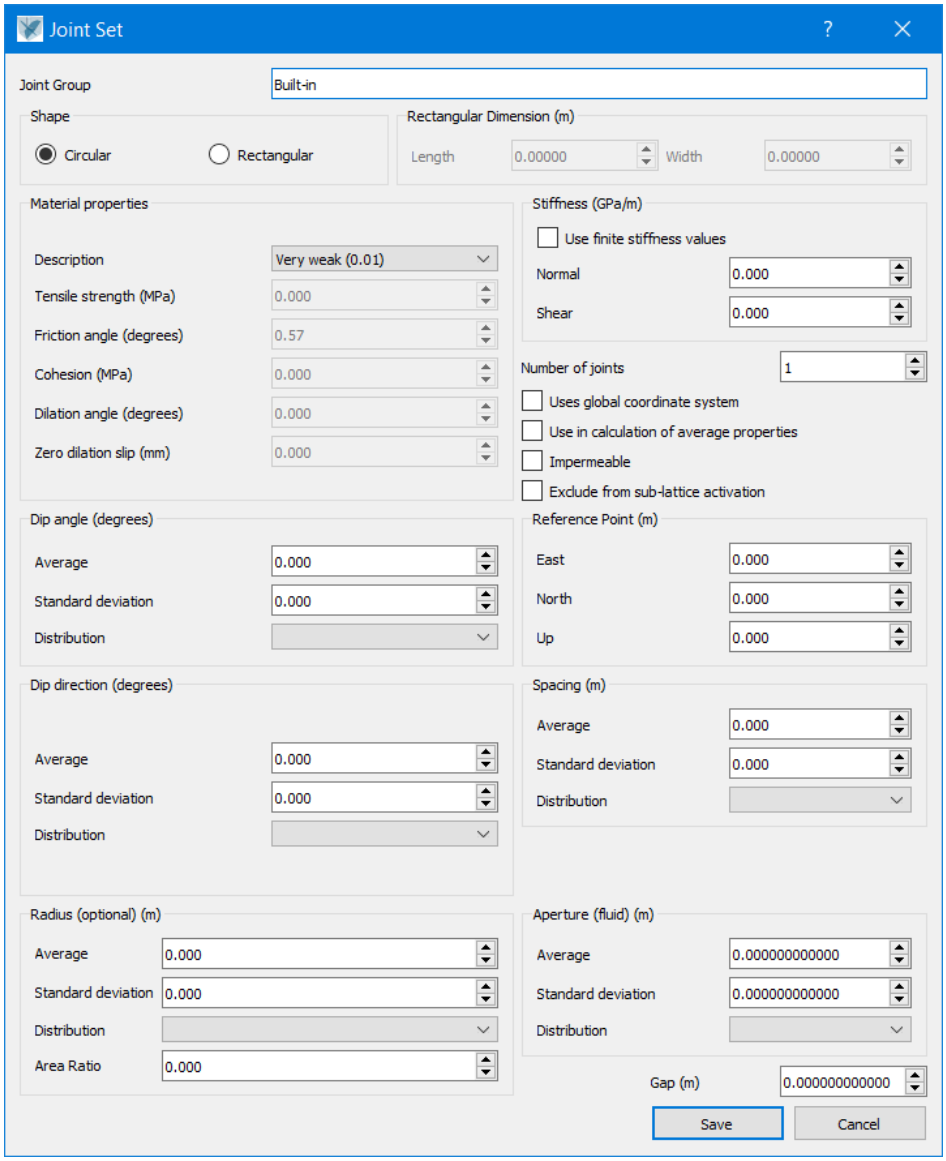


Figure 64 Joint sets can be generated and properties assigned using the Joint Set dialog probabilistically or manually.

The dialog from Figure 64, evoked by pressing the **Add** button, is the built-in joint generator editor. When the various fields are completed and the **Save** button is clicked, a new joint set is added to the model. It should be noted that the joints created by the built-in joint editor are not "committed" until the lattice is built; thus, changes to the joint sets may be made at any time before to the first **Simulate** command. A joint set may consist of several continuous planar discontinuities or circular ("penny-shaped") cracks. Each distinct joint node is subject to given statistical variations in angle, extent and spacing. The fields in the **Joint Set** dialog box are described as follows.

Joint properties input data

Joint Group	An arbitrary name given to the joint set. The **Joint Traces** plot items may use this field to color a group a set of joints with the same name.
Description of material properties	Named materials are pre-defined in the pull-down list, with their friction coefficient indicated in parentheses. Thus, **Medium Weak** material has a friction coefficient of 0.25 (friction angle of approximately 14°). Alternatively, the **User Defined** material may be selected. In this case, the friction angle (in degrees), cohesive and tensile strength, together with a user-specified stiffness, should be entered. Dilation angle and shear displacement for which dilation becomes zero also can be specified.
Dip angle	The average dip angle (downward from horizontal) of the joint set is specified. When there is more than one joint segment, the **Standard Deviation** controls the range of angles according to the specified statistical distribution (**Gauss** or **Uniform**).
Dip direction	The average dip direction (the angle from North toward East) of the joint set is specified. When there is more than one joint segment, the **Standard Deviation** controls the range of angles according to the specified statistical distribution (**Gauss** or **Uniform**).
Radius	If the average radius is given as zero, continuous joint planes are created. Otherwise, circular cracks of the given average radius are created at random within each notional joint plane until the ratio of total crack area to total joint-plane area (within the model) is equal to the given **Area Ratio**. The cracks are subject to the given **Standard Deviation** of radius and angle (dip and dip direction) if the angle parameters have associated **Standard Deviations**.
Stiffness	**Joint Normal** and **Shear** stiffness can be specified. In the mechanical calculation mode, if default values of zero for the stiffness fields are used, the stiffness of the spring that represents the joint is equal to the original matrix stiffness intersected by the joint (i.e., the joint stiffness effectively is infinite). Added joint stiffness will act in series with the matrix stiffness. If the field **Use finite stiffness** is used, the user must enter values for normal and shear stiffness.

Number of joints	This parameter sets the target number of joint planes to be generated. Planes are generated starting from the **Reference Point**, alternately creating planes on both sides of the initial plane. There is no problem (apart from increased processing time) in specifying more planes than would fit within the model extent — planes that fall outside are simply discarded.
Uses global reference	If this field is marked during the sketch plot and model initialization, the field **Reference Point** specified in the **Main Rock/Geometry** tab will be subtracted from the values given in the reference fields described below. If an azimuth value is given, it will be applied to the translated joint reference point. After the model is created, the **Joint Traces** plot items should be used to verify if the joint was created in the proper location.
Use in calculation of average properties	If set, joint properties will be included in the calculation of spring properties when exporting or plotting quantities that are upscaled by using the grid of zones.
Impermeable	If this field is checked, the fluid nodes of the joint will be marked impermeable, and they are not included in the fluid calculation. The fluid nodes of the joint will become permeable only after they slip or open (i.e., fail in shear or tension) when these fluid nodes will lose cohesive and tensile strengths. Permeable fluid nodes are included in the fluid flow calculation. The **Fluid/Pipes** plot item has an attribute for displaying permeable and/or impermeable pipes.
Exclude from sub-lattice activation	If this flag is checked, the springs intersected by the joint will not be used to activate sub-lattice zones.
Reference point	The first plane generated has its origin at the given coordinate.
Spacing	The target spacing between generated planes is given. If the **Number of Joints** is greater than 1, then the given **Standard Deviation** applies to spacing.

Aperture (fluid)	The hydraulic aperture is specified. Non-zero aperture must be specified for flow to occur in the joint. If all joints have zero apertures and flow calculation is requested, the code will display a warning and no calculation will be carried out. If multiple joint segments are generated, the given **Standard Deviation** is applied to the distribution of apertures.

Three-dimensional DFNs also may be imported into XSite (refer to **Import DFN** items under **File** in the **Main Menu Bar** section). The following formats are available:

- Golder Associates FRACMAN format.
- ITASCA FISHLab format (text, space-delimited format). This format assumes joints are circular disks.
- XML format. Assumes that joints are triangular faces.

A joint set imported from a file with either the FISHLab format or the XML format cannot be edited. The joint sets imported from FRACMAN format allow editing of the fields **Joint group**, **Stiffness**, **Uses global coordinates system**, **Impermeable**, **Exclude from sub-lattice activation** and **Aperture**.

Itasca FISHLab format

The CSV format consists of a comment line followed by a line containing the number of joints in the DFN (e.g., 4) and the number of extra variables or user-defined data (e.g., 1). The rest of the file contains joint information in the following order: "label" (in quotations), id, *x*-center, *y*-center, *z*-center, dip, dip-direction, disk radius, material, normal stiffness, shear stiffness, dilation angle and initial dilation slip. The following is an example of a DFN with four joints:

```
;project xyz DFN generated from 3FLO, June 2011
4          1
"Set A"    41277    -16.2    -24.8    -41.7    30.0    10.0    5.9    1    1.1    2.2    5.0    1.2
"Set A"    41274    12.4     -51.4    24.1     62.0    227.0   8.2    1    3.3    4.4    5.0    2.2
"Set A"    41277    -36.0    -34.7    -31.0    30.0    10.0    5.9    1    5.5    6.6    5.0    3.3
"Set A"    41274    42.1     -41.6    14.9     62.0    227.0   8.2    1    7.7    8.8    5.0    4.4
```

XML Format

The XML format consists of a series of interbedded data structures beginning with an XML declaration statement, a schema (i.e., some standard set of rules for the data), XSite version, mechanical properties for each joint, and the position of each triangular face (centroid and triangular vertices, ordered counterclockwise) making up the joint. An example of an XML DFN text file follows. While data order is important, any number of joints or triangular faces defining a joint can be specified and attributes modified. Any fracture shape can be represented by discretizing into triangles.

```
<?xml version="1.0" standalone="yes"?>
<itasca-dfn xsi:noNamespaceSchemaLocation="itasca-dfn.xsd" xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance">
<version>1.0.0</version>
 <joints>
  <joint>
   <joint_id> 1 </joint_id>
   <description>Medium-strong (0.75)</description>
   <aperture>0.000100</aperture>
   <tensile_strength>1.0</tensile_strength>
   <cohesion>2.0</cohesion>
   <friction>3.0</friction>
    <normal_stiffness>0.0</normal_stiffness>
    <shear_stiffness>0.0</shear_stiffness>
    <dilation_angle>0.0</dilation_angle>
    <zero_dilation_slip>0.0</zero_dilation_slip>
   <x>3.5643</x>
   <y>4.9874</y>
   <z>-0.4809</z>
   <triangle>
    <x2>5.6972</x2>
    <y2>7.9761</y2>
    <z2>-0.7571</z2>
    <x3>3.5443</x3>
    <y3>8.6117</y3>
    <z3>-1.1303</z3>
   </triangle>
   <triangle>
    <x2>3.5443</x2>
    <y2>8.6117</y2>
    <z2>-1.1303</z2>
    <x3>1.3990</x3>
    <y3>7.8630</y3>
    <z3>-1.2554</z3>
   </triangle>
  </joint>
 </joints>
</itasca-dfn>
```

Seams

Seams are layers (i.e., infinite planes with a thickness) or finite volumes of different material within the model domain. Existing model seams are indicated as shown in Figure 65.

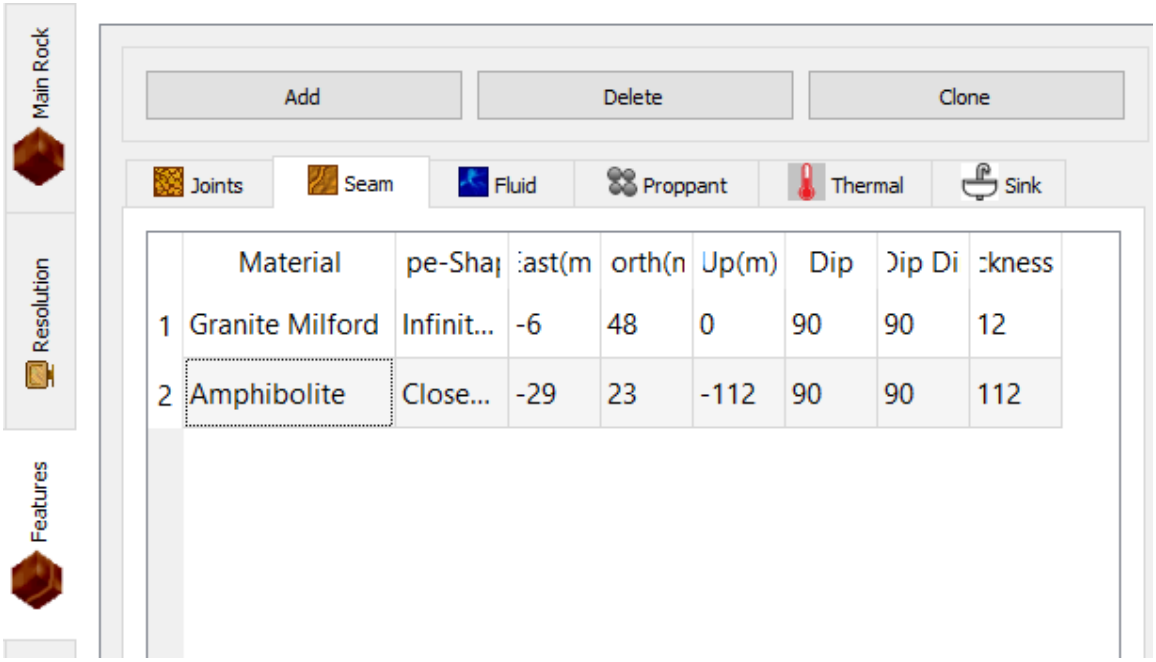


Figure 65 New seams can be added to the model or existing ones viewed under the Features tab.

New seams can be added to the simulation by clicking on the **Add** button, which will bring up the **Seam Properties dialog box** as shown in Figure 66. Clicking on the **Clone** button while an existing seam is selected will copy that seam, allowing it to be edited to create a new seam as a variation of the existing.

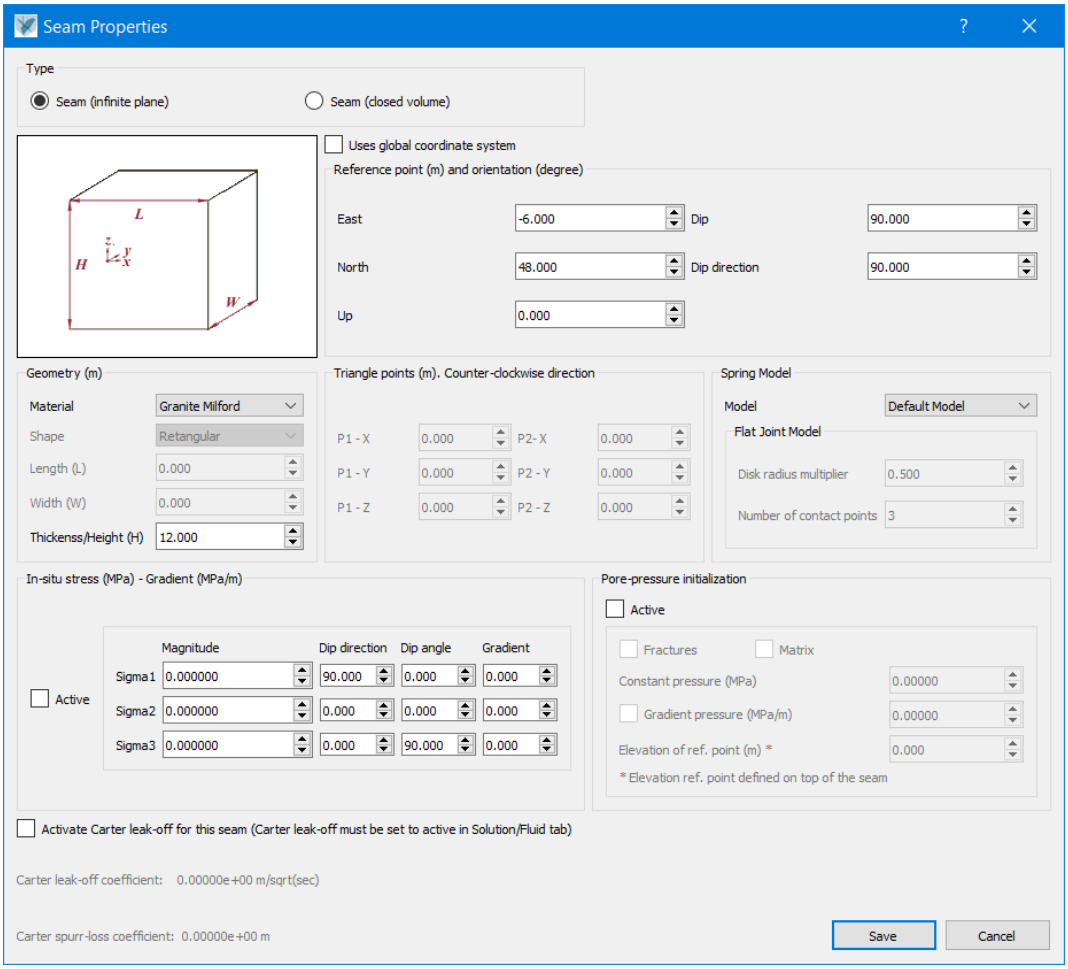


Figure 66 Seams can be defined using the Seam Properties dialog.

The **Seam Properties** dialog allows a region of a different material to be added to the model. The parameters **Reference Point**, **Orientation** and **Uses global coordinate system** have the same function as those already described for Joint Sets.

Two types of seams are available: infinite plane and closed volume. An infinite plane seam is regarded as a layer of material (at the given location, angle and given **Material**) of thickness given by the **Thickness/Height** parameter, as shown in Figure 63. A closed-volume seam can be defined using a rectangular, cylindrical, or triangular region via the **Shape** dropdown, with the parameters shown in Figure 67. Length is measured along the East-West or *x*-axis.

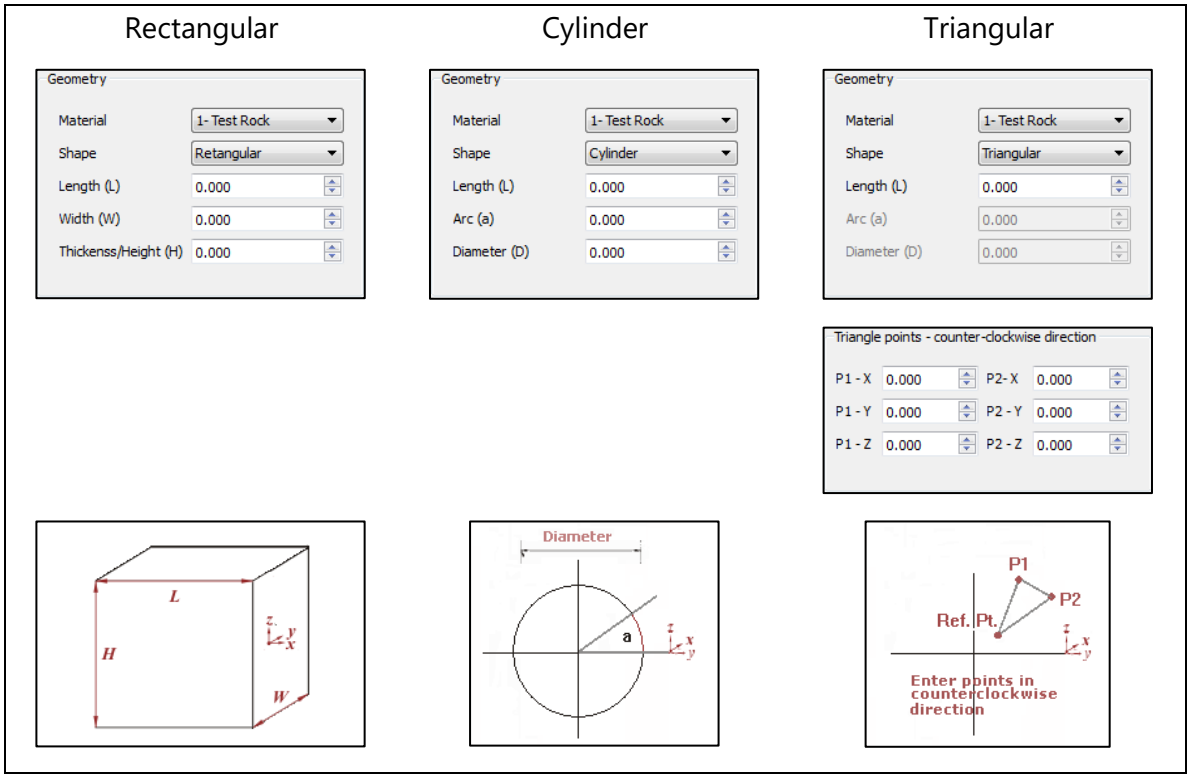


Figure 67 Geometrical properties for closed volume seams

A different in-situ stress state can be assigned for each seam by specifying the magnitude and orientation of its principal stresses. See the section Stress/Calibration tab for a description of input fields in the in-situ stress section.

A different spring model may be applied to a specific seam region by selecting from the **Spring Model** dropdown box.

If the **Uses Flat Joint Model** check box is selected, this seam domain will be constructed with the Flat Joint (FJ) model as an alternative to the default lattice spring model. For the FJ model, the user may select the number of sub-contact points and the contact disk radius. See Solution Tab/Flat Joint for further explanation on the FJ model setup.

The initial pore-pressure condition for fractures or the matrix may be set by selecting the **Activate initial pore-pressure** checkbox. If gradient pressure is used, the *z*-reference point is the top of the seam.

Fluid

The third tab on the **Features panel** controls fluid conditions, as shown in Figure 68.

Fluid Type	Select the fluid type to be used. See the Resources/Fluid menu option for a description of the fluid properties.
Fluid Resolution	This field controls the tolerance (as a ratio to resolution) for identification of neighbor fluid nodes when creating the pipe network. Thus, a small Fluid Resolution results in less connectivity of the fluid pipe network while a large **Fluid Resolution** results in a higher connectivity pipe network. Usually, the default value of 0.8 is a good compromise between accuracy and speed.
Cavitation Pressure	This field defines the minimum negative pressure that fluid can sustain and is used mainly for validation purposes. (In the current version, if the non-zero cavitation pressure is specified, the code does not check for cavitation at all.) If zero cavitation pressure is specified, the fluid cannot sustain any tension (or negative pressures).
Pressure initialization	Fluid **Constant pressure** or a **Pressure gradient** (corresponding to the gravitational gradient) can be applied to initialize a fluid pressure in the fractures or to the matrix. The value of the parameter **Reference minimum stress** defined in the Main Rock/Stress tab is applied to the constant pressure value.
Pressure boundary conditions	Specific fluid pressure may be applied to the model boundaries by checking the corresponding boundary and setting the pressure value. The value of the parameter **Reference minimum stress** defined in the Main Rock/Stress tab is applied to the constant pressure value.
Built-in depletion domains	Allows the user to create "built-in" depletion zones. Appendix 1 – Workflow for Modeling of Effects of Reservoir Depletion (Parent-Child Wells) explains how the built-in depletion zones are used.

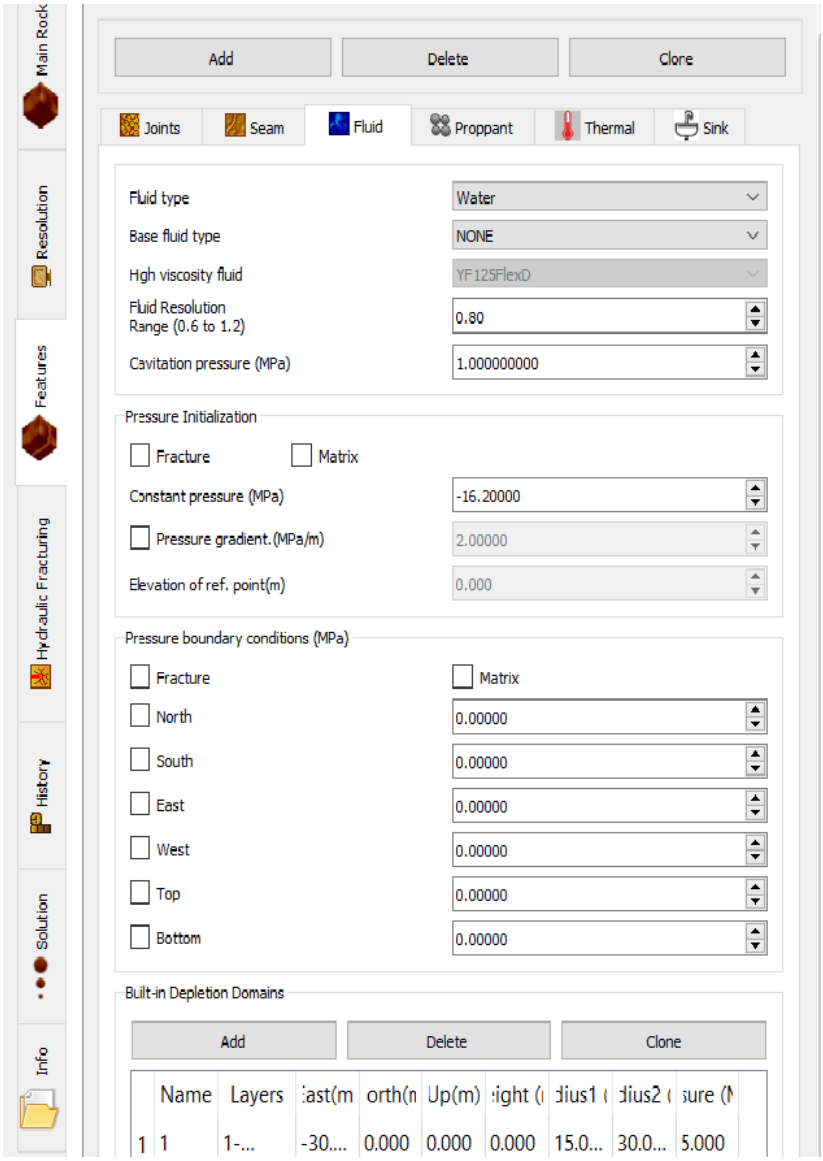


Figure 68 Fluid conditions dialog.

Proppant

The fourth tab on the **Features** panel defines proppant properties, as shown in Figure 68. Proppant is a granular material carried by the fracturing fluid and deposited in fractures to hold fractures open after a hydraulic fracturing treatment (i.e., after fluid pressure dissipation). In addition to naturally occurring sand grains, artificial proppant, such as resin-coated sand or high-strength ceramic materials, may also be used (Schlumberger, 2011) in well treatment.

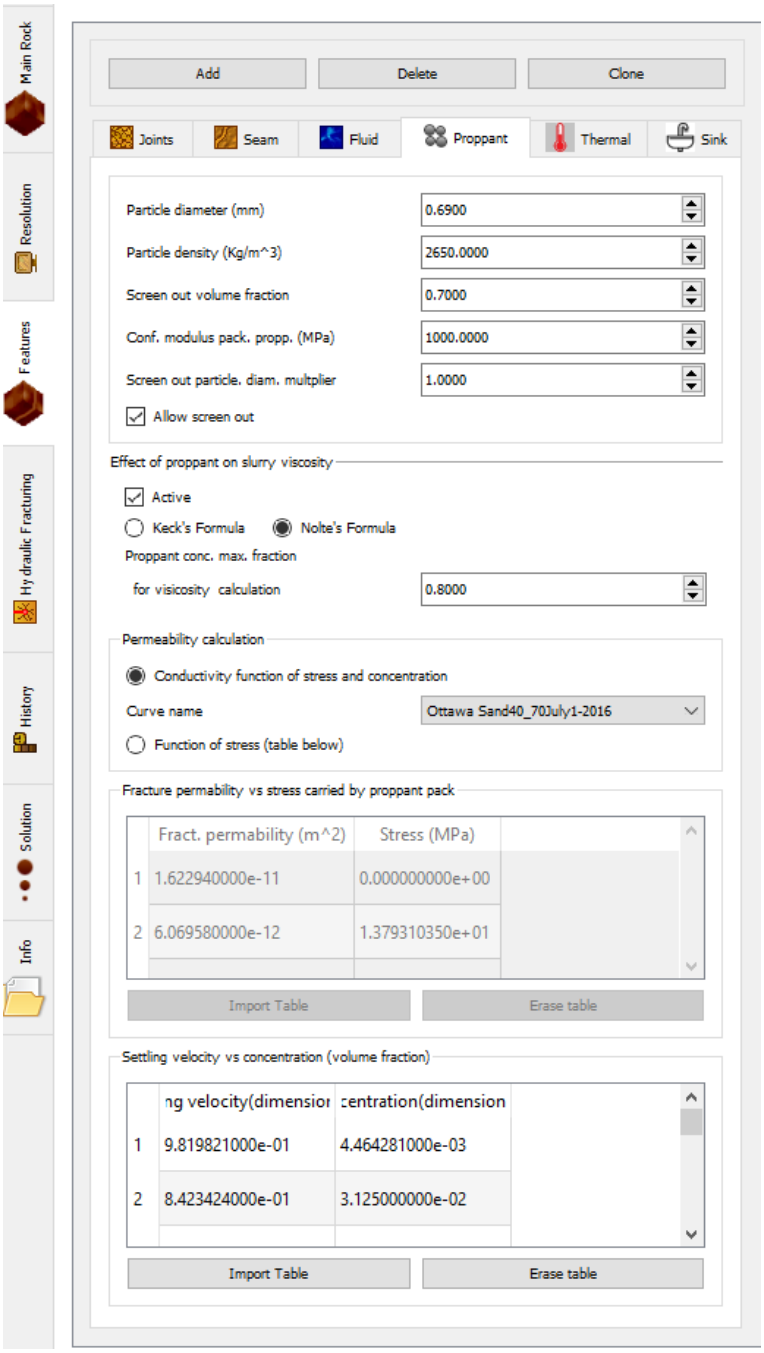


Figure 69 Proppant properties tab.

XSite does not support the use of more than one proppant type in the same model.

Screenout is a condition encountered when the proppant transported by the fluid takes load and restricts the fluid flow. A sudden increase in pressure may occur after screenout. This is an indication that the fluid aperture has been completely packed with proppant. Values between 0.6-0.8 of the volume fraction of proppant within a fracture are reasonable estimates of the condition when screenout occurred. The proppant-related model parameters that may be specified by the user are listed in the following table.

Particle diameter	Proppant particle diameter. Settlement velocity of proppant is a function of proppant diameter.
Particle density	Proppant particle density.
Screenout volume fraction	Maximum value is 1.0. If proppant concentration is greater than this value, the proppant will take load and prevent the fracture to close. The proppant becomes stationary, the permeability changes, and the fluid will flow through the proppant.
Confined modulus of the proppant pack	Confined modulus of the proppant pack is used in calculation of stiffness of the springs that represent propped fractures.
Screenout particle diameter multiplier	Screenout also occurs when the fracture aperture divided by the particle diameter is less than the value specified in this parameter.
Allow screenout	If left unchecked, code will not detect screenout.
Activate effect of proppant on slurry viscosity	If activated, XSite will use either Keck's or Nolte's formula to calculate apparent viscosity of fluid loaded with proppant. If inactive, the fluid viscosity is independent of the proppant concentration. This formula is used only when full flow logic is active.
Keck's Formula	See equation (30) by Keck et al. (1992): $$\mu_r = \{1 + [0.75(e^{1.5n} - 1)e^{-\frac{(1-n)\gamma}{1{,}000}}]\frac{1.25\phi}{1 - 1.5\phi}\}^2$$ where: μ_r = relative fluid viscosity; n = Power Law parameter (flow behavior index); γ = unladen Newtonian shear rate; and ϕ = particle volume fraction.

Nolte's Formula	See equation (8) in Nolte (1988): $\mu_r = \left[1 - \frac{\phi}{\phi M}\right]^{-2.5n}$ where: ϕM = maximum volume fraction.
Permeability calculation	The permeability calculation is performed using the values from the table of fracture permeability vs stress carried by the proppant pack or by one of the conductivity tables available in the **Resources** menu option. This option is available only when full flow logic is active.
Fracture permeability vs stress carried by proppant pack table	See description in the following text.
Settling velocity vs concentration (volume fraction) table	See description in the following text.

The relations between proppant pack stress and conductivity and the relative settling rate to proppant concentration may be defined by two tables: the **Fracture permeability vs stress carried by proppant pack** table and the **Settling velocity vs volume fraction** table.

The **Fracture permeability vs stress carried by proppant pack** table relates proppant permeability to stress carried by the proppant pack, as illustrated in Figure 70. This is the intrinsic permeability of the proppant pack within a fracture. The conductivity is calculated by multiplying the permeability with the fracture aperture.

In the table, no two adjacent values should be equal. If the actual stress is greater than the last (greatest) value in the table, then the conductivity is taken as the final value in the table. If no table is specified, the conductivity is assumed to be 0.0; thus, it is important that a table is specified to allow realistic flow through a propped fracture to be modeled.

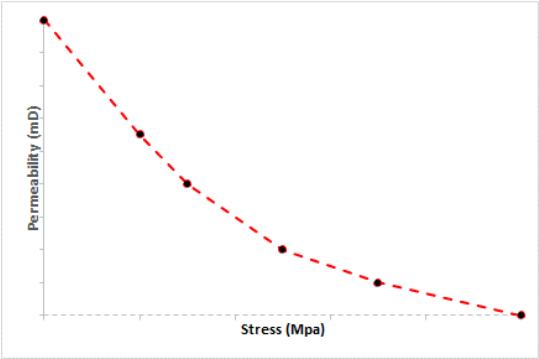


Figure 70 Proppant pack conductivity based on its stress state.

The other option to calculate permeability is to use one of the curves defined in the **Resources/Conductivity curves** menu option. The permeability is calculated by finding an entry in the table using the interpolated proppant concentration. This entry corresponds to a curve that defines the stress-conductivity relation. Using a stress-interpolated value, conductivity is determined.

The **Settling velocity vs volume fraction** table defines the relative settling rate to particle concentration. Both columns are dimensionless; the settling rate is relative to the Stokes' velocity (under gravity) of a particle of given size in a fluid of given viscosity. The particle concentration, c, is the "volume fraction," i.e., the total volume of particles per unit volume of space (crack volume). Note that $c = 1 - n$, where n is porosity and the mass-concentration is equal to $c\rho_p$, where ρ_p is the particle density parameter of proppant. Both concentration and settling rate must be within the range between zero and one. A typical form for the relation is illustrated in Figure 71.

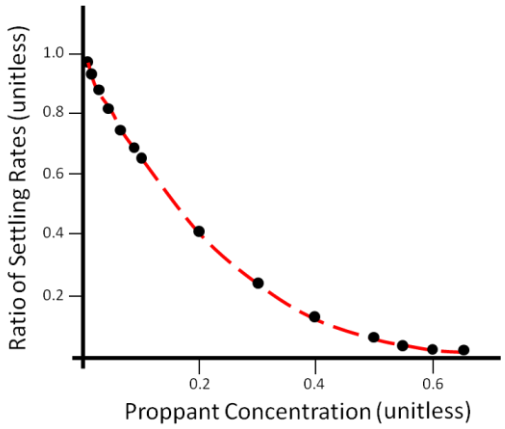


Figure 71 Typical form for the relation between relative settling rate and proppant concentration.

The first entry in the table should be (1.0, 0.0), i.e., the setting rate is equal to the Stokes' rate at a concentration of zero (isolated particles). No two adjacent values of concentration in the table should be equal. If the actual concentration is greater than the last (greatest) value in the table, then the

conductivity is taken as the final value in the table. If no table is specified, the relative conductivity is assumed to be 1.0.

A table of values should be prepared in a text file, which is then imported to XSite. There are two values per row, and a table may have any number of rows. The values in a row must be space-delimited (i.e., separated by spaces); an example of a valid table is shown in Table 2.

For example, in the **Fracture permeability vs stress carried by proppant pack** table, the first column corresponds to conductivity values and the second column corresponds to stress values.

Table 2 Example of Valid Format of Proppant Settling Relative Velocity

1.0	0.0
0.7	0.2
0.5	0.3
0.2	0.5
0.1	1.0

Thermal

The fifth tab on the **Features** panel defines rock and fluid thermal initial and boundary conditions, as shown in Figure 72.

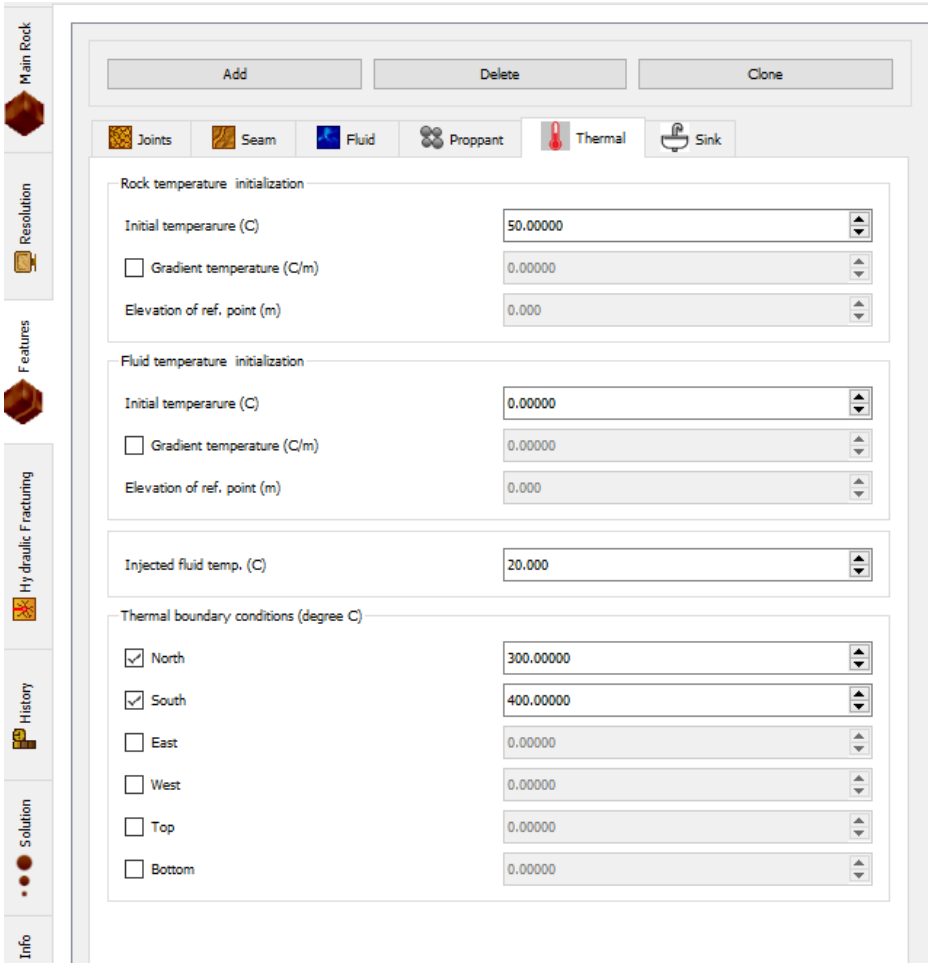


Figure 72 Initial and boundary conditions for thermal analysis.

The initial and boundary conditions for thermal analysis that can be specified by the user are listed in the following table.

Rock initial temperature	Initial rock temperature.
Rock gradient temperature and **Elevation of reference point for rock temperature gradient.**	By checking the **Rock gradient temperature** box, the user can enter a value for the gradient. The temperatures due to thermal gradient will be added to the rock initial temperature value as a function of elevation relative to the reference point. Note that if global coordinates have not been specified, the elevation at the top of the model is 0.0.
Fluid initial temperature	Initial fluid temperature.
Fluid gradient temperature and **Elevation of reference point for fluid temperature gradient.**	By checking the **Fluid gradient temperature** box, the user can enter a value for the gradient. The temperatures due to thermal gradient will be added to the fluid initial temperature value as a function of elevation relative to the reference point. Note that if global coordinates have not been specified, the elevation at the top of the model is 0.0.
Injected fluid temperature	The temperature of the injected fluid.
Thermal boundary conditions	If the user activates a box referent to one of the faces of the model, the temperature will be applied at the selected face.

Sink

The sixth tab on the **Features** panel defines the geometry and the initial conditions of a sink, as shown in Figure 73. A sink must have at least 4 fluid nodes for proper model discretization. This feature allows a user to specify boundary condition for the flow model in addition to the injection cluster. Typically, a constant pressure boundary condition would be applied, approximating the effect of a sink.

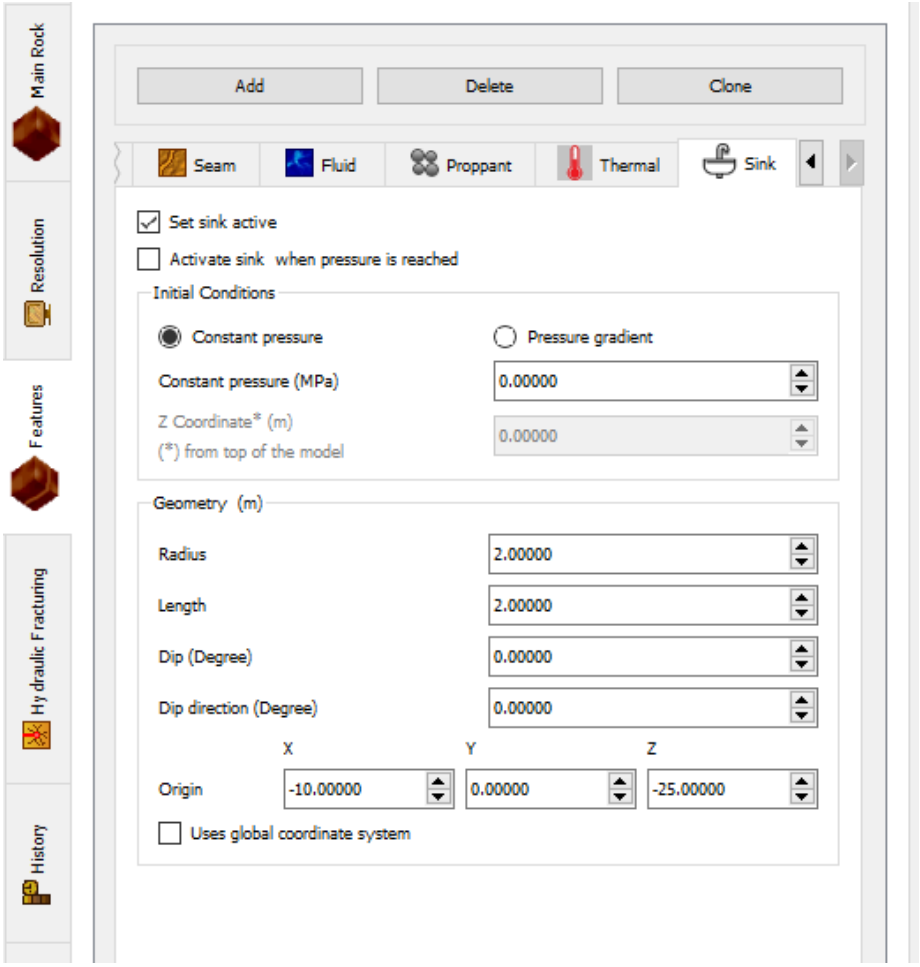


Figure 73 Sink geometry and initial conditions.

The sink initial conditions and geometry that can be specified by the user are listed in the following table.

Set sink active	If this flag is active, the sink will be activated at the start of the simulation. When this flag is set, the sink will be drawn in the **Sketch Model** plot item. The sink may be activated on the Reset Lattice dialog previously discussed.
Activate sink when pressure is reached	If this flag is set, the sink will be set during simulation when the pressure defined in the **Constant pressure** of the initial condition box is reached.
Initial conditions	The user can specify a constant pressure at the sink. If the gradient radio button is selected, a gravitational gradient pressure will be added to the constant pressure value. The *z*-coordinate is referent to the top of the model (value is 0.0) if a global coordinates system has not been applied to the model.
Geometry	The fields in the geometry box define the location and dimensions of the sink. The shape of the sink is cylindrical. If the radius of the sink is set to zero, the fluid element closest to the center of the sink will be selected to model the sink.

Hydraulic Fracturing

The third tab on the simulation **Control panel** allows definition of the boreholes and clusters within the model. It has three sub-tabs: **Trajectory**, **Boreholes** and **Simulation sequence**. The details of these sub-tabs are discussed in the following sections.

Trajectory Tab

The **Trajectory** tab provides tools to facilitate setup of a model for multi-stage operations with complex borehole arrangements. A **Stage** defines a set of clusters on a borehole that will be stimulated simultaneously. Multi-stage operations allow the stages on a borehole to be executed sequentially or in any order in models with multiple wells and stages (e.g., "zipper" pattern).

Workflow to set up a borehole using a well trajectory

A complete borehole with a set of stages and clusters can be assembled from a well trajectory. The workflow to set up a borehole is as follows:

1) Ensure that **Cluster Design** items are created (see menu option *Resources/Cluster Design*).
2) Ensure that **Stage Design** items are created (see menu option *Resources/Stage Design*).
3) Create boreholes. The user can create boreholes in two different ways: using the **Borehole/Segment/Cluster** dialogs or by using **Well Trajectories**.

The option to create a **Borehole/Segment/Cluster** is described in the *Hydraulic Fracturing/Boreholes* tab. The option to create boreholes using well trajectories is described in this workflow. The advantage of using **Well Trajectories** to create boreholes is the fact that the creation of stages and associated clusters, joints for fracture initialization, adaptive resolution domains and cluster histories can be automated. The steps to create boreholes using **Well Trajectories** are as follows (and detailed further in the following sections of this document):

a) Import or edit existing **Well Trajectories**.
b) Associate **Stage Designs** to **Well Trajectories**.
c) Build **Boreholes**: This step will merge a selected **Well Trajectory** item with **Stage** and **Cluster Design** items to create a borehole with segments and clusters.

4) Add **Simulation Sequences** via the *Simulation Sequence* tab.

Setting up well trajectories

Well trajectories can be imported, added, edited, deleted or cloned using the available buttons in the **Hydraulic Fracturing/Trajectory** tab. A list of **Well Trajectories** created in the model or imported is displayed in this tab, as shown in Figure 74. A **Well Trajectory** may have multiple segments and multiple stages associated with it. A segment may be defined by using either True Measure Depth (TMD) or Measure Depth (MD) values. A **Stage** is uniquely named within a well and associated with the **Cluster Design** items.

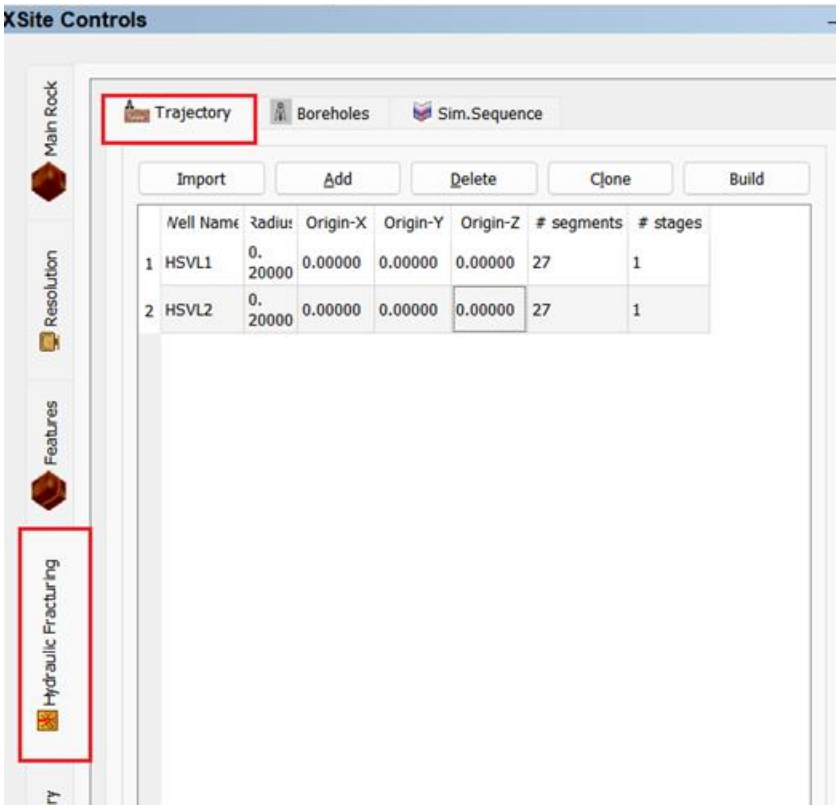


Figure 74 Well Trajectory tab.

Pressing the **Import** button will invoke the **Well Trajectory Import Text Dialog** (Figure 75). This dialog allows a user to import data from a text file in CSV format (**Import** button) or paste cells copied directly from EXCEL or other text files (**Paste** button). The format of the data can be matched with columns available in the program by dragging the buttons into the **True Vertical Depth – Measured Depth Segments** panel. The **Skip** button is used for the cases in which the input data column does not have a matching button in the dialog. For example, the dialog shown in Figure 75 contains data pasted from all seven columns of the spreadsheet shown in Figure 76. Column E in the example file, VS, was paired with the **Skip** option. The other columns, MD, INC, AZI, TVD, N/-S and E/-W, were matched with their respective corresponding pairs. Conversion from feet to meters was applied to the columns MD and TVD. The **Save** button will dismiss the dialog and a new **Well Trajectory** item will be added to the model. The **Cancel** button will dismiss the dialog without saving the data. Note that you may enter either TVD or MD values. In this example, both are entered.

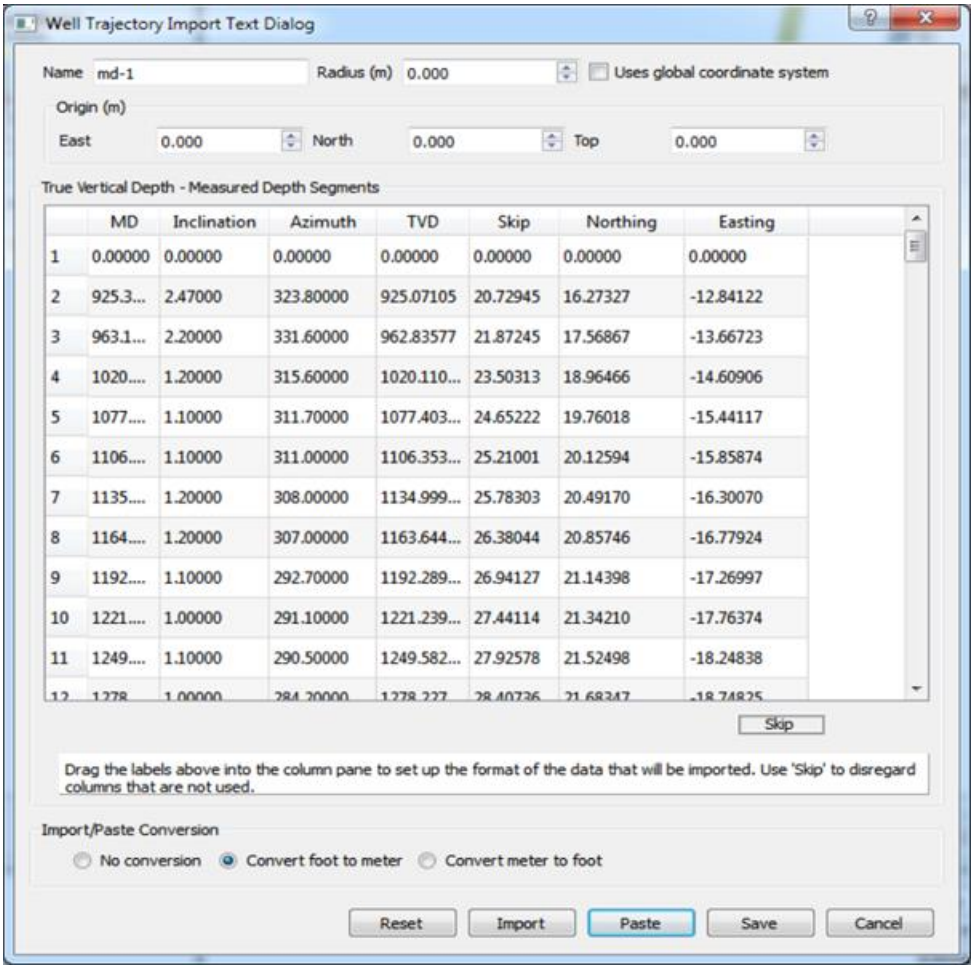


Figure 75 Well Trajectory Import Dialog.

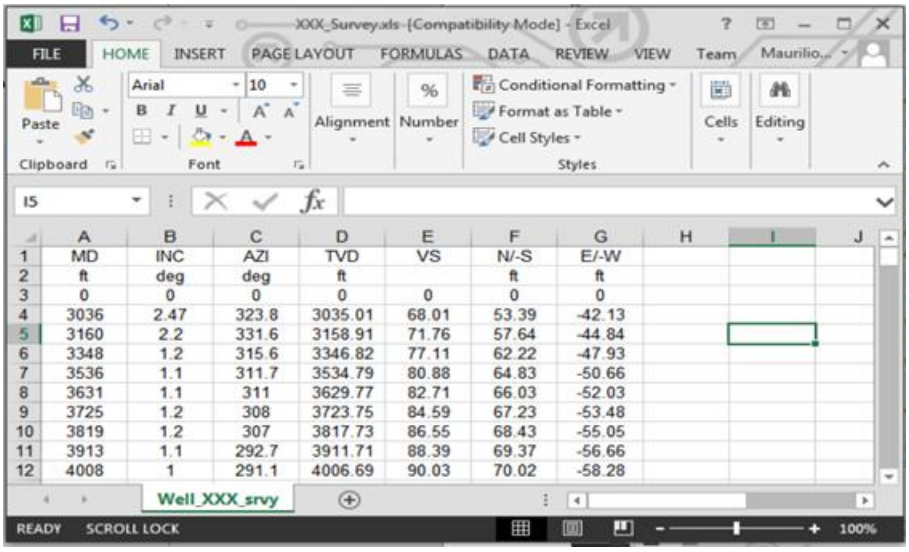


Figure 76 Well trajectory data.

In the **Hydraulic Fracturing/Well Trajectory** tab, displayed in Figure 74, by double-clicking a **Well Trajectory** item or pressing the **Add** button, the **Well Trajectory Dialog** (Figure 77) will be displayed. This dialog allows the user to input or edit the well trajectory information. Also in this tab, the **Clone** and **Delete** operations can be performed on items that are selected previously from the list of available **Well Trajectory** items.

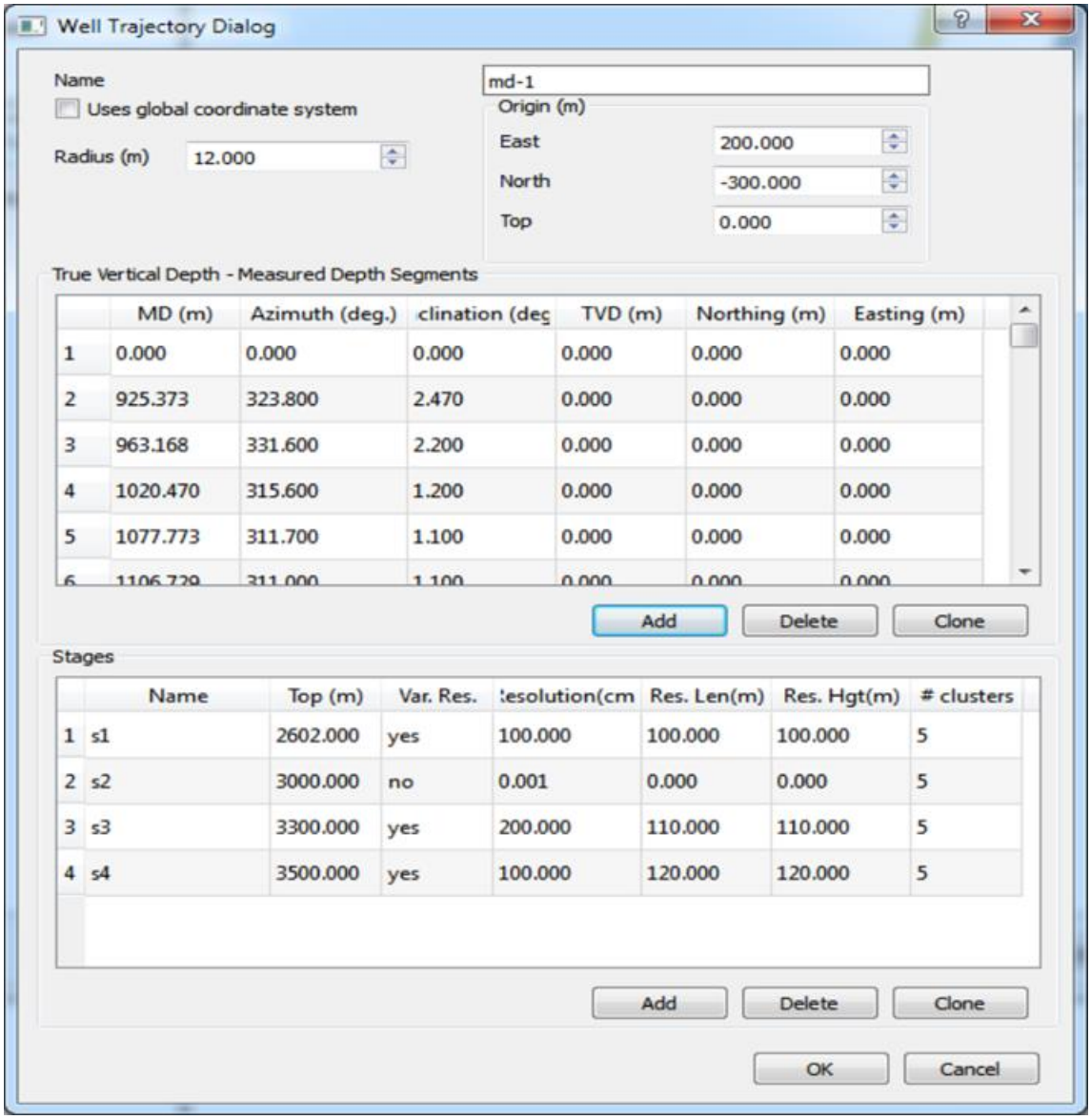


Figure 77 Well trajectory dialog.

In the **Well Trajectory** dialog, the user may provide a name that will uniquely identify the well in the model, its radius and coordinates. By double-clicking an item in the **True Vertical Depth – Measured Depth Segments dialog** or pressing the **Add** button, the user will invoke the **True Vertical Depth – Measured Depth Segment dialog** (Figure 78). In this dialog, it is possible to edit TVD or MD data for a specific segment.

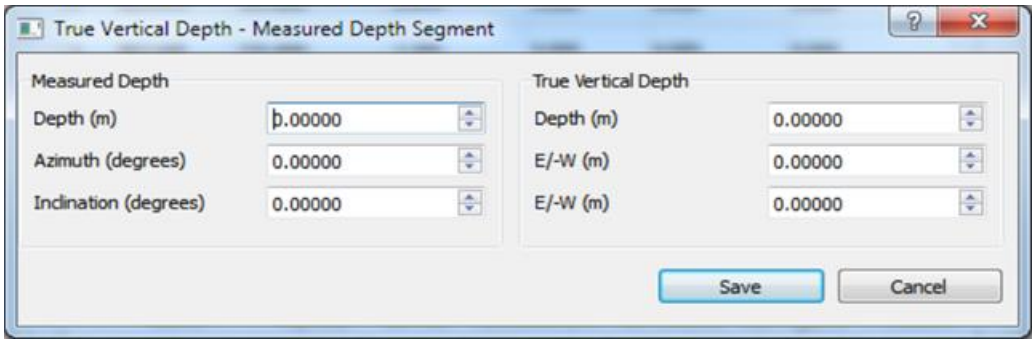


Figure 78 True Vertical Depth – Measured Depth Segment dialog.

Associate Stage Designs to Well Trajectories

A **Stage** defines a sub-set of clusters on a borehole that will be stimulated simultaneously. There can be only one stage active per borehole at a given time. By double-clicking an item in the **Stages** panel in Figure 77 or pressing the **Add** button, the user will invoke the **Stage** dialog (Figure 79). This dialog connects a stage design (previously defined on the **Resources/Stage Design** menu option) with boreholes. Stages that belong to the same well must have unique names.

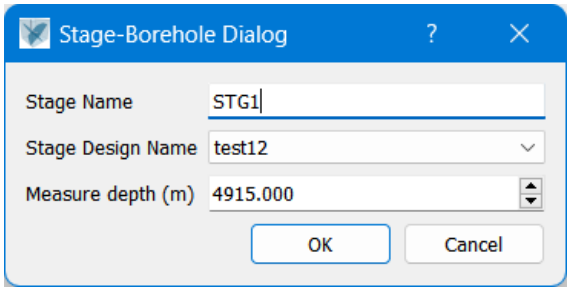


Figure 79 Dialog to define association between well trajectory and stages.

Build Boreholes

By selecting one or more **Well Trajectory** items and pressing the **Build** button in the **Hydraulic Fracturing/Well Trajectory** tab, the user can generate boreholes using the **Build Boreholes** dialog, shown in Figure 80.

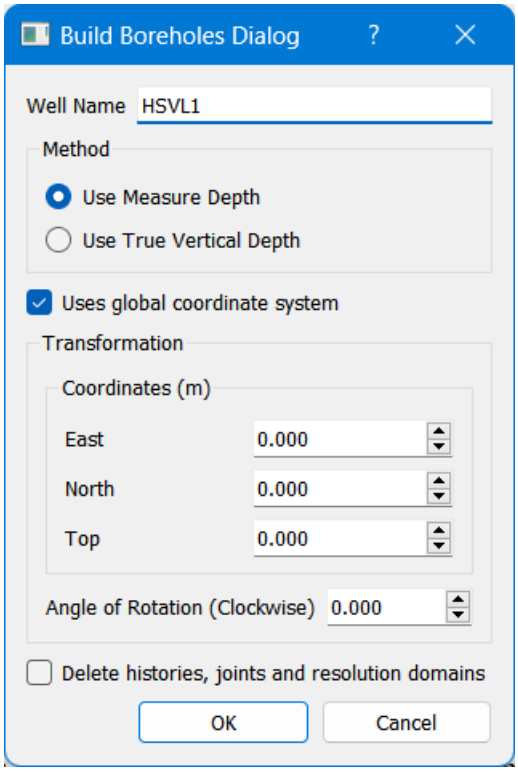


Figure 80 Build Boreholes Dialog.

Depending on the data entered when creating the well trajectory, select either **Use Measure Depth** or **Use True Vertical Depth** and click OK to create boreholes. The corresponding borehole, borehole segments, stages and clusters will be created. Transformation to global coordinates is available. Variable resolution domains, joints and cluster histories may be created also if the **Well Trajectory** item had been specified previously with these options. Existing histories, joints and resolution domains will be deleted if the checkbox in the dialog is selected.

Use the **Sketch Model** plot item to verify if the borehole and other items (histories, joints, variable resolution domains, etc.) were created properly.

Boreholes Tab

Boreholes are comprised of segments. Each segment can have a set of clusters for which variable injection rates and proppant concentrations over time can be specified. New boreholes can be added to the simulation by clicking the **Add** button (Figure 81), which will bring up the **Borehole dialog** box, as shown in Figure 82. Existing boreholes can be modified by double-clicking a particular borehole, as listed in Figure 81, or removed by selecting the borehole and clicking the **Delete** button. **Clone** button will copy the currently selected borehole.

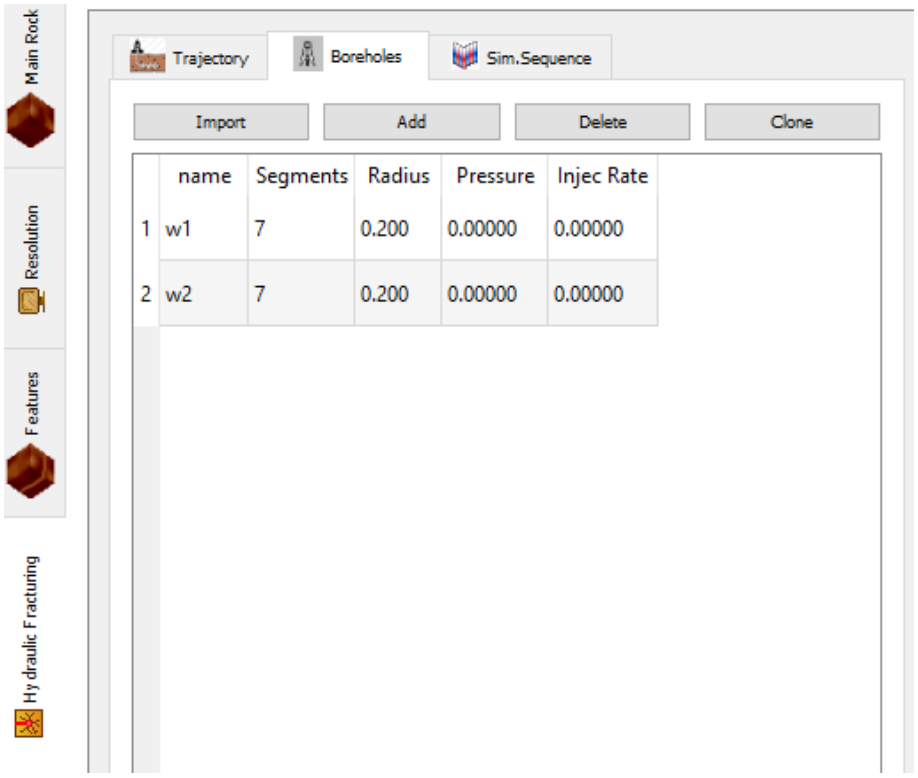


Figure 81 Boreholes can be added to the model or existing ones viewed and modified, deleted or copied (cloned) under the Hydraulic Fracturing tab.

When adding or modifying a borehole, the **Borehole editor dialog** (Figure 82) will become visible. This dialog has 3 main regions: borehole **Properties**, **Segments** and stage **Injection rate schedule**.

Borehole Properties

Enter a unique name for the borehole. The field **Uses global coordinate system** will work as described in the **Joint Set** section. The borehole **Radius** field must be specified, whereas the **Injection rate** field may be specified. If injection rate is specified, it is applied as constant rate during the simulation and the borehole logic is not used in the computation. If the injection schedule data is entered, the borehole **Injection rate** field is disabled.

When the field **Apply proppant directly into clusters** is checked, the proppant concentration will be obtained from the injection schedule and applied directly into the cluster. Otherwise, the borehole logic will calculate the proppant concentration that will be applied into each borehole segment taking into consideration pressure loss. This flag has no effect when the code runs fluid simulation using the simplified toughness-dominated regime.

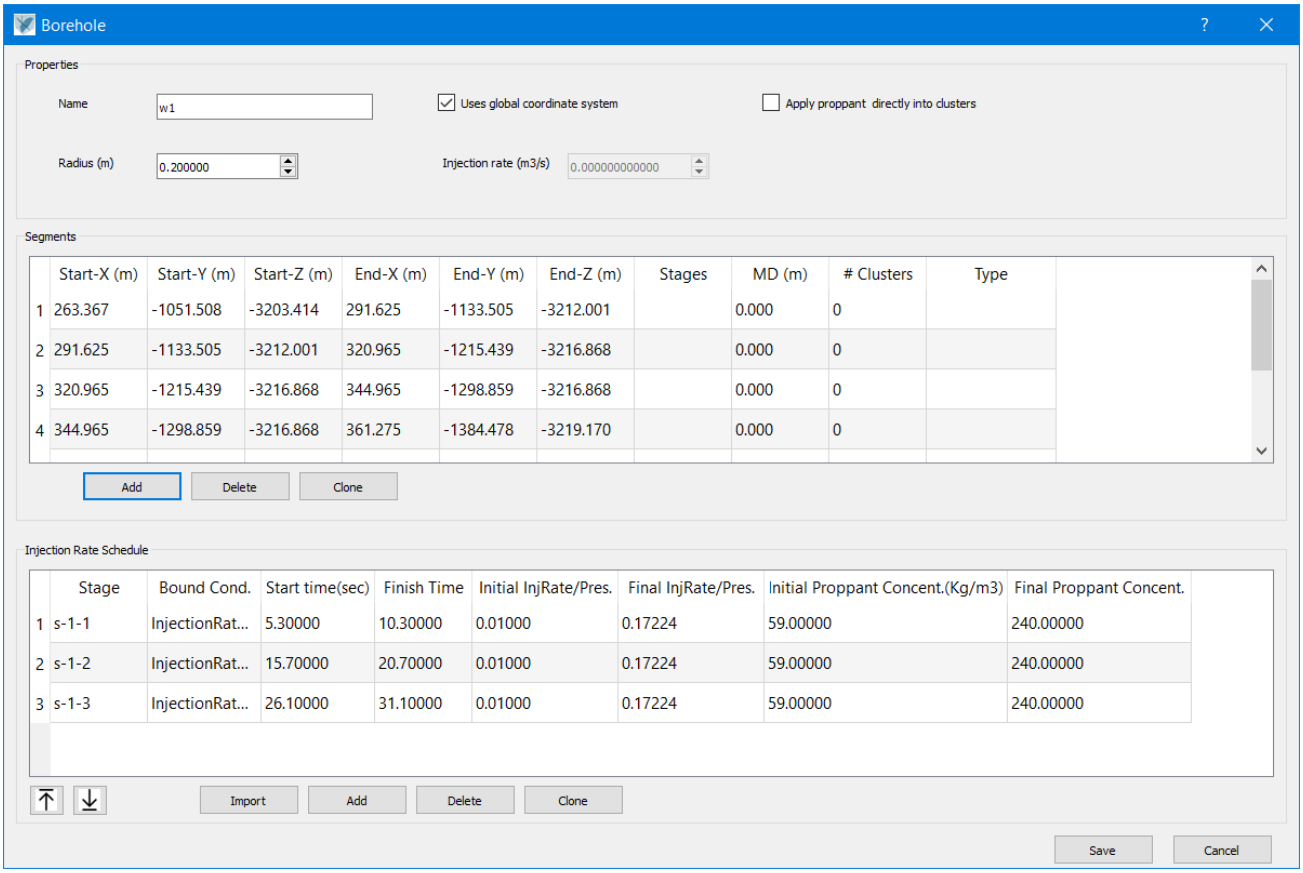


Figure 82 Borehole editor dialog.

Borehole Segments

Boreholes can consist of multiple segments that can be created by clicking on the **Segments Add** button (Figure 83). Segments also can be edited, deleted or cloned by selecting a segment listed and clicking on the appropriate button. When adding or editing a borehole segment, the start and end position of the segment are required (Figure 72).

If casing is used inside the borehole, the **Casing wall thickness** and the **Liner resolution** must be entered. The liner resolution should be at least 3 times smaller than the casing wall thickness to have the casing properly discretized. In addition, a set of domain resolution regions enveloping the borehole also must be added to gradually transition the liner resolution to the background resolution.

The recommendation is to apply the liner (cement and/or casing) after the model has reached mechanical equilibrium after borehole core removal (i.e., borehole excavation). These operations are available on both the Reset Lattice dialog and the **Batch Item dialog**.

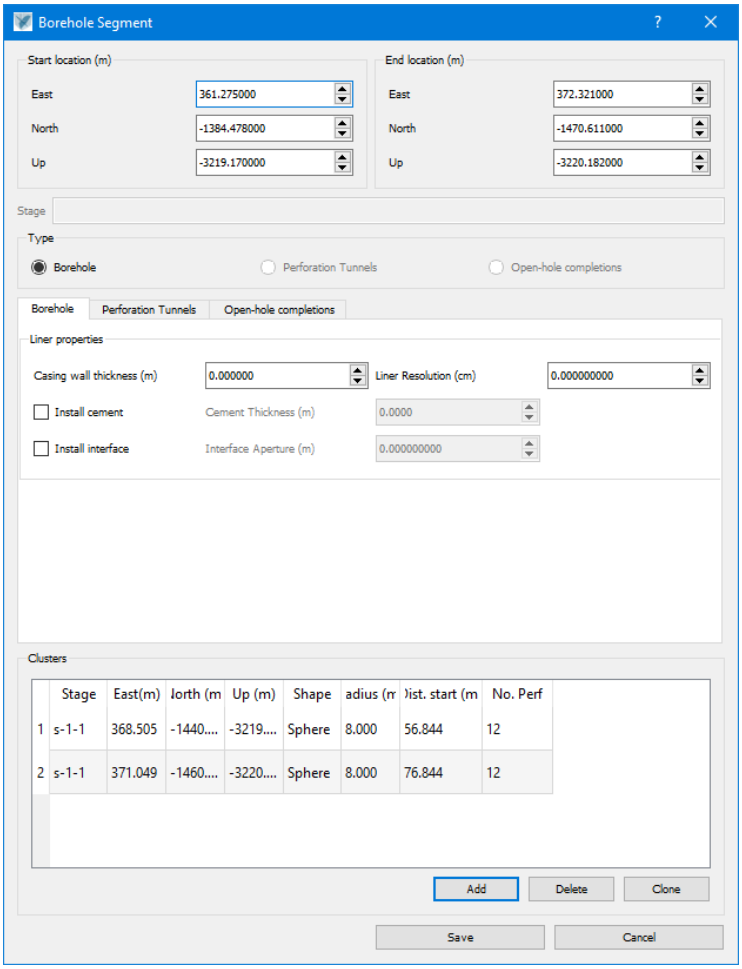


Figure 83 Borehole segment editor defines the borehole length and position based on its start and end locations.

A segment may be selected as **Open-hole completion** or **Perforated tunnels**. The **Sketch mode** plot item will distinguish the segment base on the selection made on these fields. In the case the segment type selected is **Perforated tunnel**, the radius, number of tunnels per meter, the length of the tunnels and the shift in degrees between consecutive tunnels (**Phasing**) must be specified.

The perforated tunnels and open-hole completion core should be removed when the model is in mechanical equilibrium. These operations are available in the Reset Lattice dialog and in the **Batch Item** dialog. When the core is removed, fluid nodes will be installed on the rock surface inside perforated tunnels and open-hole completions. To have proper discretization of the model, make sure that the resolution around the perforated tunnels is sufficient to represent stress gradients around the perforation tunnels (i.e., at least four times less than the tunnel diameter). See section Add domain resolution around perforated tunnels on generation of a graded resolution lattice around the perforation tunnels.

Clusters

One or more optional clusters (i.e., injection points along the borehole segments) can be defined in a segment by clicking the **Cluster/Add** button and using the **Cluster editor dialog** (Figure 84). As with other elements, **Clusters** can be edited, deleted or cloned.

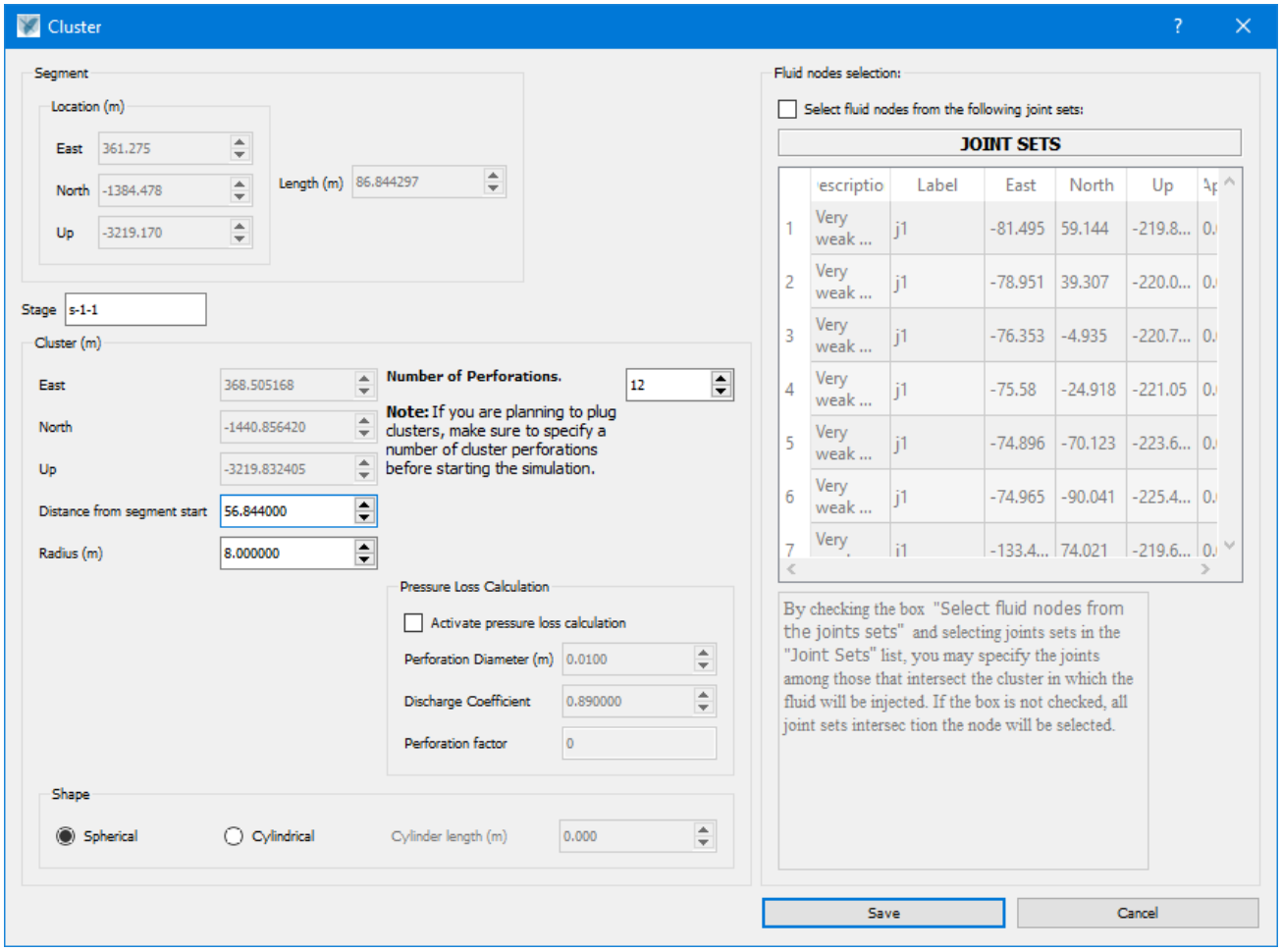


Figure 84 Cluster editor dialog.

The **Stage name** must be entered. This field will be inserted in the drop-down box of the Injection Schedule dialog, allowing to associate stages with injection schedules.

A cluster position is specified relative to the start of the segment, and its absolute coordinates (**East**, **North**, **Up**) will be calculated and displayed. A cluster may be spherical or cylindrical. Radius and length for cylindrical clusters must be specified. The **Pressure Loss Calculation** fields are used to calculate additional pressure losses due to the presence of perforations and non-Darcy effects. The fields **Number of Perforations** and **Discharge Coefficient** are dimensionless, and the field **Perforation Diameter** may be entered either in inches or in meters, depending on the unit system selected.

If the **Select fluid nodes from** checkbox is checked, fluid will be injected only in the fluid nodes that are intersected by the selected joint that intersects the cluster. If this box is not checked, the nodes from all joints will be marked to receive fluid.

The **Save** button must be clicked to preserve any changes; alternatively, any changes can be cleared by clicking the **Cancel** button.

Injection Rate Schedule

Injection rate schedule can be specified by pressing the **Injection Rate/Add** button or can be edited by double-clicking an existing **Injection schedule** item. These actions will cause the **Stage Boundary Conditions** (Figure 85) to be displayed. The user must enter a stage name or select one of the stages of the drop-down box that were previously defined in the Cluster dialog. The data entered in this dialog will be applied defining an initial and final injection rate or initial and final pressure, depending on the condition selected and for a time interval defined in the dialog. Optionally, an initial and final proppant concentration may also be entered for the same time interval.

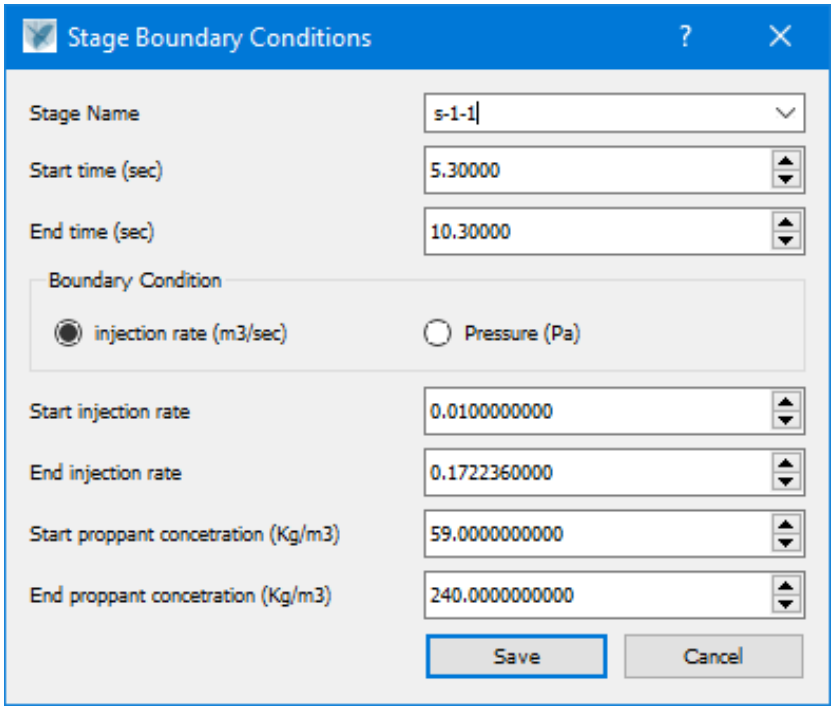


Figure 85 Injection rate dialog used to construct more complex injection rate schedule.

Simulation Sequence Tab

Simulation Sequence items associate existing stages with **Pumping Schedule** items and **Variable Resolution domains**. This association is used to create batch records and the borehole injection schedule. The **Simulation Sequence** also defines the order in which the different stages will be stimulated (batch execution sequence) in the model.

The **Simulation Sequence** dialog can be used to do the following:

1) Establish the order in which stages of the same or different wells will be stimulated.
2) Associate **Pumping Schedule** items to borehole stages. **Injection Rate** items will be added automatically to the boreholes defined in the model.
3) Create batch items to execute the **Adaptive Resolution** logic during the stimulation. The resolution in defined domains can be modified at different points during the simulation.
4) Create batch items to run the model to equilibrium and reset pressures and node displacements.
5) Create batch items that will run the model in coupled fluid-mechanical-thermal mode for a determined period.

Figure 86 shows the **Simulation Sequence** tab. Simulation sequence items simplify how the steps of a simulation are specified. The simulation sequence operation will automatically add batch items and create an injection schedule for a complete simulation.

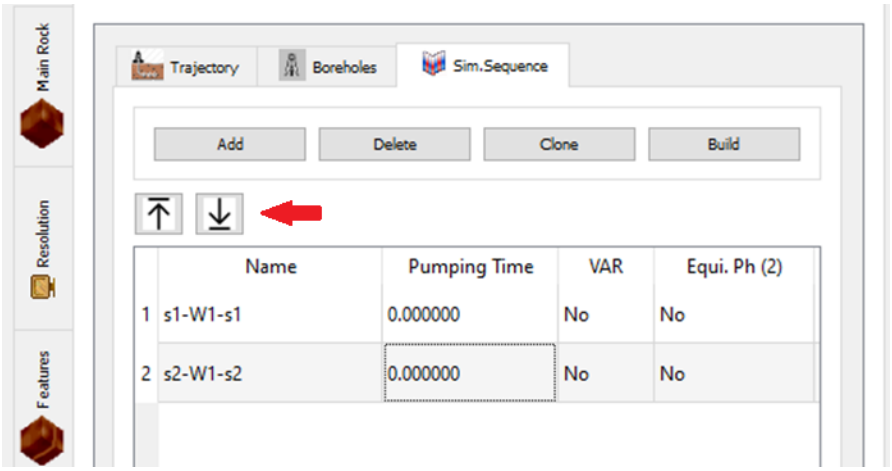


Figure 86 Simulation Sequence tab.

Before creating **Simulation Sequence** items, make sure that **Boreholes**, **Stages**, **Pumping Schedule** and **Variable Resolution domains** are created in the model. In the **Hydraulic Fracturing/Simulation Sequence** tab, the user may edit (by double-clicking an item), add, delete or clone a **Simulation Sequence** item.

The simulation sequence is defined by the order in which the items appear in the list, with the top items executed first. By selecting an item and pressing one of the keys (up or down), indicated in red in Figure 86, the item will move up or down in the list, changing its execution order.

When the user presses the **Add** button, the dialog shown on Figure 87 will be displayed. The dropdown window contains predefined simulation sequence items previously defined via Resources/Simulation Sequence Design. At this point, the user must have defined boreholes, stages/clusters and resources for pumping schedule and simulation sequence.

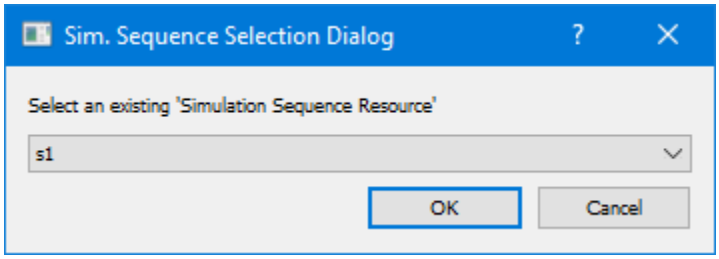


Figure 87 Simulation sequence predefined item selection dialog.

It is recommended to always use predefined simulation sequence items (see Resources/Simulation Sequence Design menu option) to avoid reentry data, and, consequently, reduce the possibility of introducing errors. If the user presses the **OK** button, the **Simulation sequence dialog** shown in Figure 88 will be displayed. The dialog will be populated with the data from the selected predefined simulation sequence. If the user presses the **Cancel** button, the dialog will not be populated. This dialog can also be invoked by double-clicking one of the existing items in the list.

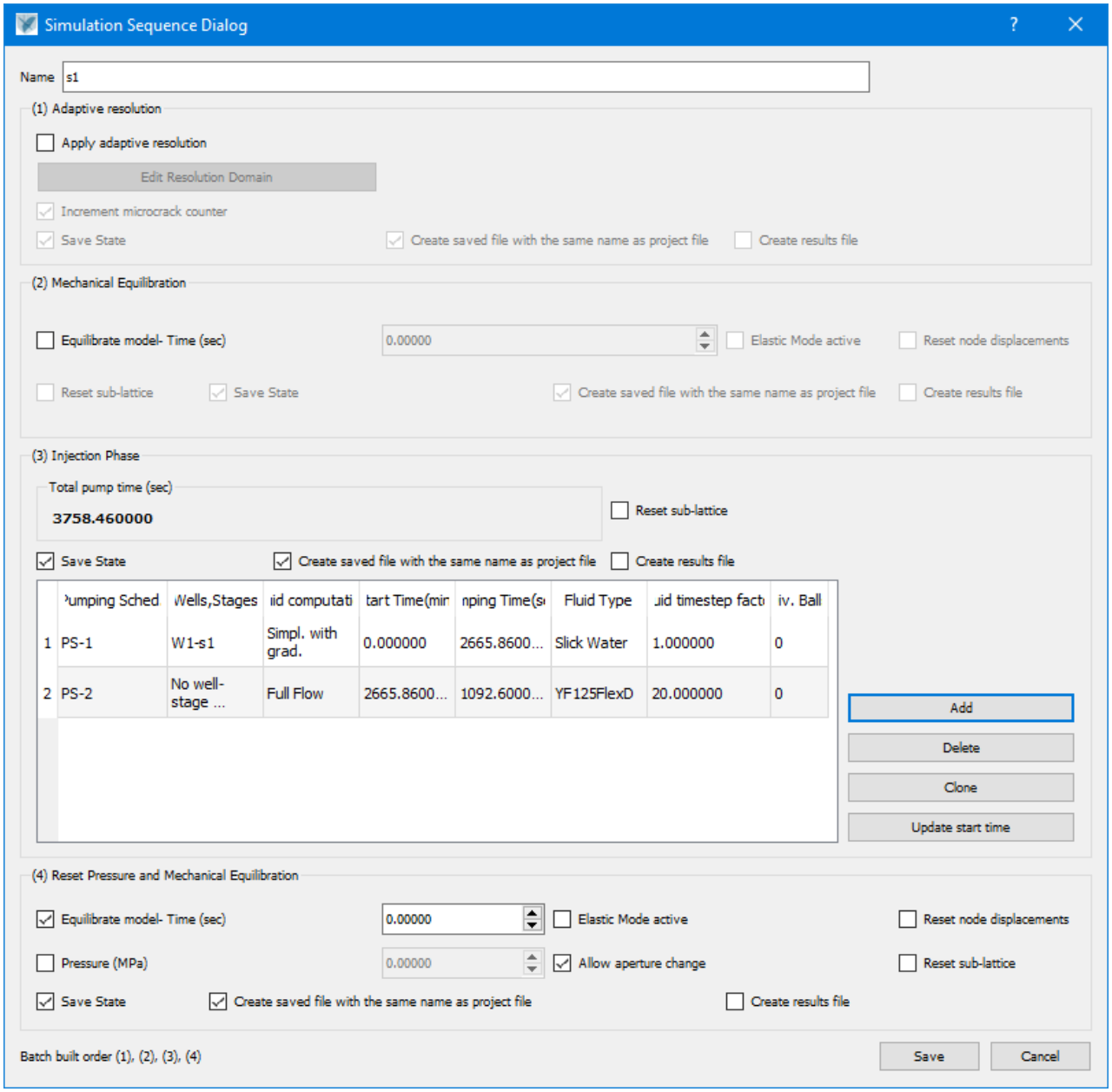


Figure 88 Simulation Sequence Dialog.

Each simulation sequence item in the list will be used to create the necessary steps to simulate one stage. The **Simulation Sequence** has four main groups: **(1) Adaptive resolution**, **(2) Mechanical equilibration**, **(3) Injection Phase** and **(4) Reset pressure and mechanical equilibration**. The groups need to be activated to be included in the generated batch items.

The group **(1) Adaptive resolution** will create a new set of resolutions to solve that stage. The setup of **Adaptive resolution** is explained in section Tools/Batch Simulation/Apply adaptive resolution. For this group to be included in the batch setup, the check box **Apply adaptive resolution** must be checked.

The group **(2) Mechanical equilibration** will add one batch item to perform mechanical equilibration according to the time entered by the user. For this group to be included in the batch setup, the check box **Equilibrate Model** must be checked.

The third group **(3) Injection Phase** associates the pumping schedules from resources to a stage. The pumping schedule injection time is used to set the simulation time on the batch items. The user needs to establish a relationship with the pumping schedule and the stage that the pumping schedule will be applied to. This is done by double-clicking the desired record in group (3). This action will invoke the dialog shown in Figure 89. This dialog is also invoked if you press the **Add** button located in this group.

In the well/stage-pumping schedule dialog, a previously defined pumping schedule is selected in drop box (1). When you select the pumping schedule, the values from column (3) are populated. The values in column (4) (pumping time, fluid type, and fluid time step factor) can be entered or automatically filled by pressing the **Apply Default** button.

After the pumping time is defined, press the **Add** button (5) to create a link between the well and the stage in which the fluid will be applied. By clicking the **Add** button or double-clicking an item in the **Well/Stage** selection, the dialog box shown in Figure 90 will appear to select a well and a stage from the drop-down list. Field (6) allows to include an option to add diversion balls in the pre-processing tab of the batch item that will be created for this item. Finally, the **Fluid Simulation** mode is selected in box (7).

The **Start Time** field of a borehole **Injection Rate** is the summation of all injection (final time – start time) and equilibration times of all previous **Simulation Sequence** stages. Once the start time and final times are defined, the **Injection** items defined in the **Pumping Schedule** item will be copied to the borehole **Injection Rate** list until the total simulation time defined in the **Simulation Sequence** dialog is achieved. The associated **Stage** is used to define the stage in the borehole **Injection Rate**. One or more **Stages/Pump Schedule** items can be used in a **Simulation Sequence** to generate the borehole injection rate list.

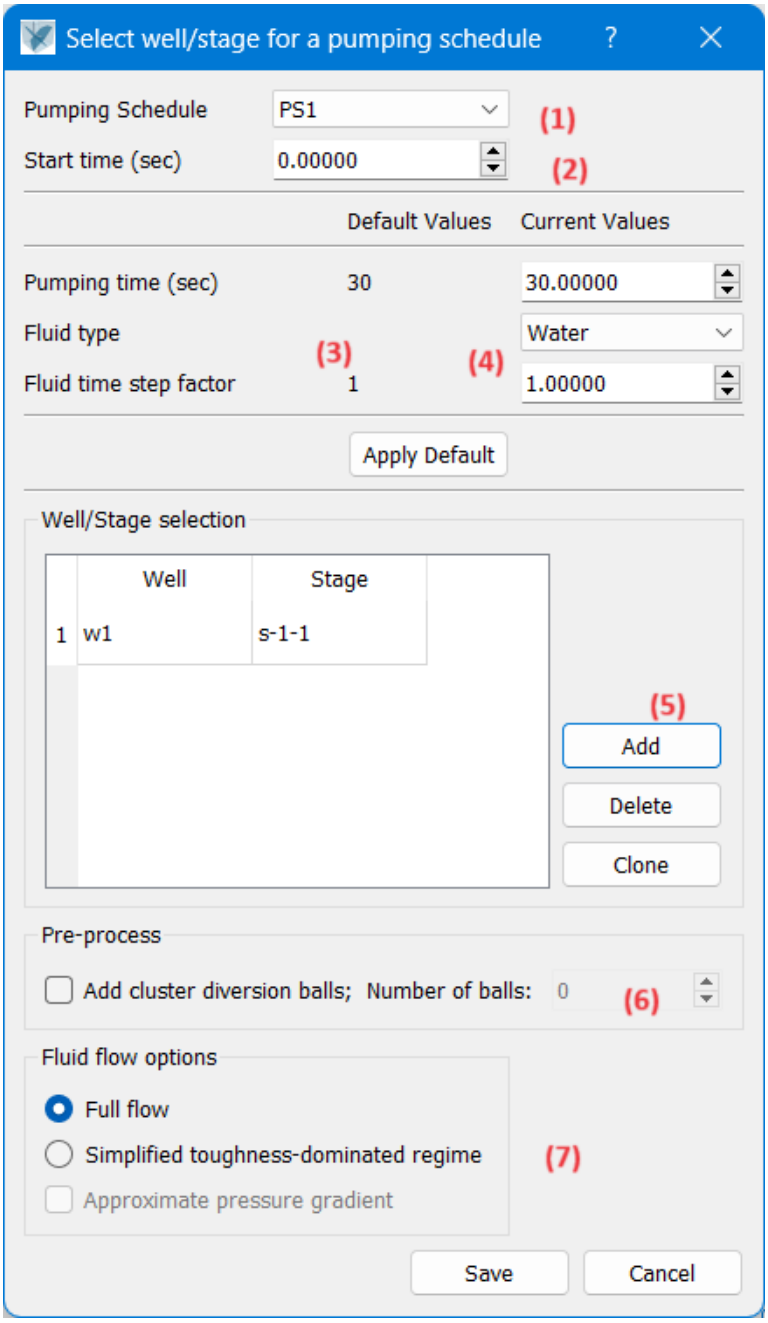


Figure 89 The dialog to create a relation between a pumping schedule and a stage.

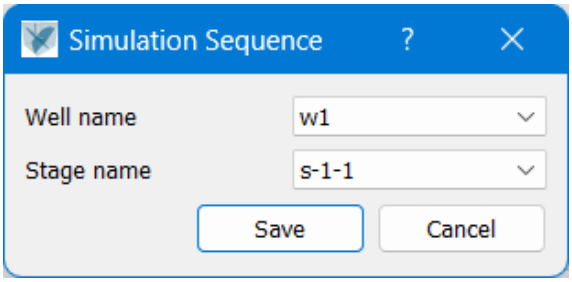


Figure 90 Select a well and a pumping schedule.

Buttons **Delete** and **Clone** will delete or clone the selected item.

The **Build** button will use the information entered in the **Simulation Sequence Items** and will create pumping schedule and batch items. You do not need to select a **Simulation Sequence Item**; all items in the panel will be used to generate the data. A dialog **Yes/No/Abort** will be prompted. If you select **Yes**, all the current pumping schedules for all wells and all the batch items will be deleted. If you select **No**, the new items will be appended to the current ones. **Abort** will exit the operation without processing. You should check the **wells** tab on the **Borehole** tab and the **Batch items** dialog to verify the data were properly created.

Note that batch items and pumping schedule items generated via the **Built** button will be added according to the order of the simulation sequence items list.

History Tab

The fourth tab on the **Control panel** allows history records to be added, removed or cloned (Figure 91). Histories are records of select variables that may be plotted as graphs at any stage during a simulation. The required histories must be specified *before* starting a simulation. Each history is requested by clicking the **Add** button, which brings up the dialog box shown in Figure 92. Each history record added will be displayed in the history list shown in Figure 91. A record in the history list may be edited by double-clicking the item. History records for **Number of Cracks**, **Fluid Time Step** and **Stage Unbalanced Volume** are added by default in a model. The field **History Sample Interval** will define how often the history record will be recorded.

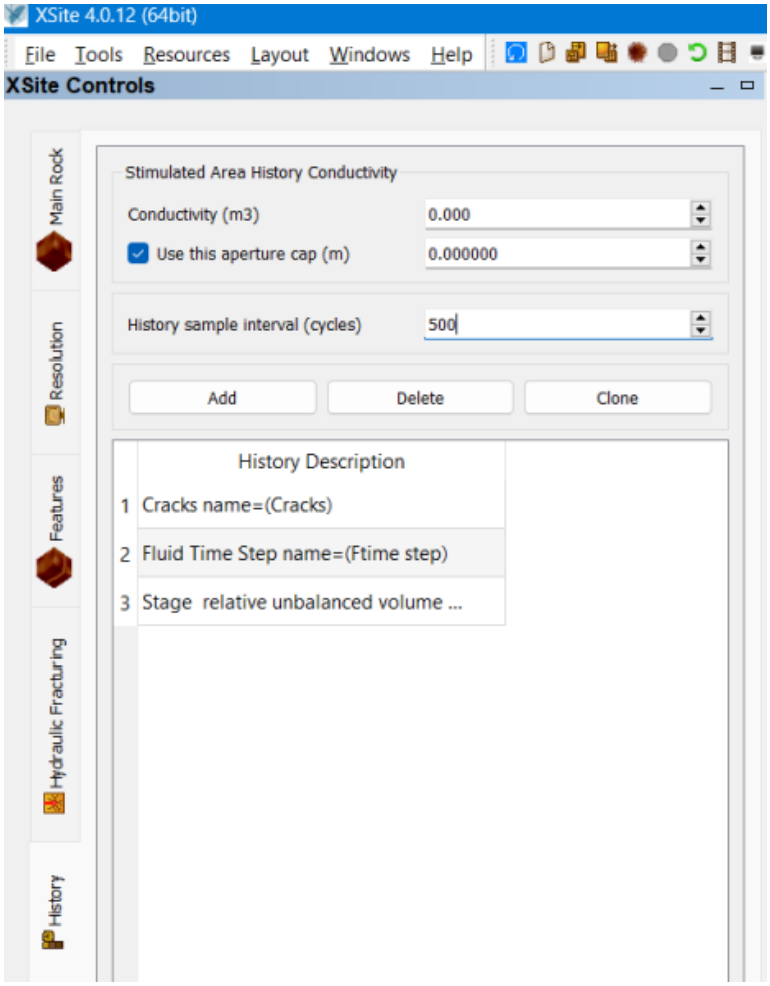


Figure 91 Histories can be added, or existing ones modified under the History tab.

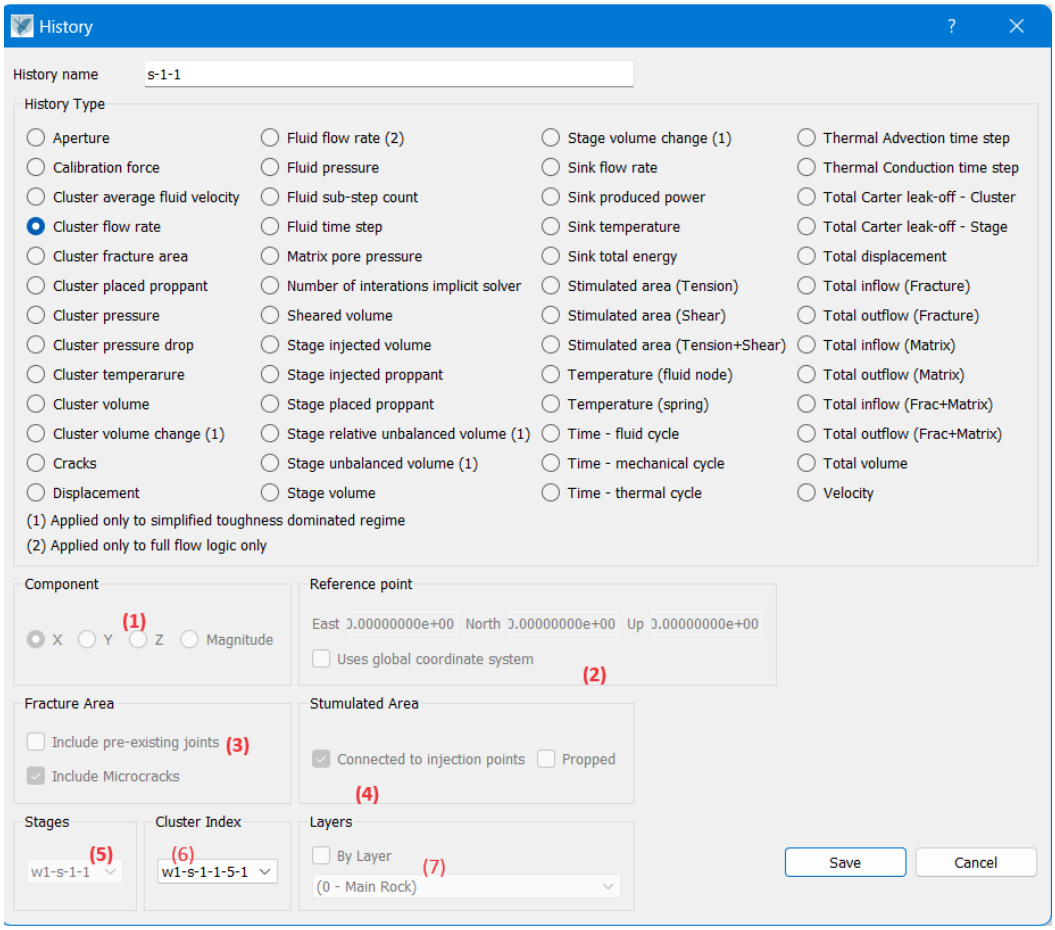


Figure 92 History dialog allows users to specify types and other properties of histories to be recorded.

Description of history types

The following history types can be recorded and displayed during the simulation (or viewed after the simulation is complete). Depending on the history type, a specific box (Figure 92) numbered (1) to (7) will be enabled, allowing the user to specify the correct history property. Note that the **History Name** is optional, but it is useful to give a descriptive name so that it will be easier to select a history type when plots are generated.

Aperture	Record the fluid element aperture over the duration of the simulation. The location of the fluid element must be specified in box (2). Note that the fluid element may not exist at the time the model is initialized, and it will be assigned when the fluid network has reached the distance of one resolution from the specified coordinate.

Calibration force	The total reaction force in a calibration. This history is valid only when the model is running in calibration mode (the check box Calibration Mode in the **Main Rock/Stress Calib.** tab is checked).
Cluster average fluid velocity	This history is recorded only when the simulation is running using the toughness-dominated regime. The average velocity of the fluid inside the selected cluster is recorded. The cluster is identified in box (6) by the borehole, segment and cluster ID.
Cluster flow rate	This history is recorded only when the simulation is running using the full-flow logic. Records the fluid flow rate by adding the positive and negative flow of the pipes that intersect the boundaries of the selected cluster. The cluster is identified in box (6) by the borehole, segment and cluster ID.
Cluster fracture area	Records the history of the fracture surface area in the model. The user can check in box (3) if pre-existing joints and/or microcracks created during the simulation are to be included. In box (7), a specific layer can be specified if the **By Layer** checkbox is selected, otherwise the history is recorded for the entire model. The cluster is identified in box (6) by the borehole, segment and cluster ID.
Cluster placed proppant	Records the accumulated proppant volume multiplied by the proppant density of the fluid nodes connected to the specified cluster over time. The cluster is identified in box (6) by the borehole, segment and cluster ID.
Cluster pressure	Records the average pressure of the fluid nodes inside the specified cluster over time. The cluster is identified in box (6) by the borehole, segment and cluster ID. Cluster pressure history can give an idea of how the simulation is evolving over time. For example, the user should see a curve with a breakout pressure at the beginning of the injection phase then gradually decreasing if the fracture propagates without restraint throughout the remainder of the injection phase.
Cluster pressure drop	Records the average pressure drop (i.e., difference in the pressure inside and outside of the cluster) for the specified cluster. Saturation is taken into consideration. The cluster is identified in box (6) by the borehole, segment and cluster ID.

Cluster temperature	Records the average fluid temperature of the fluid nodes inside the selected cluster. The cluster is identified in box (6) by the borehole, segment and cluster ID.
Cluster volume	Records the volume of the fluid nodes inside the selected cluster. The cluster is identified in box (6) by the borehole, segment and cluster ID.
Cluster volume change	This history is recorded only when the simulation is run using the toughness-dominated regime. The cluster volume change is recorded. The cluster is identified in box (6) by the borehole, segment and cluster ID.
Cracks	Records the total number of cracks in the model. Only one crack history may be added to the model.
Displacement	Records the relative displacement of a mechanical node at a given location (East, North, Up). The component (X or Y or Z) or magnitude of the displacement vector must be specified in box (1). The location of the node must be specified in box (2). Relative displacement is the nodal displacement since the last reset node displacement command.
Fluid flow rate	This history is recorded only when the simulation is run using the full-flow logic. The flow rate of a fluid node located nearest to a given location (East, North, Up) is recorded. The location of the node must be specified in box (2). Note that the fluid element may not exist at the time that the model is initialized, and it will be assigned when the fluid network has reached the distance of one resolution from the specified coordinate.
Fluid pressure	The fluid pressure of a fluid node located nearest to a given location (East, North, Up) is recorded. The location of the node must be specified in box (2). Note that the fluid element may not exist at the time that the model is initialized, and it will be assigned when the fluid network has reached the distance of one resolution from the specified coordinate.
Fluid sub-step count	This history is recorded only when the simulation is run using the full-flow logic. Records the fluid sub-step count (i.e., the number of fluid flow steps per one mechanical step) providing indication if the fluid calculation is properly converging.

Fluid time step	Records the fluid time step. Only one **Fluid Time Step** history may be added to the model.
Matrix pore pressure	Records the matrix pore pressure at the spring nearest to a given location (East, North, Up). The location of the node must be specified in box (2).
Number of iterations of the implicit solver	Records the number of iterations that the implicit solver is taking to converge per cycle. The minimum and maximum number of introductions are defined by the user in the Solution/Fluid tab. If the number of iterations is equal to the maximum number interactions for a long period of time, the model may not be properly converging, and the user needs to reduce the fluid time step multiplier.
Shear Volume	Accumulates the volume of the rock where shearing of joints has occurred.
Stage injected volume	This history is recorded only when the simulation is running using the full-flow logic. Records the fluid volume injected into the selected stage. The stage is identified in box (5).
Stage injected proppant	Records the proppant injected in all clusters connected to the selected stage, based on the injection schedule, The stage is identified in box (5).
Stage placed proppant	Records the accumulated proppant volume multiplied by the proppant density of the fluid nodes connected to all clusters within a selected stage over time. The stage is identified in box (5).
Stage relative unbalanced volume	Tracks the relative difference between the volume of the fluid node apertures for all clusters connected to the selected stage and the fluid volume injected into the stage. This history is recorded only when the simulation is run using the simplified logic for the toughness-dominated regime. The stage is identified in box (5).
Stage unbalanced volume	Tracks the difference between the volume of the fluid node apertures for all clusters connected to the selected stage and the fluid volume injected into the stage. This history is recorded only when the simulation is run using the simplified logic for the toughness-dominated regime. The stage is identified in box (5).

Stage volume	Records the volume of the fluid nodes connected to the clusters within the selected stage. The stage is identified in box (5).
Stage volume change	This history is recorded only when the simulation is running using the simplified logic for the toughness-dominated regime. The volume change of all fluid nodes connected to the clusters within the selected stage is recorded. The stage is identified in box (5).
Sink flow rate	Display the total flow rate of the pipes connected to the fluid nodes located inside the sink.
Sink produced power	Integrates and accumulates the produced power of the fluid extracted by the sink over the duration of the simulation.
Sink temperature	Displays the average temperature of the fluid elements inside the sink.
Sink total energy	Accumulates the total energy carried by the heated fluid extracted from the sink over the duration of the simulation.
Stimulated area in tension	Tracks the total surface of the microcracks created in tension. In box (4), it is possible to differentiate and monitor the stimulated areas that are connected to the injection cluster or that are propped. In the case of the propped fractures, the user will be able to specify the conductivity threshold. In box (7), it is also possible to track the number of microcracks by layer.
Stimulated area in shear	Tracks the total surface of the joints that have slipped. In box (4), it is possible to differentiate and monitor the stimulated areas that are connected to the injection clusters or that are propped. In the case of the propped fractures, the user will be able to specify the conductivity threshold. In box (7), it is also possible track the number of microcracks by layer.
Stimulated area in tension and shear	Tracks the total surface of the microcracks created in tension and in shear. In box (4), it is possible to differentiate and monitor the stimulated areas that are connected to the injection points or that are propped. In the case of the propped fractures, the user will be able to specify the conductivity threshold. In box (7), it is also possible track the microcracks by layer.
Temperature (fluid node)	Tracks the fluid temperature at the spring nearest to a given location (East, North, Up). The location of the node must be specified in box (2).

	Note that the fluid node may not exist at the time that the model is initialized, and it will be assigned when the fluid network has reached the distance of one resolution from the specified coordinate.
Temperature (spring)	Tracks the rock temperature at the spring nearest to a given location (East, North, Up). The location of the node must be specified in box (2).
Time (fluid cycle)	Tracks the time spent in one fluid cycle. Only one of this history type is allowed per model.
Time (mechanical cycle)	Tracks the time spent in one mechanical cycle. Only one of this history type is allowed per model.
Time (thermal cycle)	Tracks the spent in one thermal cycle. Only one of this history type is allowed per model.
Thermal advection time step	Tracks the thermal advection time step during the simulation. Only one of this history type is allowed per model.
Thermal conduction time step	Tracks the conduction time step during the simulation. Only one of this history type is allowed per model.
Total Carter leak-off (cluster)	Shows the accumulated Carter leak-off during the simulation for a selected cluster. The cluster is identified in box (6). Only one of this history type is allowed per model.
Total Carter leak-off (stage)	Shows the accumulated Carter leak-off during the simulation for all clusters within a selected stage. The stage is identified in box (5). Only one of this history type is allowed per model.
Total displacement	Records the total movement (not affected by resetting displacements) of a mechanical node at a given location (East, North, Up) since the start of the simulation. The component (X or Y or Z) or magnitude of the displacement vector must be specified in box (1). The location of the node must be specified in box (2).
Total inflow (fractures only)	Records the total inflow of fluid into fractures in the model. The inflow can occur in the fractures intersecting boundary with fixed fluid pressure. See boundary conditions in Feature/Fluid tab. Only one of this history type is allowed per model.

Total outflow (fractures only)	Records the total outflow of fluid into fractures in the model. The outflow can occur in the fractures intersecting the boundary with fixed fluid pressure. See boundary conditions in Feature/Fluid tab. Only one of this history type is allowed per model.
Total inflow (matrix only)	Records the total inflow of fluid into the matrix in the model. The matrix inflow can occur on the boundary with fixed fluid pressure. See boundary conditions on Feature/Fluid tab. Only one of this history type is allowed per model.
Total outflow (fractures only)	Records the total outflow of fluid into matrix in the model. The outflow can occur on the boundary with fixed fluid pressure. See boundary conditions on Feature/Fluid tab. Only one of this history type is allowed per model.
Total inflow (fractures and matrix)	Combines the total fracture and matrix inflow described above. Only one of this history type is allowed per model.
Total outflow (fractures and matrix)	Combines the total fracture and matrix outflow described above. Only one of this history type is allowed per model.
Total volume	The total volume provides a summation of the volume of all the model fluid nodes. Only one of this history type is allowed per model.
Velocity	The rate of movement of a node at a given location (East, North, Up) in a specified component direction (X or Y or Z). The component (X or Y or Z) or magnitude of the displacement vector must be specified in box (1). The location of the node must be specified in box (2).

Solution Tab

The **Solution** tab in the **Model Control** panel is used to control the model solution scheme for mechanical, fluid, and thermal modes. It also allows the setup of the sub-lattice computation mode. Computation allowing determinist results has not been implemented yet, and the model deterministic behavior should be left as non-deterministic.

Mechanical Tab

Figure 93 shows the **Solution/Mechanical** tab. The fields for the mechanical setup behavior are described in the following text.

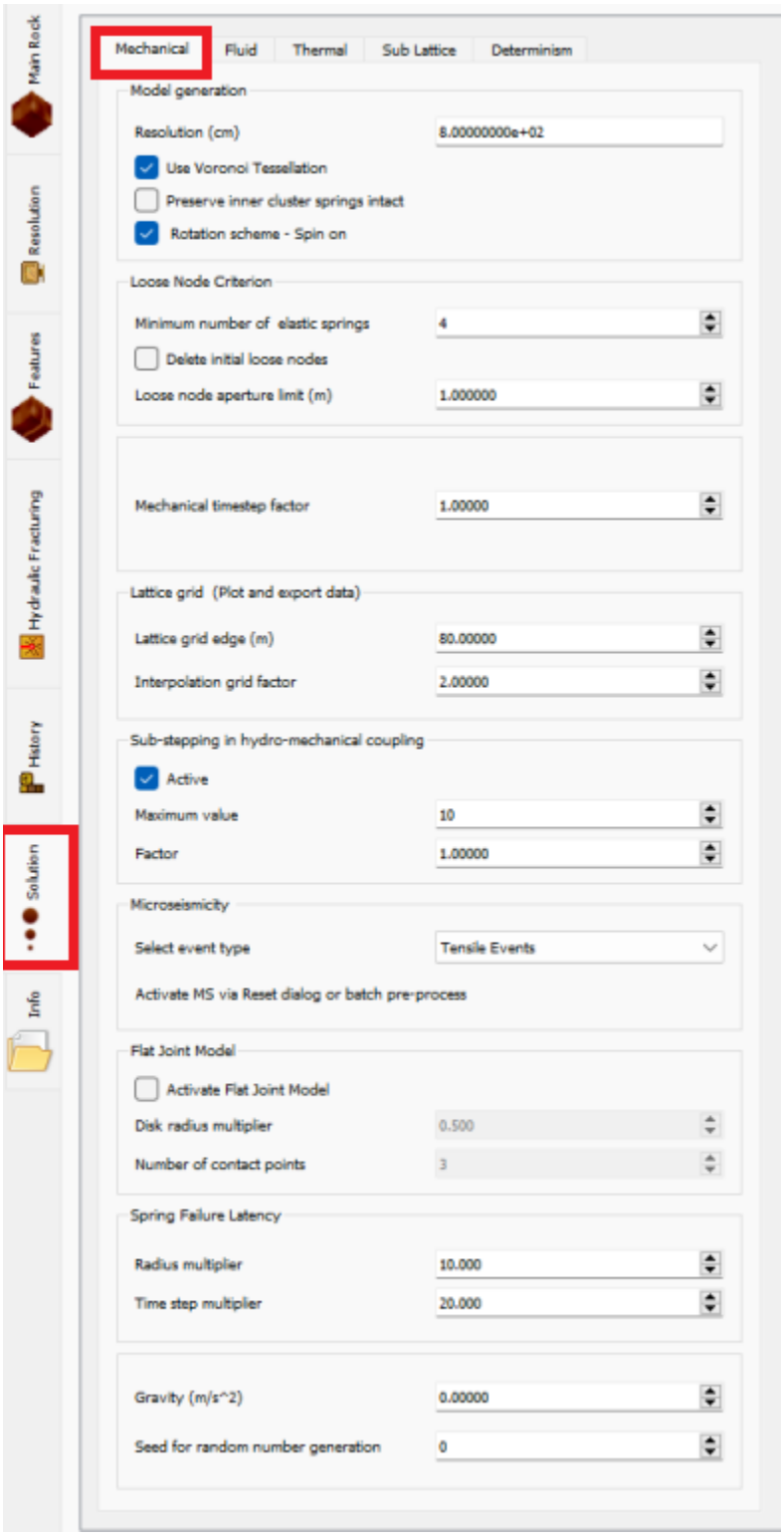


Figure 93 Solution/Mechanical tab.

Resolution

This field specifies the background resolution of the lattice. The background resolution is overridden by the settings defined in the Resolution tab. The Resolution tab describes how the user can set up a lattice with different resolution regions, allowing specification of finer resolution in areas of interest (e.g. induced fracture region). Resolution is the average lattice-node spacing (i.e., particle size). A small resolution will produce more accurate results than a large resolution, but the simulation time will be longer. Initial trials of a new model should be done with coarse resolution, as there are likely to be initial difficulties with setting up any new model. Afterward, the model can be run at a finer resolution for final simulation. The number of nodes across a given model dimension should be considered before setting the resolution. For example, if approximately 50 nodes across a 500-m block are required, then the resolution should be set to 1000 cm.

Use Voronoi tessellation

A spherical-grain-based ("regular") lattice has a significant level of noise, or force (strain) variability, which becomes a problem if the lattice is upscaled. Consequently, there is a tendency for diffuse cracking. A Voronoi lattice is representative of polygonal granular geometry with zero porosity. A Voronoi lattice results in more uniform strains (less dispersion). The Voronoi tessellation option uses a lattice generated from Voronoi tessellation instead of connecting springs between nodes using a packing of spherical particles. A spring between two nodes represents the contact/common face between polyhedral blocks that contain those two nodes. The spring stiffness is set proportional to the area of contact face. This reduces the variation in spring forces for a uniform stress state. This option generates more springs; therefore, the code will run slower. This option is recommended when the lattice resolution is large compared to the rock grain size (i.e., greater than 1 cm).

The regular lattice uses only tensile strength to calibrate the spring strength. The input fracture toughness should not matter for the regular lattice. Instead, the apparent fracture toughness will be a result of the tensile strength and particle radius. Thus, the apparent toughness will increase with an increase in the particle radius. For typical tensile strength and particle radius greater than ~1 cm, the regular lattice should overestimate the fracture toughness.

The Voronoi lattice uses both the tensile strength and the toughness to calculate the spring strength. Effective strength is smaller of two. For typical properties and resolution greater than 1 cm, the fracture toughness will control the spring tensile strength.

Preserve inner cluster springs intact

If selected, the nodes inside a cluster will be treated as elastic material. It is recommended to select this option to avoid spurious cracking in the cluster region and avoid any errors in the cluster logic.

Rotation scheme – spin on

If checked, the moment will be computed on mechanical nodes. Checking the rotation scheme is recommended. Spin calculations are important in problems where bending and/or rotation is anticipated. Turning spin calculation off can optimize some simulations, but the user should be aware of its effect on model accuracy.

Minimum number of elastic springs

A node keeps track of the number of solid springs connected to it. When a spring is broken, its nodes update their solid spring count. A solid spring is either an intact spring or a spring that is not intersected by a joint. When the number of connected solid springs is less or equal to the minimum number of elastic springs, the node is marked as a loose node. A spring is broken, forming a microcrack, only if the number of solid springs on both nodes connected to the spring is greater than the minimum number of elastic springs. In other words, springs are considered unbreakable if their nodes have a solid count less than the loose node criteria. Loose nodes do not update their moment.

Delete loose nodes

If this variable is checked, loose nodes will be deleted during lattice initialization. Loose nodes are explained above.

Loose node aperture limit

Defines the maximum aperture value for fluid nodes for springs with loose nodes. The purpose of this limit is to prevent the loose nodes acting as infinite sinks in the coupled hydro-mechanical simulations.

Mechanical time step factor

Factor applied to mechanical time step, which cannot be greater than 1. The purpose of the factor is to reduce the mechanical time step if there is an indication of numerical instability. The time step is calculated from the condition of stable simulation. However, some non-typical conditions may result in instability for the calculated time step.

Lattice grid edge (for plotting and export data)

The field Lattice grid edge defines the length of the zones in a three-dimensional grid that will be used to upscale lattice quantities and calculate, for example, stresses and permeability. It is very important to have the edge length of the grid cell set correctly to have a representative number of springs inside a

given cell. Cells without springs may occur if the edge is too small compared to the lattice resolution. It is recommended that the grid cell edge length be four to six times the lattice resolution length. The edge length may be changed after the model is created via the Reset Lattice dialog.

Interpolation grid factor

Dimensionless experimental value used to calculate the lattice grid edge. The recommended value is 2.0.

Hydro-mechanical sub-stepping – activation, maximum value, and factor

The simulations start with equal fluid and mechanical time steps. Subsequently, in typical problems, due to fractures opening, the fluid time step becomes smaller than the mechanical time step. If this feature is active, the code will carry out sub-stepping by conducting multiple flow steps per one mechanical step. The sub-stepping and the number of fluid time sub-steps is computed as the following:

nflstep = substepFactor * (fTimeInc0 / fTimeInc) / 2

where: nflstep is the new number of fluid sub-steps.

fTimeInc0 is the initial fluid time step equal to the mechanical time step.

fTimeInc is the current fluid time step;

The new number of fluid steps is a value between 1 and the maximum value entered.

Microseismicity – Select event type

Microseismicity is caused by bonds breaking (tensile events) in the rock matrix and sliding along joints (shear events). The program can track tensile and shear events.

When clustering the joint slip events, the largest N (the number specified by the user) events will be considered as outliers and removed from the computation. The activation of the microseismicity tracking feature and the activation of the feature that combine the shear events can be set on the Reset Lattice dialog.

Flat Joint Model

If **Uses Flat Joint Model** is activated, the lattice model will be constructed with the Flat Joint (FJ) spring model as an alternative to the default lattice spring model. The flat joint model simulates contacts between polyhedral (not spherical) blocks. The flat surface between such blocks prevents their relative rotation even after a contact is broken. Note that both UCS and tensile strength can be matched by the lattice if the FJ model is used. The number of sub-contact points in the FJ model can be set by the user (default value is three). The sub-contacts are used to represent progressive failure and non-uniform deformation within the contact. Gaps may open and sliding may occur at each sub-contact point separately. Even if all sub-contact points fail, the FJ model can resist rotation (i.e., can transfer moment)

if there is compressive stress in the contact. For a three-point FJ contact, the execution time is 1.5 times slower than a default lattice spring model.

The disk radius multiplier is applied to the spring resolution to determine the FJ radius. The flat joint radius multiplier is intended to reduce the flat joint radii to obtain a valid microstructure and avoid overlap between nodes.

Spring failure latency

The purpose of these parameters is to prevent spurious cracking in the model related to inertial effects caused by energy released immediately after formation of a microcrack (i.e., breaking of a spring). A microcrack cannot be formed if a close spring was broken within a certain interval time. The spring resolution times the "radius multiplier" defines the search region in which springs will be checked (i.e., marked as "neighboring springs"). In every mechanical step, the interval of current time minus the time when the spring was broken is calculated. If the interval is less than the "time step multiplier" times the time step, the neighboring springs cannot be broken. The default values should be used unless the user has a good reason to change them.

Gravity

This value should be equal to zero if stresses are uniform in the model. If a gradient is used in stress initialization, the gradient should be consistent with gravity and rock densities.

Seed for random number generation

Allows a new seed to be used where variable random generation is used (e.g., DFN distribution).

Fluid Tab

Figure 94 below shows the **Solution/Fluid** tab. The fields for setup of fluid flow simulation are described in the following section.

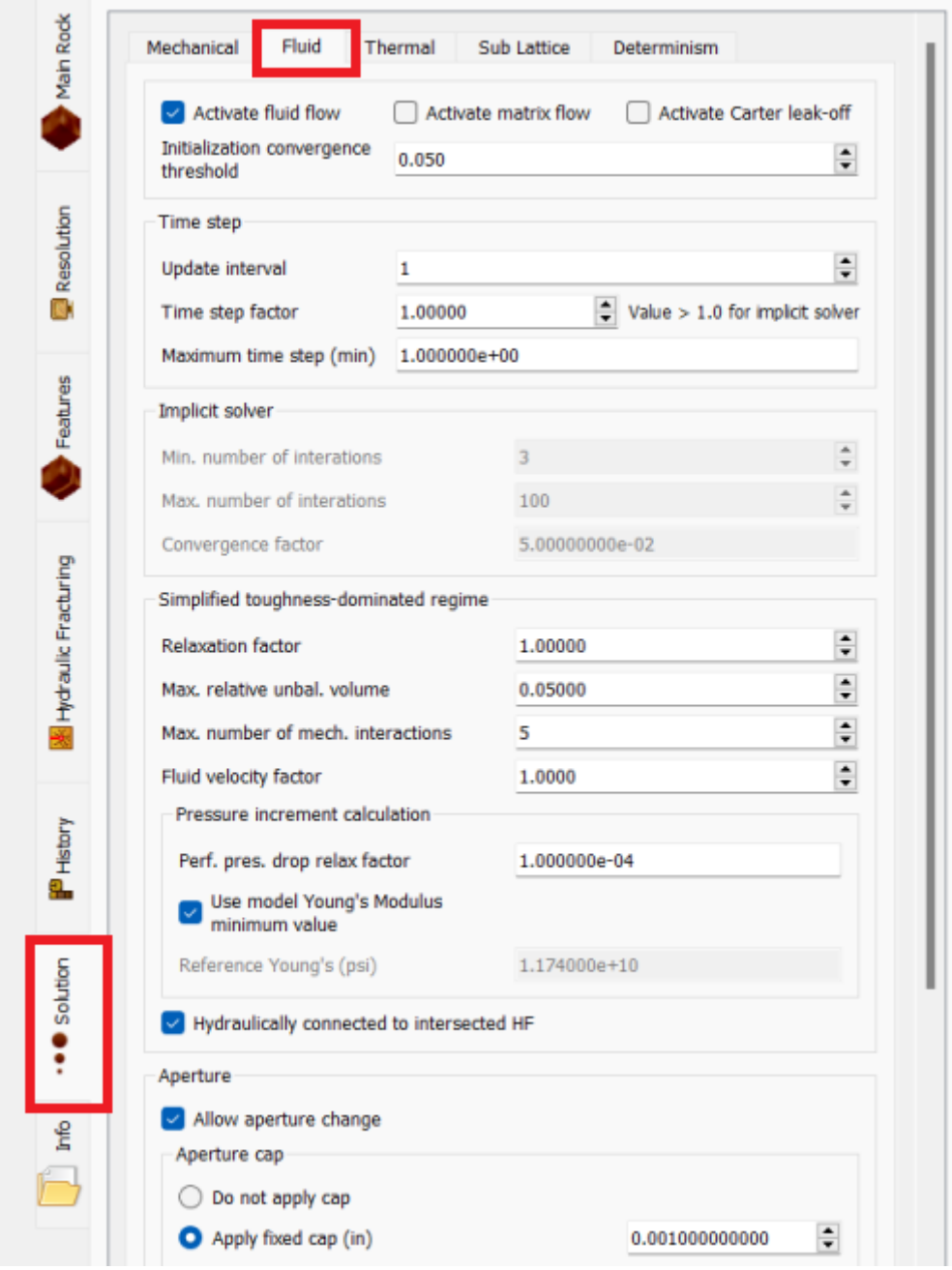


Figure 94 Solution/Fluid tab settings.

Activate fluid flow

This selection enables flow calculation in joints and induced fractures. When enabled, fluid nodes and the pipe network will be created where the joints intersect springs during the lattice initialization. Fluid nodes and pipes will be installed automatically in newly formed cracks.

Activate matrix flow

This selection enables flow calculation within the rock matrix.

Activate Carter leak-off

Activates Carter leak-off computation.

Fluid initialization convergence threshold

Possible values in the range 0.0 to 1.0.

This controls the accuracy of the relaxation scheme in calculating the steady-state of the initial fluid pressure and flow. This calculation is performed only once and is intended to be more rapid than the regular non-steady simulation for establishing the initial state.

A small value for this factor will give greater accuracy but require a longer time to solve.

If the initial relaxation scheme is not to be executed, then a factor of 1.0 may be specified.

Fluid time step update interval

Sets frequency of the fluid time step recalculation. Default value is 1 step. By increasing this value, the model will run faster, but the potential for numerical instability also will increase.

Fluid time step factor

Full Flow option solves fluid diffusion through the entire pipe network. If the Fluid Time Step Factor (FTSF) is equal to or less than 1.0 (the default value is 1.0), the explicit numerical scheme will be used to solve evolution of fluid pressures. If the FTSF is greater than 1.0, the code will use the implicit numerical scheme. The increase in the FTSF will result in a longer simulation time step, which means shorter simulation time but may also result in numerical instability.

Maximum time step

Determines the maximum fluid time step for the simulation. The purpose of this parameter is to prevent very large time steps at early simulation stages (when initial apertures are small). The large time steps

can result in non-physical response (e.g., excessive cracking due to sudden increase in fluid pressure) in nonlinear problems.

Implicit Solver

The explicit solver is always stable, but it requires a small time step, resulting in a long simulation time to solve the flow calculation. The implicit scheme uses the Crank-Nicolson method, which has an order 2 in time, allowing a larger time step. A system of pressure head equations must be solved, with one equation for each fluid node. Flow-only calculation is always stable, but coupled simulations may not always be stable. In a simulation, the pipe network and pipe aperture grow over time. Therefore, the number of equations and matrix coefficients are changing continually. Since it is inconvenient and time consuming to store and constantly rebuild a potentially very large matrix, the solution uses the Jacobi iterative procedure. Hence, there is no need to create or store a matrix. In this method, even if the solution converges, accuracy can be affected by long time steps.

The variables listed below control the implicit solver calculation. They are enabled when the fluid time step factor is greater than 1.0.

Minimum and maximum number of iterations

For each fluid flow cycle, the implicit solver will cycle at least the **Min. number of iterations** and at most, the **Max. number of iterations**. The default value for these fields is 3 and 500, respectively. It is recommended to set a "Number of iterations of the implicit solver" history to verify if the number of iterations is reasonable to avoid possible accuracy problems.

Convergence factor

The implicit solver breaks from the cycle if the maximum cycle pressure head-change is less than the maximum pressure head times the **Convergence factor** (4).

Simplified toughness-dominated regime options

The simplified toughness-dominated regime is recommended when hydraulic fracturing can be approximated by the toughness-dominated propagation regime. Pressure will only be applied to fluid nodes that are connected to an active cluster. No pressure-drop (i.e., uniform pressure in the fracture) is assumed by default. If the approximate pressure gradient is selected, the pressure gradient is estimated assuming that the fluid velocity is equal to the rate of advance of the fracture tip. If this option is selected, the calculation of fluid pressure evolution with time is faster than the full-flow calculation using the explicit numerical scheme (i.e., FTFS = 1).

Relaxation factor

To avoid cluster pressure oscillations and potential instability when the simplified logic is used and sudden changes in distribution of flow between perforation clusters occur (e.g., after diverters plug some of the clusters), the perforation pressure drop calculation uses a relaxation scheme. The relaxation factor prevents sudden perforation pressure drop changes (even if the flow rate changes are sudden). Practically, the smaller the relaxation factor, the slower perforation pressure-drop changes are. For each step, the stage pressure increment is recalculated. The relaxation factor is applied to the pressure increment computation, allowing the solution to converge more slowly (values less than 1.0), avoiding possible numerical errors.

Maximum relative unbalanced volume and Maximum number of mechanical interactions

The fluid volume injected into a stage is accommodated by the volume of the cracks connected to the stage. Increase in pressure results in the creation of new cracks and opening of all cracks. The simplified logic for the toughness-dominated regime uses a relaxation scheme by calculating the pressure increments based on the unbalanced volume (the difference between the injected fluid volume and the volume of the cracks) with the goal of making the unbalanced volume zero. The relaxation scheme stops when either the relative unbalanced volume drops below the specified maximum relative unbalanced volume or the number of iterations reaches the maximum number of mechanical iterations. Use the **Stage relative unbalanced volume** history plot item to monitor the simulation convergence.

Fluid velocity factor

Factor applied in calculation of the proppant settling velocity during the simplified logic for the toughness-dominated fracturing regime.

Perforation pressure-drop relaxation factor

Factor applied in calculation of the stage pressure drop during the simplified logic for the toughness-dominated fracturing regime.

Use model Young's Modulus minimum value

The relaxation scheme in the simplified logic for the toughness dominated regime uses the smallest Young's modulus in the model multiplied with the Relaxation Factor as a relaxation parameter to calculate the pressure increments. However, a layer of very soft material may be applied to the model boundaries to approximate stress boundary conditions. If the Young's modulus of such a layer is used in the simplified logic, it will cause the simulation to converge very slowly. By unchecking this field, the user can override the automatically selected Young's modulus and specify the Young's modulus of the rock in the region where fractures will occur.

Hydraulically connected to intersected HF

If this field is checked, intersected fractures during simulation using the simplified logic will be hydraulically connected.

Aperture Cap

During coupled hydro-mechanical simulations, the joint apertures can be considered fixed (i.e., independent of deformation) or allowed to change due to deformation.

An excessive increase in aperture can lead to relatively small calculation time steps and time-consuming numerical simulations. Setting a cap on the apertures can optimize such runs without significantly affecting accuracy of the numerical solution.

Possible capped aperture values are:

a) **Not capped**: aperture is the actual fluid node aperture value.
b) **Fixed cap**: the aperture is the minimum value between the quantity specified by the user or the aperture value of the fluid node.
c) **Cluster average cap**: If the fluid node is connected to a cluster, the aperture cap is the minimum value of the fluid node aperture or the average of the aperture of all fluid nodes connected to the cluster multiplied by a factor defined by the user. The default value for the factor is 2.0. The average cap is calculated according to the average update interval field.

Aperture cap effect

These options apply only to the simulation running in full flow mode. If the option **no pressure-drop** is selected, connected fluid nodes are grouped based on the apertures of pipes. The pressure for the group is calculated and assigned to all the fluid nodes that belong to the same group. Fluid nodes are reassigned to groups according to the specified update interval field. The **limit flow rate** option performs the calculation as explained in the XSite Formulation document.

Thermal Tab

Figure 95 shows the **Solution/Thermal** tab. The fields for thermal simulation setup are described next. For a complete explanation of the thermal usage in *XSite*, please refer to the companion Thermal Analysis Users Guide.

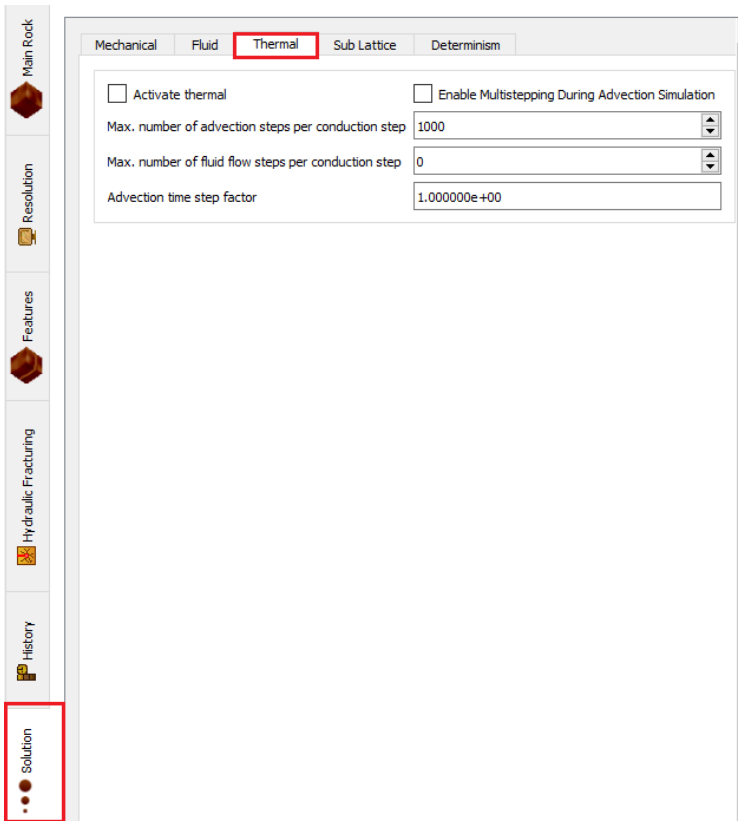


Figure 95 Solution/Thermal tab.

Activate thermal

Heat transport (and thermo-hydro-mechanical response), including heat conduction in the rock, heat advection by fluid and heat exchange between fluid and rock, is enabled.

Enable multistepping during advection simulation

This field enables multistepping during thermal advection simulation. The multistepping divides the flow pipe network in multiple groups depending on their heat advection time steps and executes the heat advection calculation for each group at different frequencies depending on their time steps.

Max. number of advection sub-steps per conduction step

The critical time step of heat advection is much smaller (orders of magnitude) than the critical time step of heat conduction numerical simulation. The optimum sub-stepping would mean execution of each

heat transfer model at its critical time step. However, if the conduction time step is executed more frequently (e.g., every hundred or thousand advection time steps), the execution time will not be affected significantly while the evolution of temperature in the rock (driven by heat conduction) would be smoother.

Max. number of fluid flow sub-steps per conduction step

In tightly coupled THM simulations, the heat conduction time step is limited relative to the fluid flow time step, which is significantly smaller than the heat conduction time step.

Advection time step factor

The factor can be used to reduce the advection time step. The default is 1.0, and it cannot be greater than 1.0.

Sublattice Tab

The idea of the dual-lattice scheme is that the sub-lattice is used in the limited region around the tip of the propagating fracture(s) to correctly resolve the conditions of the fracture propagation. Use of sub-lattice in the limited region of the model allows greater flexibility in selecting a finer resolution for the sub-lattice and better accuracy of prediction of fracture propagation without affecting the critical time step (a function of the main lattice resolution) and overall simulation time.

Figure 96 shows the **Solution/Sub-lattice** tab. The fields for sub-lattice setup behavior are described next.

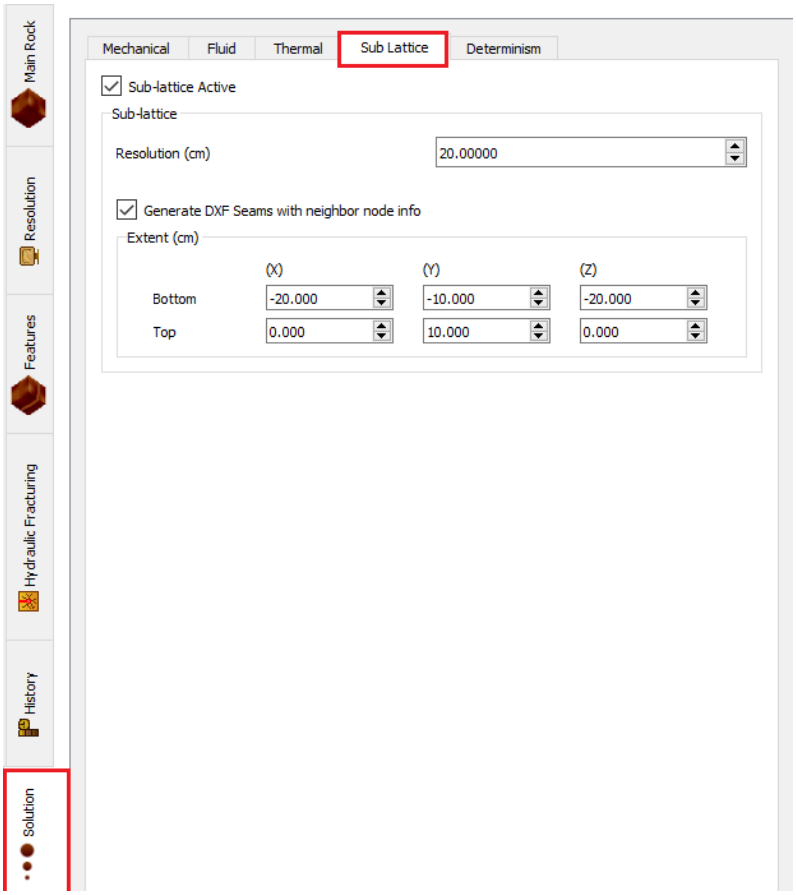


Figure 96 Solution/Sub-lattice tab.

Sub-lattice active

Activates the sub-lattice computation.

Sub-lattice resolution

Sub-lattice should allow a longer time step for fully coupled simulations. The reason is that the sub-lattice does not solve fluid flow. Therefore, smaller resolution of the sub-lattice should not affect the flow time step.

Generate DXF seams with neighbor node information.

If this flag is checked, when the sub-lattice zone is being initialized, the seam information that will be applied to the sub-lattice node will be inferred from the neighboring lattice node. This speeds up sub-lattice zone activation. Otherwise, the seam information will be inferred from the seam geometry.

7.0 Appendices

Appendix 1 – Workflow for Modeling of Effects of Reservoir Depletion (Parent-Child Wells)

A1.1 Introduction

XSite includes functionalities that allow efficient modeling of parent-child well interaction. It is assumed that reservoir stimulations from both the parent and child wells are modeled in XSite. The production from the parent well is simulated using a reservoir simulator software (RSS), with reservoir properties (e.g., permeabilities and porosity) that can be obtained by upscaling XSite results (of reservoir stimulation from the parent well) such as created hydraulic fractures and propped apertures of both pre-existing and hydraulic fractures. The depleted pressure field, after some time of production from the parent well will have effect on stimulation from the child well. Thus, the model of the child well should be initialized by the reservoir state (fracturing, propped apertures, and stresses) from the parent well at the end of the simulation of its stimulation phase, and by the depleted pore pressure field resulted by production from the parent well at the time of stimulation of the child well. The methodology of modeling parent-child well interaction in XSite is described in this appendix.

A1.2 Model Creation

Two independent models, for parent and child wells, should be created with the same geometry. The first model (M1) will include parent wells that will be stimulated first and provide the permeabilities for the RSS. The second model (M2) will include child wells that will be stimulated after production from the parent wells and the depletion of the pore pressure field. Figure 97 shows the geometries and well designs for models M1 and M2, which should have a soft layer on the top boundary to allow model subsidence caused by depletion.

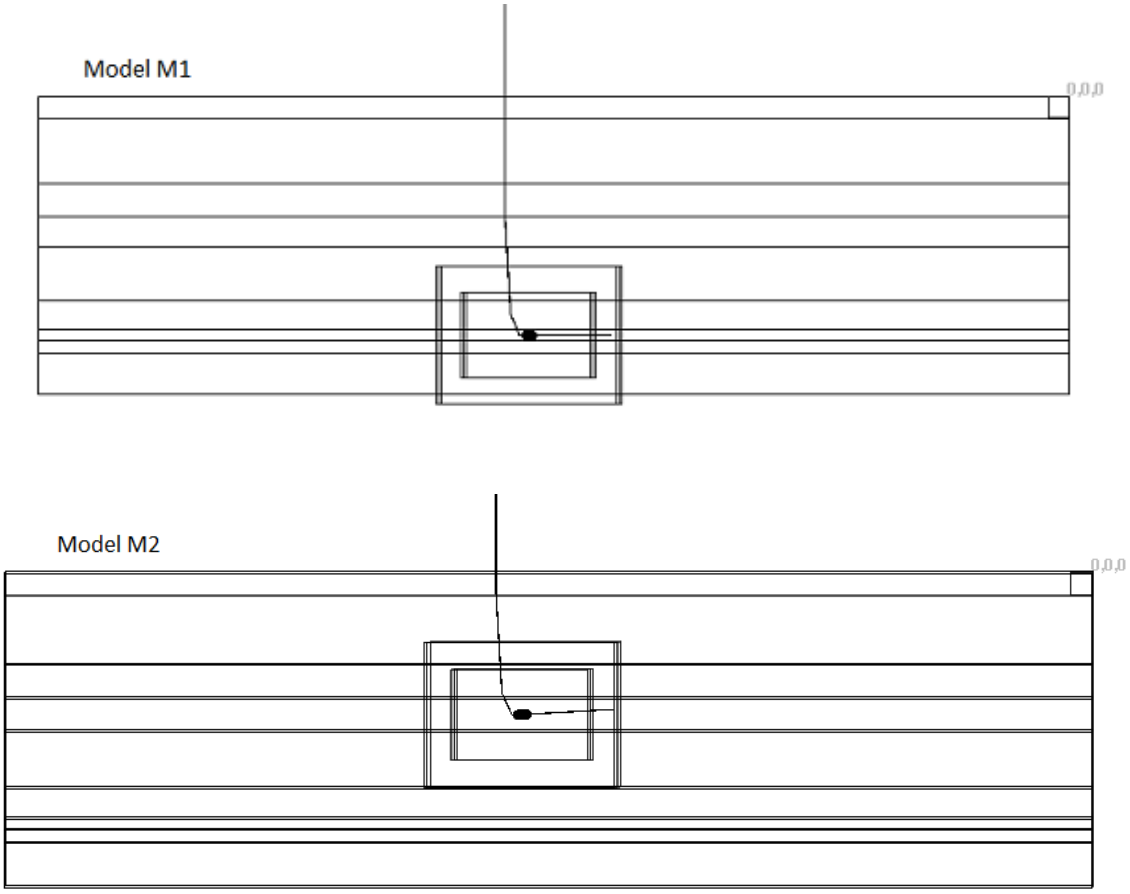


Figure 97 Geometries of the parent- (Model M1) and child-well (Model M2) models

A1.3 Child-Well Model Initialization

After running the simulation of Model M1, permeability data are exported to the RSS and depletion data will be available after the RSS production simulation. Model M2 includes the geometries of the child wells. Although it does not need to include the geometries of the parent wells, the child-well model must include the state at the end of the stimulation of the parent well (i.e., the end of simulation of Model M1).

The dialog box shown in Figure 98 is a screenshot from Reset Lattice/Import Data. The same dialog box is also available in Batch Simulation/Pre-Process (Import). The dialog shown in Figure 98 provides control of the functionalities that enable proper initialization of the child-well model (Model M2). There are two ways to import the fracturing data into Model M2.

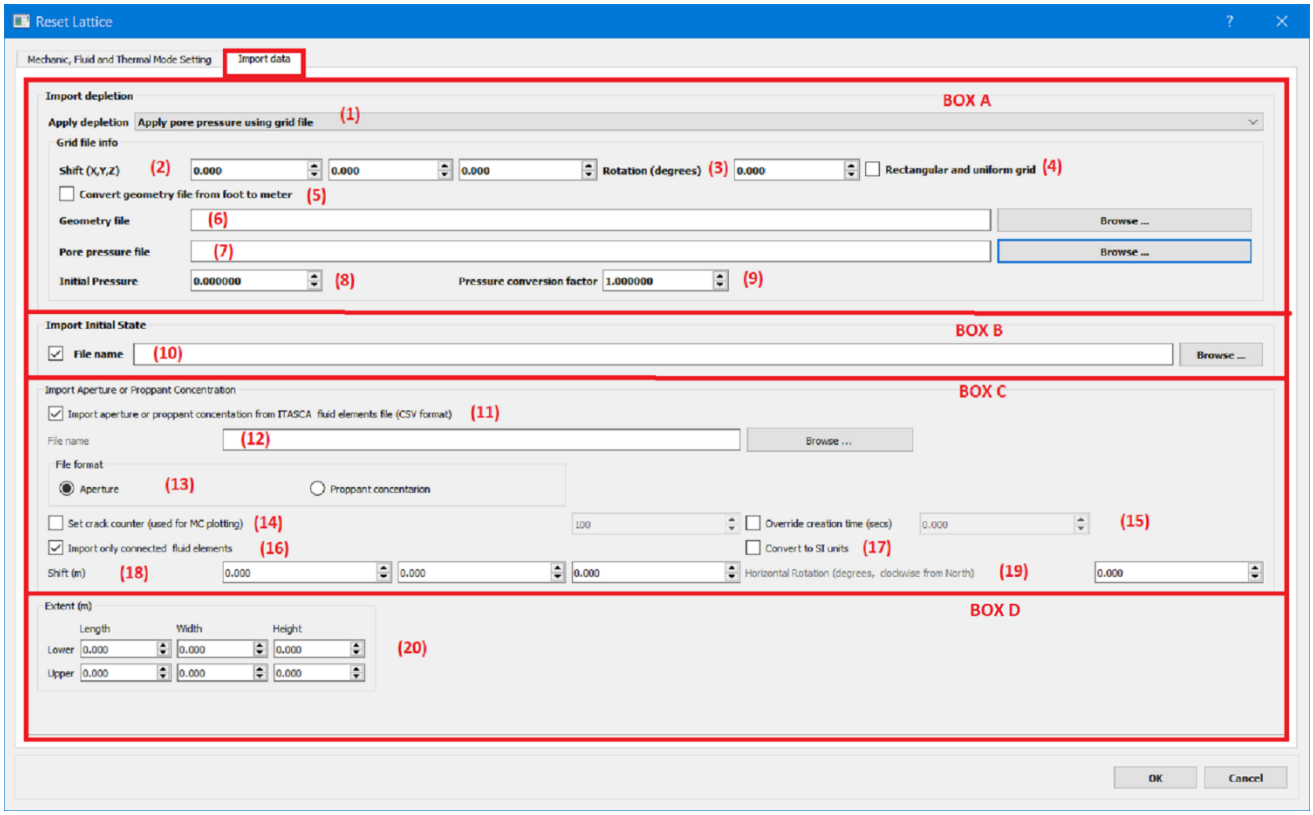


Figure 98 Import data using "Reset Lattice" dialog.

Importing directly from XSite save file.

Load model M2 and initialize it (run simulation for 0 seconds). Use dialog in Figure 82 to import Model M1 last simulation state.

In "**Box B**" of Figure 98, select "Import Initial State", navigate to the directory where the Model M1 simulation states are located, and select the proper state file (at the end of the simulation of Model M1). Press the "OK" button to load the state.

The code will verify if the dimensions of the models are the same. For this version of the code, the width of the models may be different, but the length and height must be the same. The springs in Model 2 that exceed the width of Model 1 will be initialized with stress conditions defined in Model 2.

Add a "Matrix" and "Stress Contour" and "Microcracks" plot items to check if the model was properly imported. We recommend adding a "cut plane" before using the contour plotting because construction of contour plots sometimes may take a while. Also, the contour plot item is more illustrative (less fluctuations) if the lattice grid edge is four to five resolutions long. You may also use the Microcracks plot items to display the imported fractures. Make sure that the "connected" flag of this plot item is not checked.

Importing from CSV file

The second method provides more flexibility (e.g., not required matching geometry between Models M1 and M2) but requires following of the multiple steps. These are the steps to import data (fracture geometry, apertures, and proppant concentration) from models used for the simulation of parent wells.

1) Export in **.csv format** a fluid elements file from the model where the parent wells were simulated (Model M1) using menu options "File/ File/Export data/Fluid Elements".
2) Load the child well model and initialize the model (i.e., run a simulation for zero seconds). Applying adaptive resolution before importing is recommended.
3) Import apertures (more details in the following sections).
4) Plot "Microcracks" to verify that the data is imported correctly.
5) Run in the mechanical mode with elastic option until model reaches equilibrium. (To make sure if model is in equilibrium, plot "Velocity Filed" and check if the largest velocity is less than 1e-6 m/s).
6) Run in the mechanical mode until model reaches equilibrium.
7) Plot "Fluid/Pipe/Apertures" and compare with the parent well model to make sure that there is a reasonably good match.
8) Import proppant concentration (more details in the following sections).
9) Plot "Fluid/Proppant Concentration" to verify if proppant concentration was properly imported.
10) Run in mechanical mode with elastic option until model reaches equilibrium.
11) Run in mechanical mode until model reaches equilibrium.

All the steps above can be easily created and added to batch items via the Batch Depletion Workflow dialog that is described in the section "Batch depletion workflow".

Importing Apertures

This operation will install microcracks, fluid nodes and pipes into the existing lattice. After the model is run to equilibrium, stresses will be in place.

The file that contains aperture data is obtained from the model used for the simulation of the parent wells. Use the menu option "File/Export Data/Fluid elements" to create the file. In the dialog "Export Pre-Existing Joints" dialog select the file format option "Pre-existing joints and microcracks." Note that this file contains both, aperture, and proppant concentration information.

To import apertures, first you need to have the model initialized. Aperture data is imported using either the "Reset Lattice" dialog (press the  button on the toolbar to access this dialog) or using batch operation. Figure 98 shows the tab "import Data" of the "Reset Lattice" dialog. The same fields are defined on the Pre-Process (Import) tab of the Batch Simulation dialog.

To import aperture, you need to fill the fields located in **"BOX C"** in the figure above:

1) Check the option "Import aperture or proppant concentration, field (11). This will activate the fields (12) to (19).

2) Enter the name of the fluid elements file exported from the parent-well model field (12). Use the browse button to navigate to the directory where the file is located.
3) Select Microcrack, field (13) on file format.
4) Fields (14) and (15) allows to set the Microcrack Counter and Override the Microcrack creation time.
5) It is recommended to select "Import only connect fluid elements (16) "to avoid importing spurious microcracks created during the parent well simulation.
6) Select the "Convert to SI Units … (17)" check box if the parent well file uses oil-field units.
7) Fields (18) and (19) allows to translate and rotate the input data. For example, to translate the data 10 m in the west direction, enter "10 m" into the "Translation/East" field. The "Horizontal Rotation" field will rotate the model in the clockwise direction from northing.
8) The default values for the extent are the upper southwest and lower northeast corners of the child-well model. By using the default values, the data will be imported in the entire model. However, if, for example, for a model with dimensions 20×20×20 m you want to import data only into the top 10 m of the model, specify "-10 m" in the "Lower" and "Height" field.
9) Press the "Import button" to import the data. A dialog box will be displayed informing how many microcrack or fluid elements were imported into the model. This number should be used to check with data from the parent well simulation. Plot Microcracks to confirm the number and location of the imported fractures. After the model is run to equilibrium plot Pipes/Aperture to verify if the apertures match the parent-well model.

Importing Proppant concentration

The procedure for importing proppant concentration is the same as for importing apertures, except that for field (13) of Box C of Figure 98 where the selection should be "Proppant concentration."

A1.4 Applying Depletion Data

The depletion data are imported using two files, the geometry and depletion data files:

1) The geometry file contains grid information, and it is the same file used to create the permeability tensor data sent to the SSR. XSite has the capability to create a grid file in the proper format. Section A1.6, (Format of the geometry file) contains an example of this file.
2) The depletion data file contains a pore pressure value for each corresponding zone in the geometry file (grid file). Section A1.7 describes the format of the depletion file.

To import the depletion data, follow these steps (with reference to **Box A** in Figure 98).

1) Select "Apply pore pressure using grid file" from the "Apply depletion" combo box by field (1).
2) If the grid file is given in global coordinates, set the transformation values for fields Shift (X, Y, Z) by field (2), otherwise set these values to zero.
3) Use field (3) to horizontally rotate the input data if necessary. The model will rotate in the clockwise direction from northing.

4) If the grid file is regular and uniform, check the filed (4) box. This will reduce the time that the program takes to import the data.
5) Select "Convert geometry file from foot to meter", field (5), if Model M2 geometry is specified in meters and the grid file is specified in feet.
6) Use the browse button to obtain the path of the grid file in field (6).
7) Use the browse button to obtain the path of the depletion file in field (7).
8) The initial pressure in field (8) is the initial pressure in the RSS model. The pressure read from the depletion file will be subtracted from this value and applied to the model (as the pressure change causing model deformation and stress change).
9) The internal unit for pressure in XSite is Pa. Enter the appropriate conversion multiplier factor in field (9) if the pressure unit in the depletion file data is not Pa. For example, to convert from PSI to Pa enter the value 6,895.0. **If no initial pressure is specified in field (6), the values in the file must be negative, otherwise the multiplier should be a negative number.**
10) Press the OK button to import the depletion data.

If the pressure read from the depletion file is zero, the pressure assigned to XSite will use the user input initial pressure. The pressure read from the depletion file will be subtracted from the initial value and applied to the XSite model (as the pressure change causing model deformation and stress change). The program algorithm works as following:

```
initialPressure  <= from user input
pres             <= read from depletion file
if (pres is equal to 0)   pres = initialPressure; // zero value is specified, assume same as background pressure
delta_pres = initialPressure - pres;              // Change in pressure
new_pres = Xsite_Pressure - delta_pres;           // Pressure to be used for matrix pressure
Xsite_Pressure = new_pres
```

After the depletion operation is completed, the dialog shown in Figure 99 will be displayed. Make sure that the data were successfully imported by verifying that the number of matched springs is not zero.

Also, use the "Matrix Pore Pressure" plot item to verify the pore pressures were properly applied (Figure 100). In the plot, the plot item "Upper Limit" was set to -1.0 to discard positive pressures and contour attribute is inverted.

After importing the depleted pore pressure field, the model should be run to mechanical equilibrium (2-10 s depending on the model size and depleted pore pressure field). The first part of the equilibration step should be in elastic mode. For example, if you are running the equilibrium phase for 2 s, run the first second in elastic mode. This will avoid the creation of spurious cracks due to the stress changes caused by pore pressure depletion.

Be aware that the Biot coefficient should be set to correct values (in material properties) so that stress changes associated with pore pressure changes are properly calculated.

Model M2 is now ready for simulation of the second well.

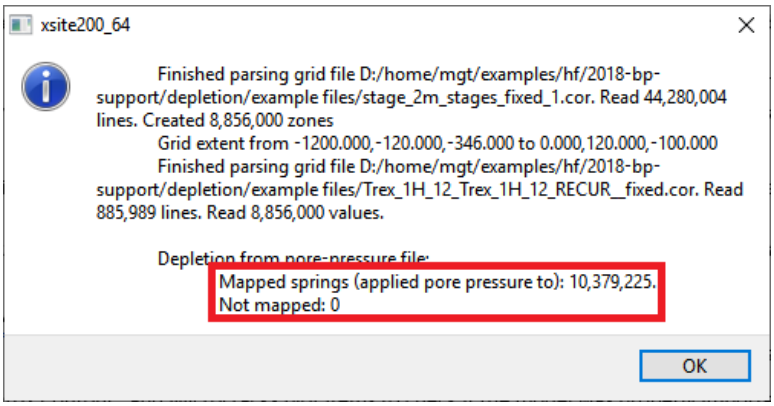


Figure 99 Message displayed after depletion data is imported.

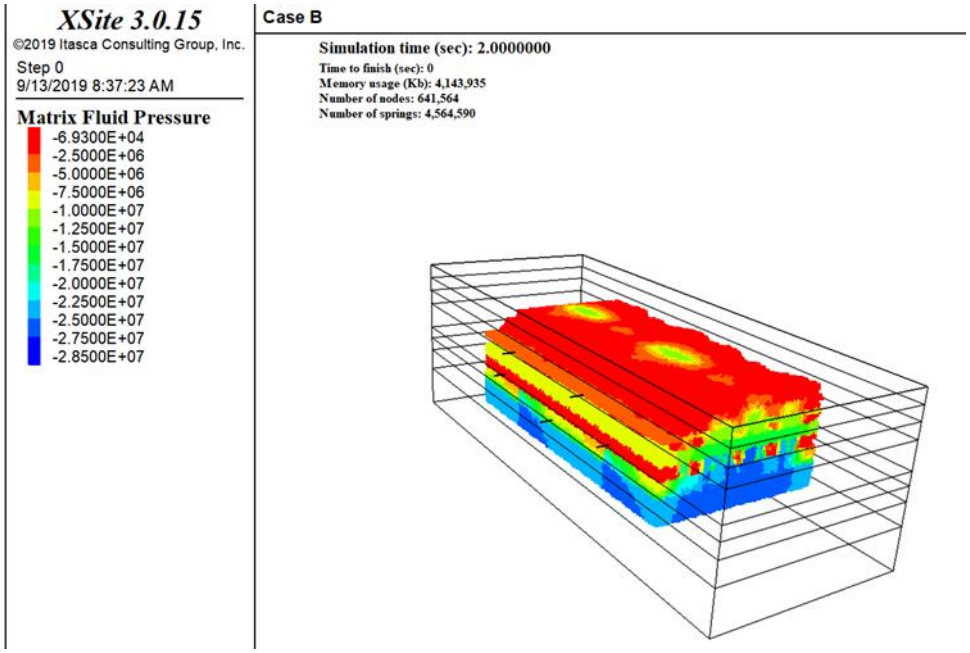


Figure 100 Pore pressure after depletion data is imported.

A1.5 Depletion workflow module

The "Depletion Workflow" module automates the process of initializing the parent-well models. With few keystrokes one can easily add to batch processing items the steps that are outlined in the previous sections. The Depletion Work module can be accessed by pressing the "Depletion Workflow" button available in the "Batch Simulation Steps" dialog as shown in Figure 101.

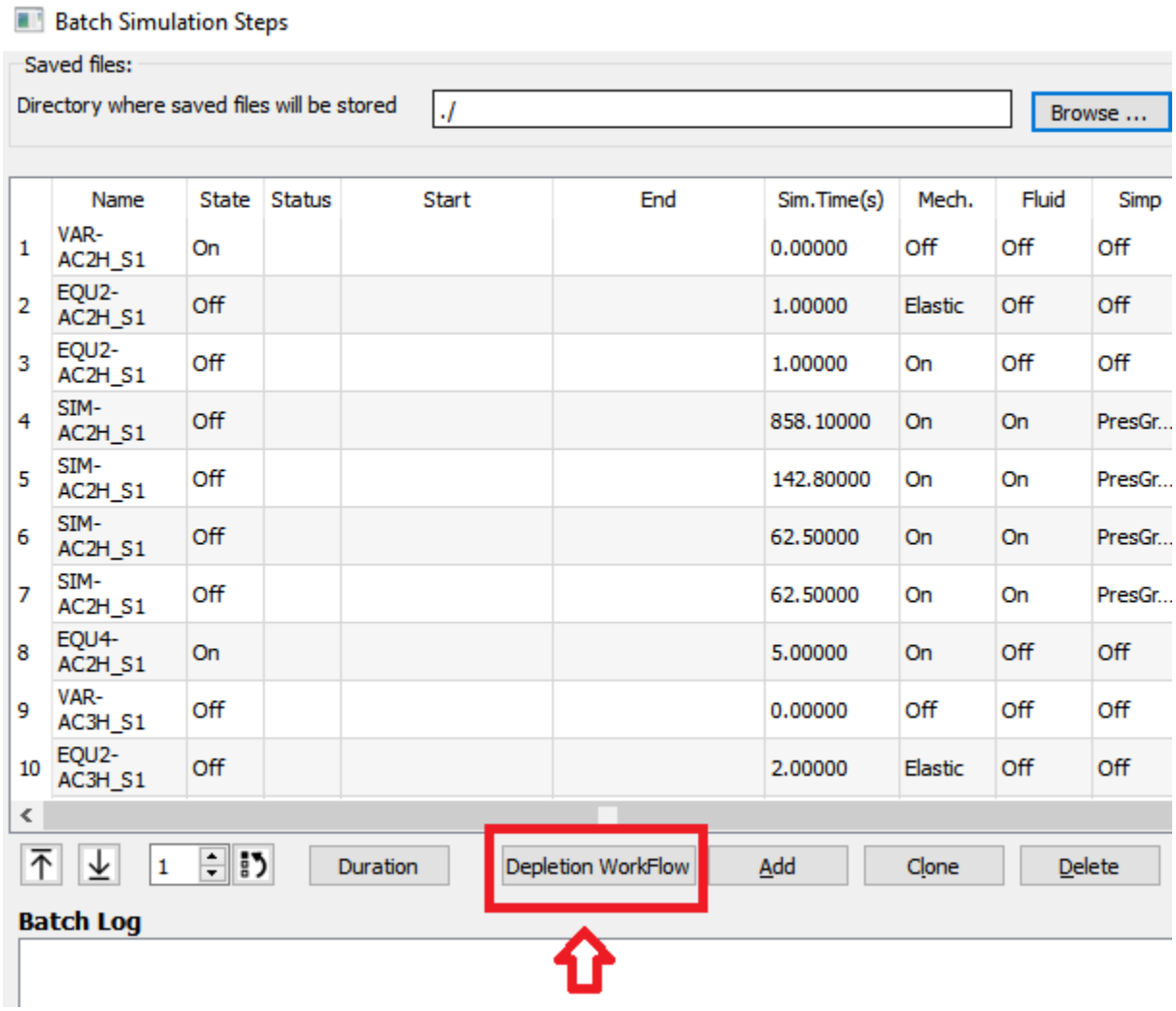


Figure 101 Accessing Depletion Workflow dialog.

Figure 102 below shows the "Depletion Workflow" dialog. The workflow is divided in four separate tabs. The checkboxes that are selected in the dialog will be added to the batch steps in the sequence in which they appear in this dialog. For example, if you select "Import Aperture from .CSV file" on tab "Step (1) Import Aperture", the fields that you populate in this section of the table will be included in the first batch step that will be generated by the workflow.

The first three tabs of the dialog are grouped according to how they will be processed during the batch operation: pre-process, simulation, and post-process. The fields available in these tabs should be populated as explained in sections above. If "Mechanical active" is also selected, a second batch step is created for the tab. Therefore, these three tabs may generate one or two batch steps each, depending on the items selected.

After executing all tasks defined in the workflow, the simulation time may be different from the start injection time defined in the pumping schedule. Tab 4, "Step (4) Simulation time" will add a batch command that will reset the simulation time to be the same as the start injection time.

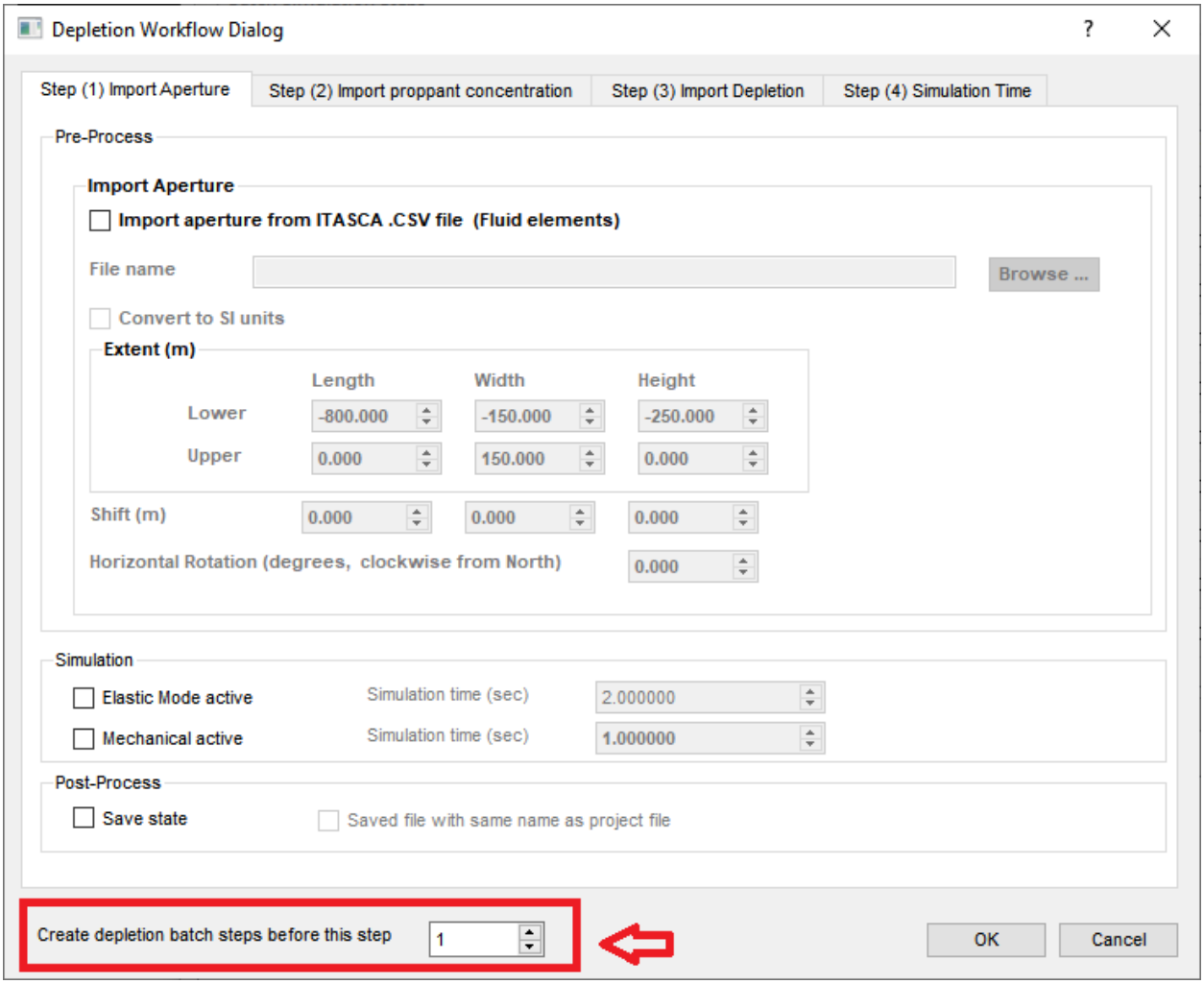


Figure 102 "Depletion Workflow" dialog

The value of the field "Create depletion batch steps before this step" will place the batch steps created by the workflow before the batch item indicated in the box. For example, six new depletion batch items will be created and if the value entered in the edit box is 2, the new batch items will be placed between locations 2 and 7 and the previous batch items located after position 2 will shift down six slots to position 8.

A1.6 Format of the geometry file

The grid size is 74 × 38 × 10 m. The location 1,1,1 is the top corner on northwest. The locations change in the following order: I or (X) most rapidly, then J or (Y), then K or (Z). The first 24 numbers represent the gridpoints of the zone on the northernmost and shallowest row of depletion from west to east.

```
C GRID BLOCK: I = 1 , J = 1 , K = 1
-664.000  114.000  -130.000  -658.000  114.000  -130.000
-658.000  108.000  -130.000  -664.000  108.000  -130.000
-664.000  114.000  -136.000  -658.000  114.000  -136.000
-658.000  108.000  -136.000  -664.000  108.000  -136.000
C GRID BLOCK: I = 2 , J = 1 , K = 1
-658.000  114.000  -130.000  -652.000  114.000  -130.000
-652.000  108.000  -130.000  -658.000  108.000  -130.000
-658.000  114.000  -136.000  -652.000  114.000  -136.000
-652.000  108.000  -136.000  -658.000  108.000  -136.000la
........
C GRID BLOCK: I = 73 , J = 38 , K = 10
-232.000  -108.000  -184.000  -226.000  -108.000  -184.000
-226.000  -114.000  -184.000  -232.000  -114.000  -184.000
-232.000  -108.000  -190.000  -226.000  -108.000  -190.000
-226.000  -114.000  -190.000  -232.000  -114.000  -190.000
C GRID BLOCK: I = 74 , J = 38 , K = 10
-226.000  -108.000  -184.000  -220.000  -108.000  -184.000
-220.000  -114.000  -184.000  -226.000  -114.000  -184.000
-226.000  -108.000  -190.000  -220.000  -108.000  -190.000
-220.000  -114.000  -190.000  -226.000  -114.000  -190.00
```

A1.7 Format of the depletion data file

Values are space separated and use 10 values per line. Each value in this file corresponds to a zone in the grid file. For zero values, XSite assumes same value as background pressure. Lines starting with the 'C' character are considered comments and they will be discarded. The pore pressure values in XSite are defined in Pa.

```
C ROOT    45          20          10
  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000
  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000
  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000
  ........
  12597.05 12597.05 12597.05 12597.049 12597.047 12597.045 12597.043 12597.041 12597.039 12597.036
  12597.03 12597.02 12597.02 12597.023 12597.019 12597.016 12597.012 12597.008 12597.004 12597.000977
  ........
  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000
  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000
  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000
  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000
  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000
  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000
  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000
  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000  0.000000
```

Appendix 2 – Parametric Study of Multistage Models

A2.1 Introduction

Preparing parametric study of multistage models may be cumbersome and error prone. XSite is capable of automatically, efficiently and accurately prepare the input files for parametric studies. In addition, XSite allows creation of different fracture design scenarios utilizing different completion designs, combining them to create new models.

A2.2 Simulation Setup

Utilizing the XSite GUI (**Parametric Study** dialog box, **Completion Design** tab in Figure 103) the user will be able to generate files that contain different variations of stage design and pumping schedules.

A2.2.1 Workflow

The workflow to create the stage design and pumping schedule files is the following:

1. Create **Cluster Design** (**Resources Menu**) items.
2. Create **Stage Design** (**Resources Menu**) items utilizing **Cluster Design** items created in item (1) of this list.
3. Create **Pumping Schedule** (**Resources Menu**) items.
4. Create **Simulation Sequence** (**Resources Menu**) items with Pumping Schedule items created in item (3).
5. Create well trajectories (**Hydraulic Fracture/Trajectory** tab). Associating stage design to well trajectories is optional. However, stage design records attached to a well trajectory may be used as the basis for creating the variation files for stage design.
6. Optionally, create Simulation Sequence items (**Hydraulic Fracture/Sim. Sequence** tab). The simulation sequence items may be used as the basis for creating the simulation sequence variation files.
7. Create intermediate files with variations of stage design and simulation sequence records that will be merged into different model configurations (**Parameters for Completion Design** tab of the **Parametric Study Setup** dialog).

A2.2.2 Creating Configuration Files

This section details item (7) of the workflow above—that is, how to create the files used to set up the different completion design configurations. The creation of the stage design and simulation sequence variation files will be performed on the tab **Parameters for Completion Design** located on the **Parametric Study Setup** dialog (Figure 103).

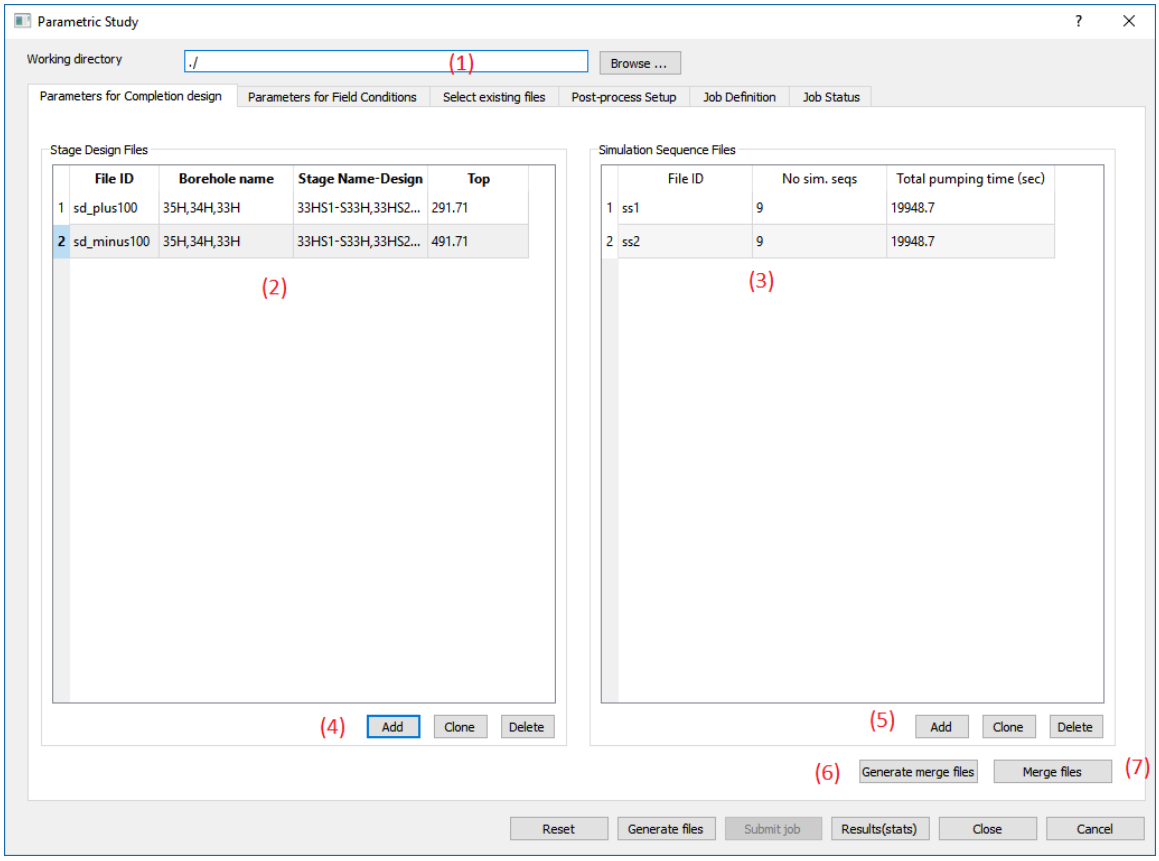


Figure 103 Dialog box for parametric study set up.

The tab **Parameters for Completion design** has the following elements:

1. The **Working directory** in field (1) specifies where configuration files will be created.
2. The left side panel, field (2), is where the user will define the completion design files. Each line on this panel represents a different stage design configuration file.
3. The right panel, field (3), is for defining the simulation sequence files. Each line on this panels represents a different simulation sequence configuration file.
4. The set of buttons in field (4) will add, clone, and delete stage design configuration files. When the **Add** button is pressed or an item in the panel is doubled-clicked in this panel, the dialog displayed in Figure 104 is showed. This dialog will be discussed in this appendix.
5. The set of buttons (5) will add, clone, and delete simulation sequence configuration files. When the **Add** button is pressed or an item in the panel is doubled-clicked, the dialog displayed on Figure 105 is showed. This dialog will be discussed in this appendix.
6. After the configurations for stage design and/or simulation sequence files are set, if the **Generate merge files**, field (6), button is pressed, the program will create all the configuration files defined in both panels.
7. Button **Merge files** in field (7) will invoke the dialog box shown in Figure 106. A merged file is the combination of a stage design and/or a simulation sequence file with the current

configuration file. The user may load and inspect the merged file to verify if the desired configuration is correct. This dialog is discussed in this appendix.

Creating Stage Design Files

The dialog shown in Figure 104 will be displayed when the user presses the **Add** button or double-clicks on one of the items of the stage design panel of Figure 103. This dialog is used to associate a file name with a set of stage designs and corresponding boreholes. The user must enter a unique file ID in field (1). This value will be used to compose the name of a stage design configuration file. The stage design configuration file names will be prefixed by "sd_" and followed by the ID value specified in field (1). For example, using the file ID value of "test1", the file name will be "sd_test1.xml". The file is in XML format. Each item in list **Stages**, field (2), is the definition of a set of stages that uses a stage design previously defined in the resources database, associating the stage with a previously defined borehole. The first cluster of this stage will be created at the measured depth specified for this item. The following clusters will be created according to the "spacing between clusters" value specified in the stage design resource record. You may select a stage design that has been previously assigned to a well trajectory and apply it to a borehole by using combo boxes (3) and (4) and pressing the button **Get Stage Design**. This action will copy all stages defined for a well trajectory to the selected borehole. You may slide the location of all clusters of a stage in the borehole by selecting a line in the Stages list, entering a value that you want to slide and pressing the **Slide on selected** button (5).

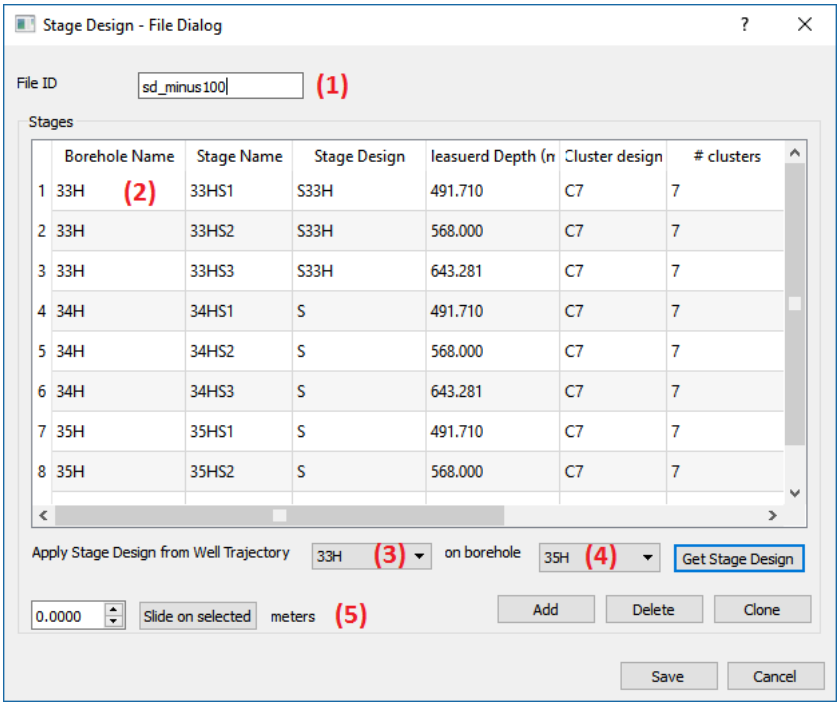


Figure 104 Dialog box for defining a stage design configuration file.

By pressing the **Add** button or double-clicking a line in the **Stages** list of Figure 104, the dialog in Figure 105 will be shown. In this dialog, the user will be able to associate a stage design to a borehole. A borehole can be selected from the drop-down box (1). The user must enter a stage name in field (2) and select a stage design resource from the drop-down box (3). Finally, in field **Measured Depth** (4), the user must enter the location relative to the start of the borehole where the first cluster of the stage will be created.

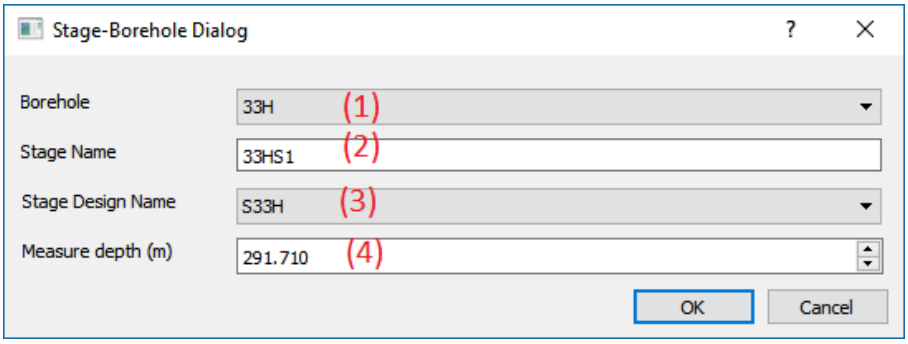


Figure 105 Stage design - Borehole dialog.

Creating Simulation Sequence Files

The dialog shown in Figure 106 will be displayed when the user presses the **Add** button or double-clicks on one of the items of the simulation sequence panel [field (3) of Figure 103]. This dialog is used to associate a file name with a set of simulation sequences and corresponding boreholes and stages. The

user must enter a unique file ID in field (1). This value will be used to compose the name of a stage design configuration file. The stage design configuration file names will be prefixed by "ss_" and followed by the ID value specified in field (1). For example, using the value specified in Figure 106, the file name will be "ss_var_2.xml". The file is in XML format. Each item in the list of simulation sequences, field (2), is the definition of a simulation sequence that uses a simulation sequence previously defined in the resources database that is associated with a previously defined stage of a borehole. You may copy the simulation sequence records defined in the current model by pressing the **Copy from model**, button (3).

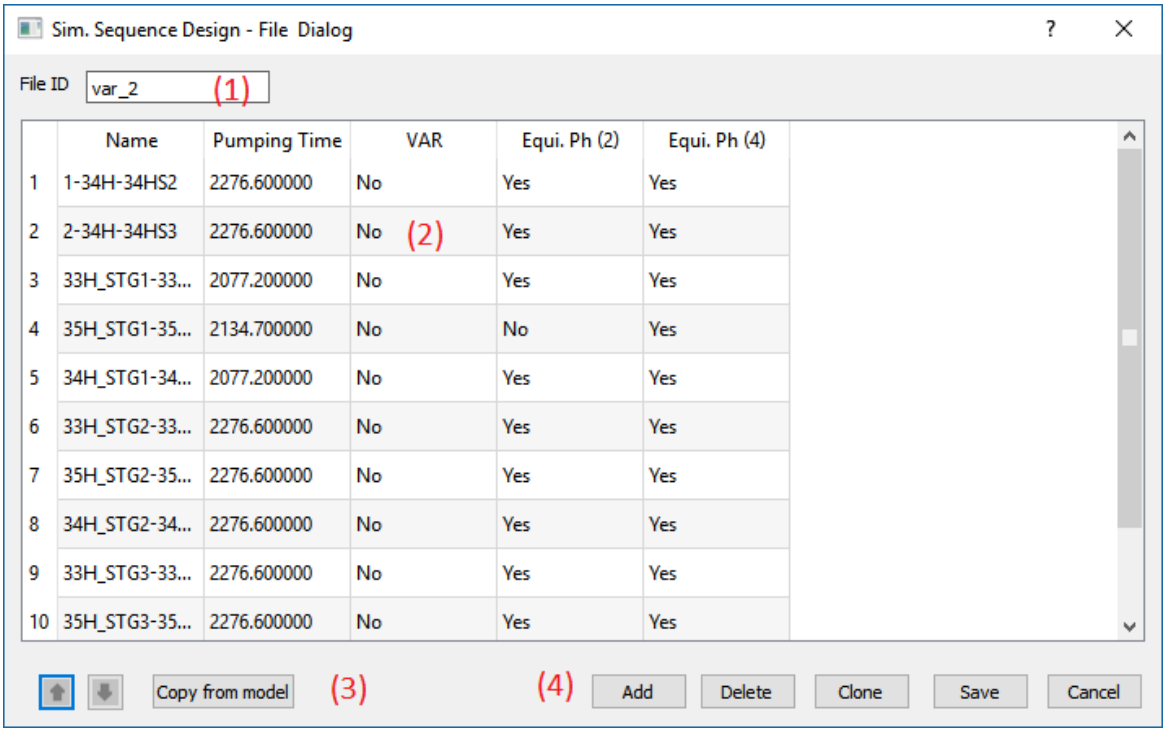


Figure 106 Simulation sequence – file dialog.

By pressing the **Add** button (4) a dialog box containing a drop-down box displaying all **Simulation Sequence** items defined in the resources database will be shown. The user may select one of the predefined simulation sequence records from this drop-down box. If the user clicks "OK", the dialog in Figure 107 will be shown. This dialog may be also invoked by double-clicking a line in the list of simulation sequences in Figure 106. In this dialog, the user will be able to associate a pumping schedule previously defined in the resources database to a stage of a given well. This is accomplished by either double-clicking one line of the pumping schedule list or pressing the **Add** button, highlighted in Figure 106. Either action will invoke the well/stage pumping schedule dialog shown in Figure 107. Note that the user must create the association of a pumping schedule with a stage for the simulation sequence file to be created.

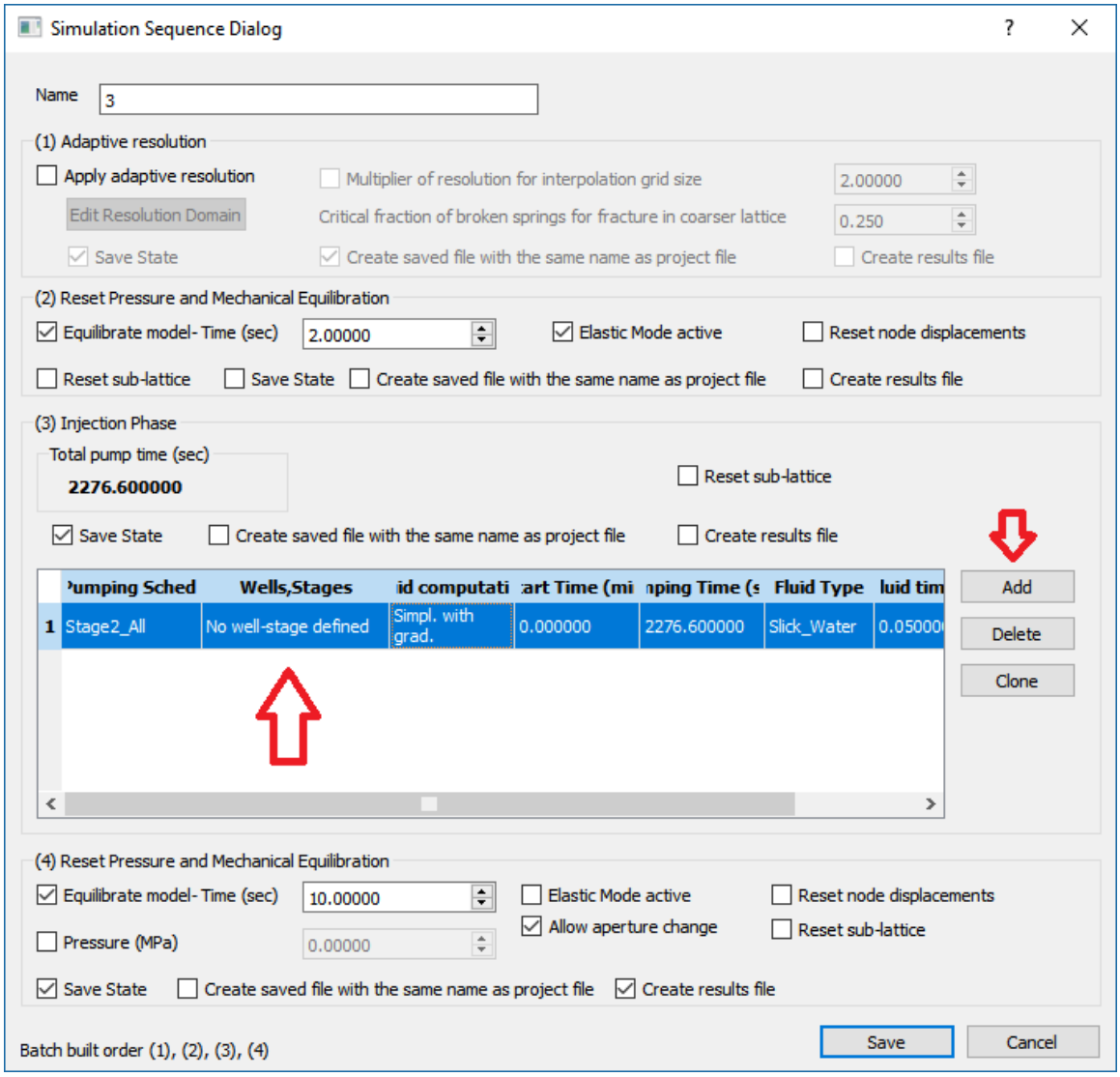


Figure 107 Simulation sequence dialog.

All pumping schedule records previously defined in the resources database will be available for selection in this dialog in the **Pumping Schedule** drop-box (1). Once the pumping schedule is selected, the user may retrieve the default values (e.g., pumping time) of the selected pumping schedule by pressing the **Apply Default** button (4). Finally, the user must associate a stage to a pumping schedule by pressing

the **Add** button (6). This association may be changed by double-clicking one of the lines of the **Well/Stage selection** list.

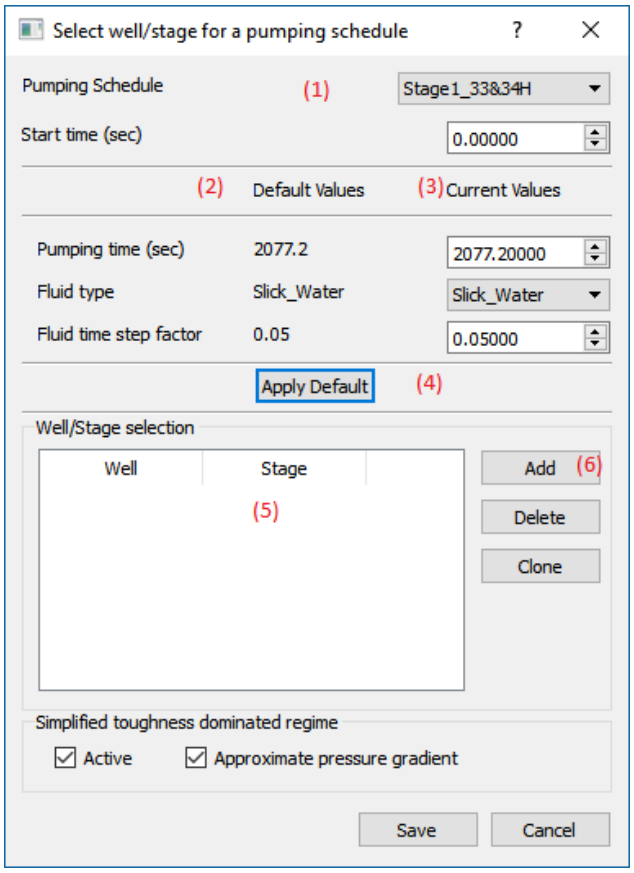


Figure 108 Stage-Pumping Schedule dialog.

Merging Configuration Files

After creating stage design and simulation sequence configuration files, the user may merge these data to a full model configuration file creating a new configuration file. The user may select to use only a stage design file, only a simulation sequence file, or both. If a stage design configuration file is selected before creating the new stages, the program will delete all clusters, built-in fluid joints, resolution domains, and histories from the full configuration file. If a simulation sequence file is used, all existing pumping schedule and batch items will be deleted from the full configuration file.

The name of the new full configuration file will be composed by concatenating the names of stage design file and the simulation sequence file after the name of the current full configuration file. For example, if the full configuration file is named “test.xml”, the stage design file is named “sd1.xml” and the simulation sequence file is named “ss1.xml”, the new full configuration file will be named “test_sd1_ss1.xml”.

Merging the configuration files can be accomplished by pressing the **Merge files** button of the **Parameters for Completion Design** tab located on the **Parametric Study Setup** dialog (Figure 103) or by specifying parameters in the command line.

Merging files

When the user presses the **Merge File** button in Figure 103, the **Select files for merging** dialog shown in Figure 109 will be displayed. Each line added to the list of selected files will result in the creation of a new full configuration file. In the example shown in the dialog in Figure 109, three new configuration files will be created. The user will enter a stage design configuration file name in field (3) and a simulation sequence file name in field (4). After fields (3) and/or (4) are populated, the user will press the **Add row** button (5) to create a new line in the selected files list.

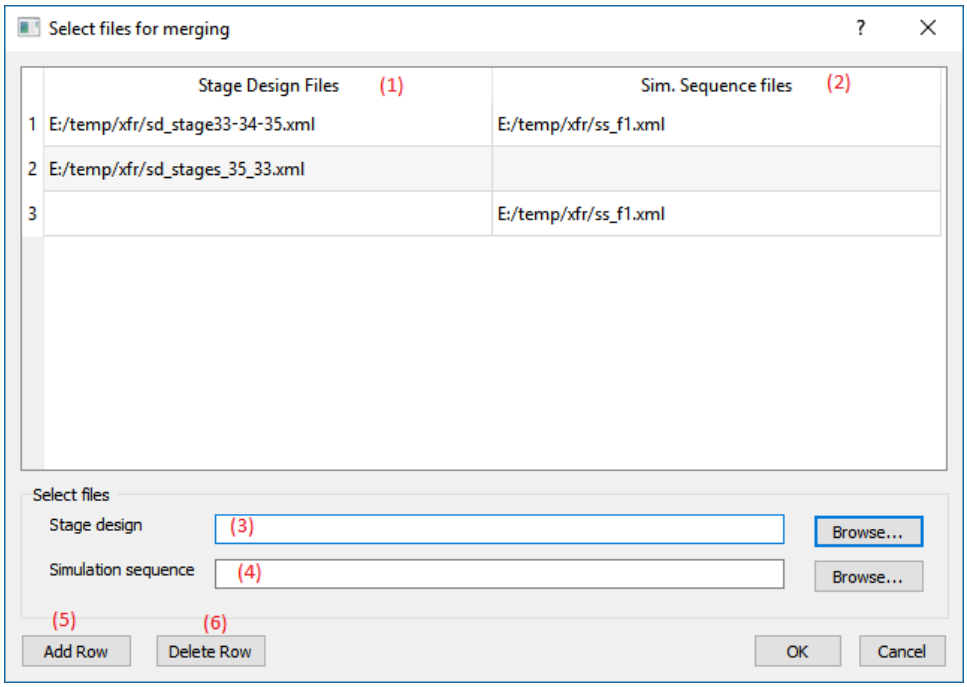


Figure 109 Select files for merging dialog.

Appendix 3 – Setting Up Multistage Models

A3.1 Introduction

XSite allows creation of different fracture design scenarios utilizing different completion designs and combining them into different new models.

The example provided has three wells (W1, W2, and W3). Each well has will have three stages with seven clusters each. They will be stimulated in the order W1-stage1, W2-satge1, W3-stage1, etc.

A3.2 Workflow

The workflow to create a multistage model is the following:

1. Add a **Sketch Model** plot item.
2. Create model geometry, adding seams, DFN's, stresses, etc.
3. Create **Cluster Design** (**Resources Menu**) items.
4. Create **Stage Design** (**Resources Menu**) items utilizing **Cluster Design** items created in step (3).
5. Save your model.
6. Create well trajectories (**Hydraulic Fracture/Trajectory** tab). Fill the well origin information and add the segments that make up the well. The segments of a well trajectory can be either defined in measure depth or true vertical depth. For each stage in the well, associate a stage design to well trajectory.
7. Build a borehole by selecting a well trajectory and pressing the build button in the top of the **Hydraulic Fracture/Trajectory** tab. Select either measured or true vertical depth, depending how the well trajectory segments were defined. In the example provided measured depth was used.
   When a borehole is constructed, adaptive resolution domains, histories and initial joints will be added to the model if these features were defined in the cluster and stage design items used to create the boreholes.
   The borehole pumping schedule will be created later when you build the simulation sequence.
8. Save your work if you are pleased with the location of boreholes and clusters in the model.
9. Create **Pumping Schedule** (**Resources Menu**) items.
10. Create **Simulation Sequence** (**Resources Menu**) items with **Pumping Schedule** items created in (8).
11. Create **Simulation Sequence** items (**Hydraulic Fracture/Sim. Sequence** tab) in the order that the stages will be stimulated. The simulation sequence items are used to create the borehole pumping schedule and the corresponding batch simulation items. If the model has three wells with three stages each, it is necessary to create nine simulation sequence items. After all **Simulation Sequence** items are created, press the button **Build** at the top of the tab. This

action will create pumping schedules for all boreholes and batch simulation items that will be executed. The Simulation Sequence dialog is discussed in Section A3.3.

12. Save your model. The model is ready to start a simulation.

A3.3 Simulation Sequence Dialog

The Simulation Sequence dialog (Figure 110) has four execution blocks, (1) **Adaptive Resolution**, (2) **Mechanical Equilibrium**, (3) **Injection Phase** and (4) **Reset Pressure/ Mechanical Equilibrium**. These blocks may be inactive, and each block has an option to save the state of the simulation at the end of block execution. A batch simulation items will be created for each active block.

Before you construct the **Simulation Sequence**, for each sequence item, you will need to assign a well and corresponding stage by double-clicking the pumping schedule item (highlighted box in red in Figure 110). This will bring the dialog box shown on Figure 111. Confirm the **Pumping Schedule** from the drop-down box. Next, press the **Add** button and a new dialog will be displayed allowing you to select the well and the stage that will be simulated.

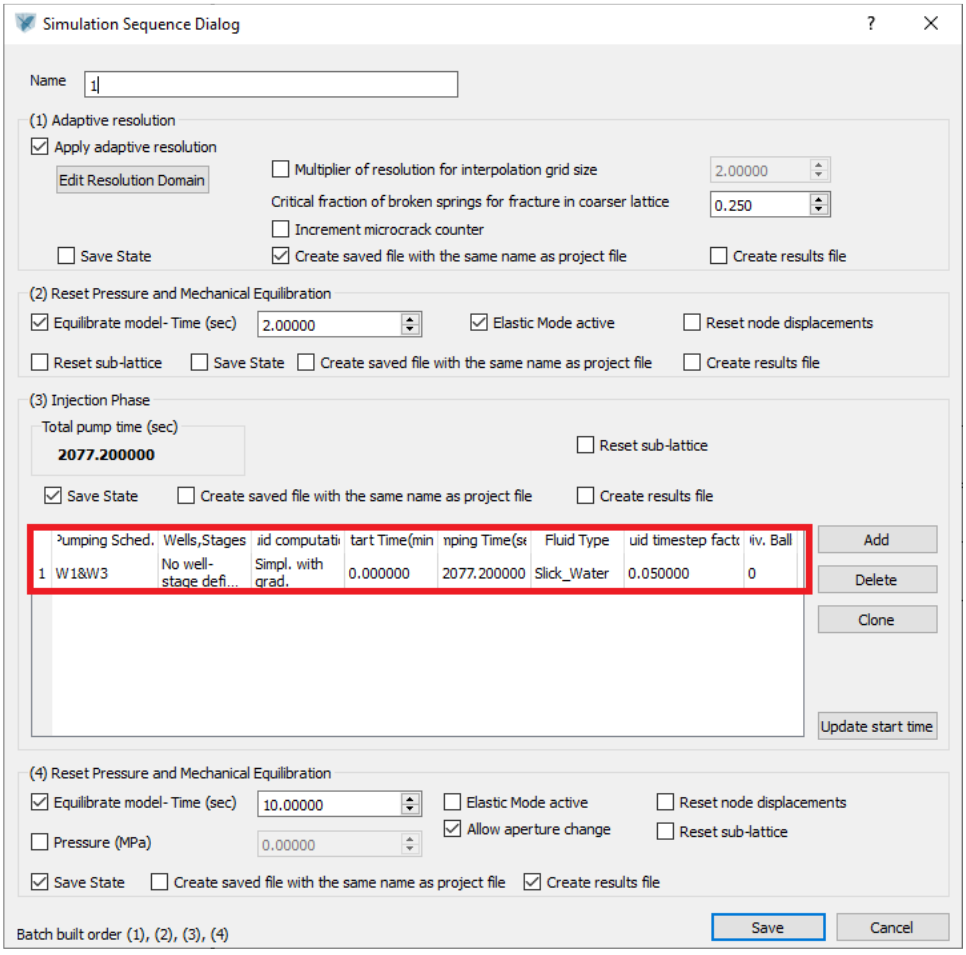


Figure 110 Simulation Sequence dialog.

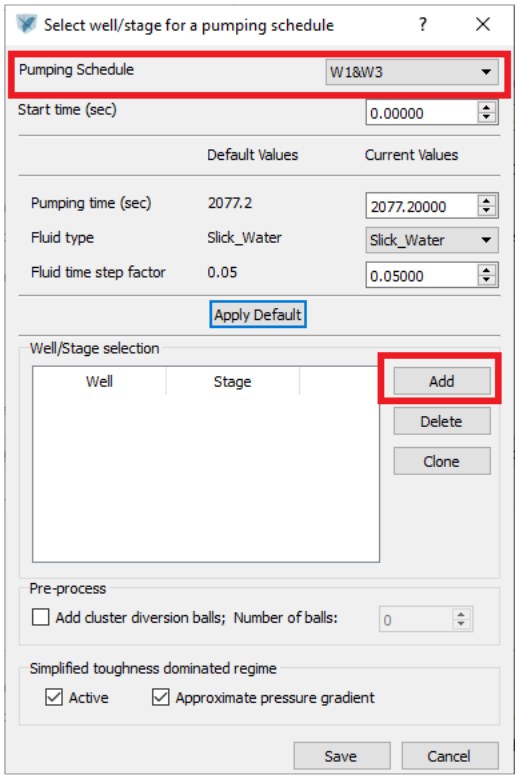


Figure 111 Well/Stage assignment.

8.0 Tutorial Examples

This section contains two tutorial examples to introduce users to XSite. Example 1 simulates the response of intact homogeneous rock (using the Test Rock in the material database) to water injection (Figure 112a), whereas three discrete joints that are free to interact with any induced fractures (Figure 112b) are added in Example 2. Please refer to Section 16.1 in XSite, Description of Formulation (Itasca, 2023) for further details and discussion on the results of these simulations.

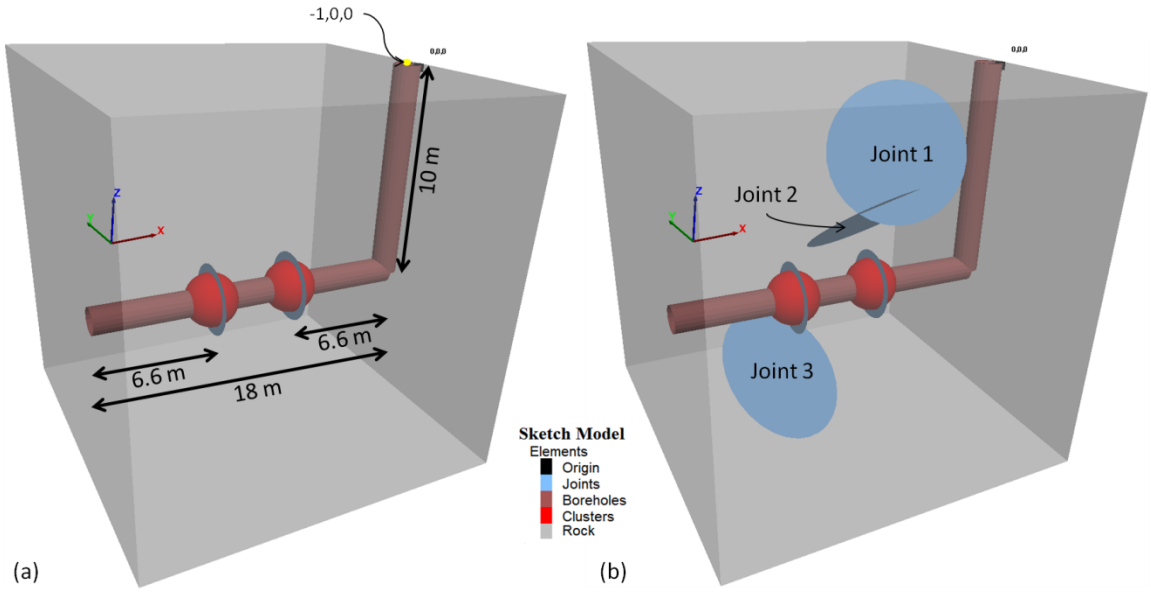


Figure 112 Sketch of example models (a) without and (b) with joints

In both examples, the model domain is 20 m on each side and contains a borehole with both a vertical and a horizontal section, with the latter containing two 1.5-m-radii injection clusters spaced 6.6 m from either end of the horizontal section. A lattice resolution of 0.5 m has been selected, and water is to be injected simultaneously from each cluster at a rate of 0.01 m^3/s. In order to initiate the fluid calculation, 2-m-radii water-filled joints have been placed at the center of each cluster, perpendicular to the horizontal borehole; their initial joint aperture is assumed to be 0.1 mm. Tables 3 and 4 contain the physical properties for both water and the intact rock (e.g., Test Rock).

The assumed stress state for both examples is anisotropic, with σ_{xx} = 1 MPa, σ_{yy} = 12 MPa and σ_{zz} = 10 MPa. It should be noted that the stress magnitudes were selected for tutorial purposes only and are not typical of reservoir conditions. However, as the initial pore pressure is assumed to be zero, the model is representative of the effective stresses and net pressures typical of reservoir simulations. Because the least principal stress is aligned with the horizontal section of the borehole, crack propagation in the direction normal to the horizontal section of the borehole is anticipated.

Table 3 Tutorial example: Water properties

Property	Value
Density (kg/m^3)	1000
Bulk Modulus (GPa)	2.2
Flow Behavior Index	1
Viscosity (Pa × s)	0.001
Thermal Conductivity (W/m × °C)	0.58
Specific Heat (J/kg × °C)	4184.0
Thermal Expansion Coefficient (1/°C)	2.07e-04

Table 4 Tutorial example: Rock material properties

Property	Value
Density (kg/m^3)	2650
Young's Modulus (GPa)	70
Poisson's Ratio	0.25
UCS (MPa)	200
Tensile Strength (MPa)	20
Fracture Toughness (Pa×m$^{1/2}$)	1.0e+06
Damping	0.1
Friction Angle (°)	25.0
Porosity (%)	20.
Permeability (m2)	1.0e-13
Thermal Conductivity (W/m×°C)	3.0
Specific Heat (J/kg×°C)	795
Thermal Expansion Coefficient (1/°C)	8.0e-06

Simulations will be run for 15 seconds in coupled mode (i.e., mechanical and fluid modes active) *after* being run 0.1 second in mechanical mode only (to ensure initial model equilibrium). Crack (i.e., damage)

counts and acoustic emissions will be tracked during simulation. The solution should include the Spin on scheme and active fluid flow. Use default values for all other settings.

As shown in Figure 112b, Example 2 adds three discrete fractures (Table 5), which are free to interact with the propagating fractures, to the scenario of Example 1.

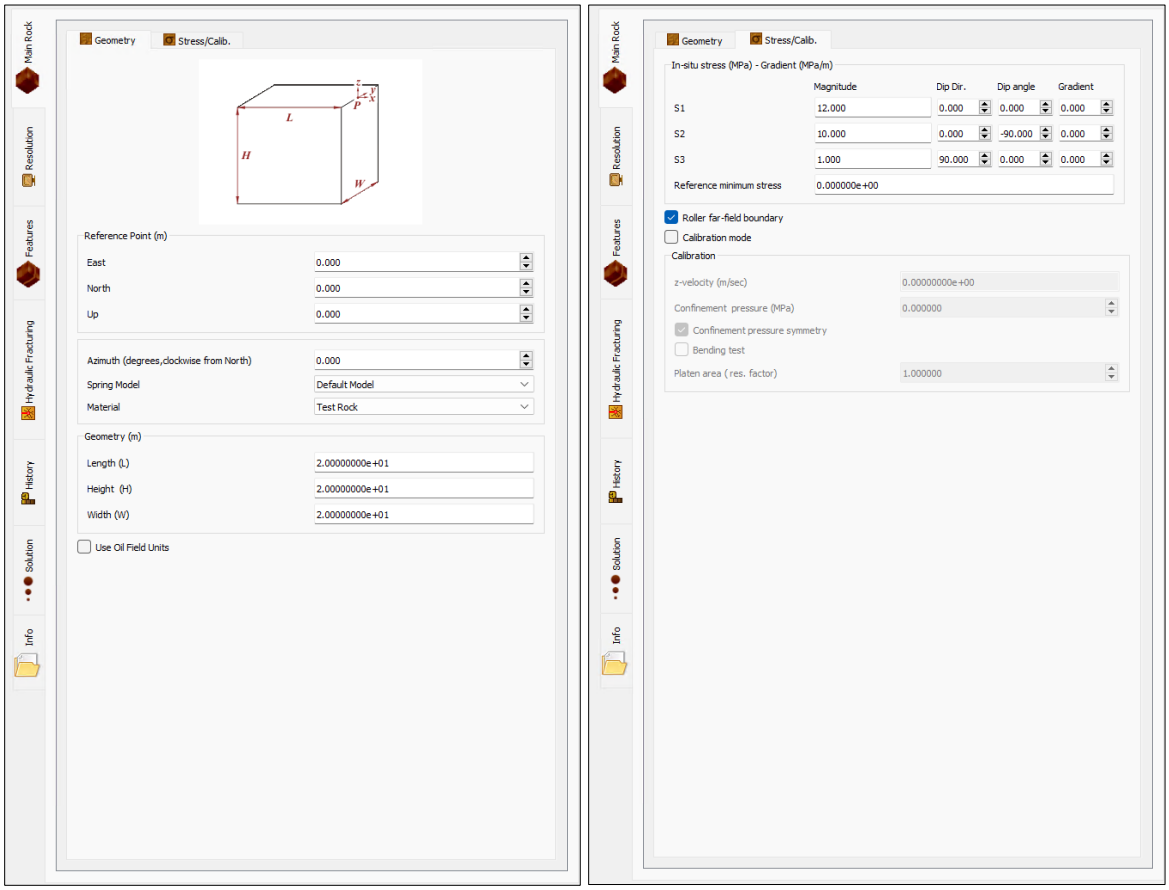
Table 5 Joints in the fractured rock

Joint ID	1	2	3
Dip (°)	-70	37	45
Dip Direction (°)	20	342	-215
Radius (m)	3.6	4.0	4.0
Easting/X (m)	-7.6	-7.6	-13.0
Northing/Y (m)	-2.0	0.0	0.0
Up/Z (m)	-4.0	-8.0	-16.5

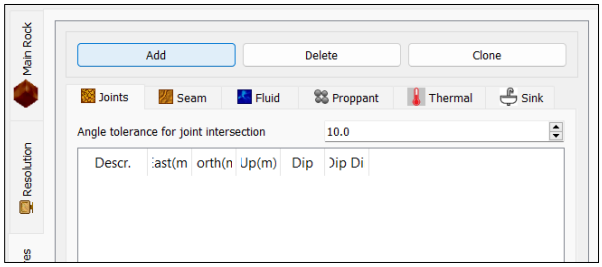
Note: Assumed that normal and shear stiffnesses are zero.

Example 1: Intact Rock

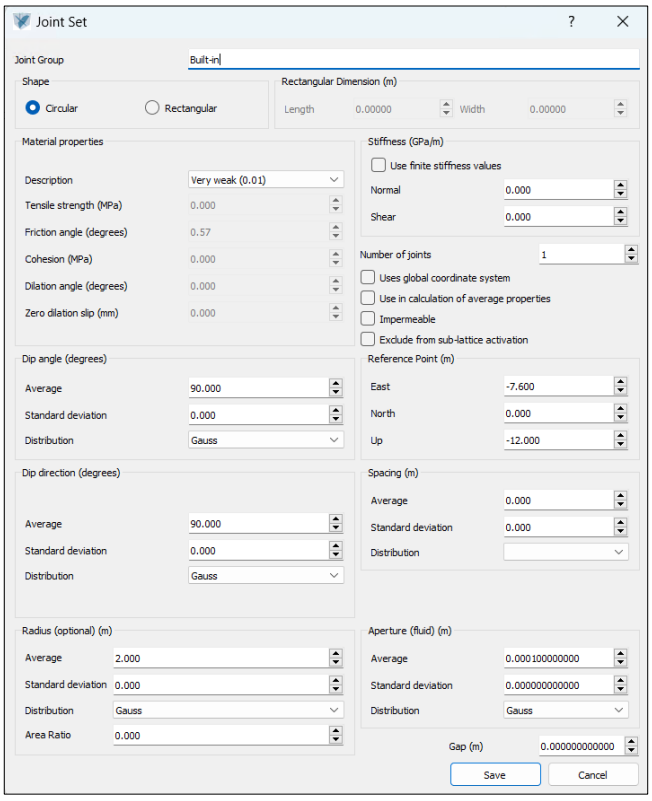
1. Define model geometry, material, in-situ stress conditions and boundary conditions using the **Main Rock** Tab as shown below.

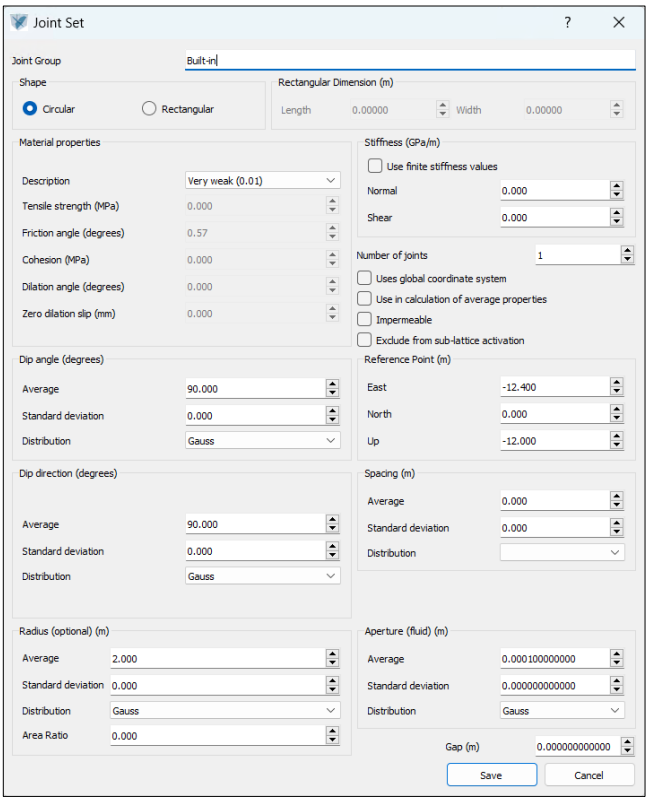


2. Although there are no rock joints per se in Example 1, it is useful to insert a joint at each cluster location in order to initiate fractures. Use the **Features** tab and click the **Add** button shown below.

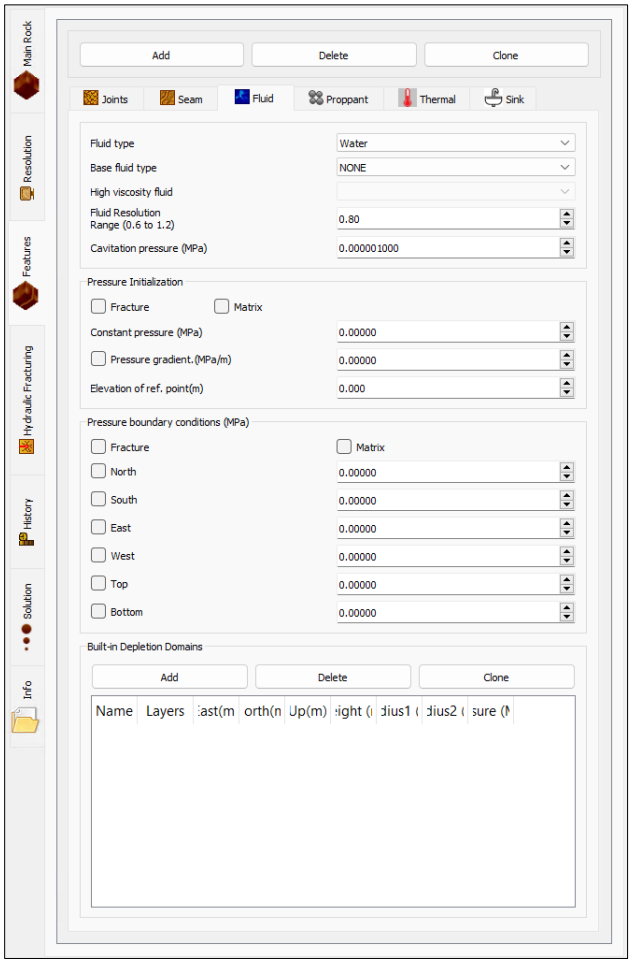


3. Both of these 4-m-wide, very weak injection joints will be positioned perpendicular to the borehole as defined below. Each joint will be added one at a time.

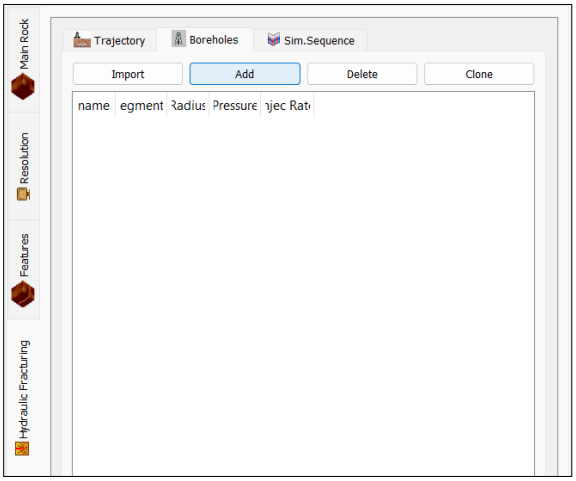




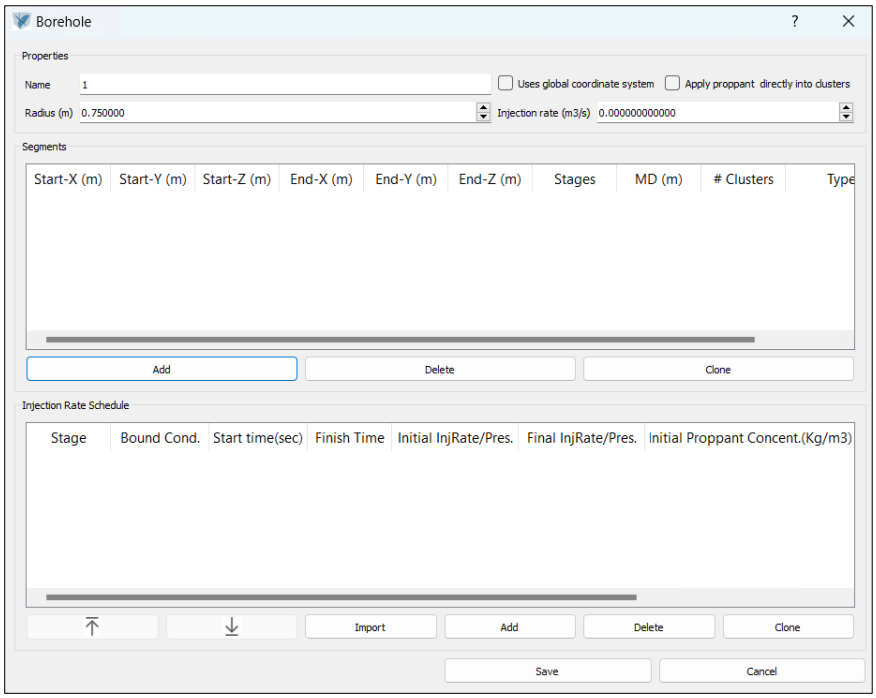
4. As Example 1 consists of a single rock material, no seams are necessary, and the **Fluid** tab under the **Features** tab now can be selected to choose the type and parameters of injection fluid (e.g., water) to be used to fracture the rock.



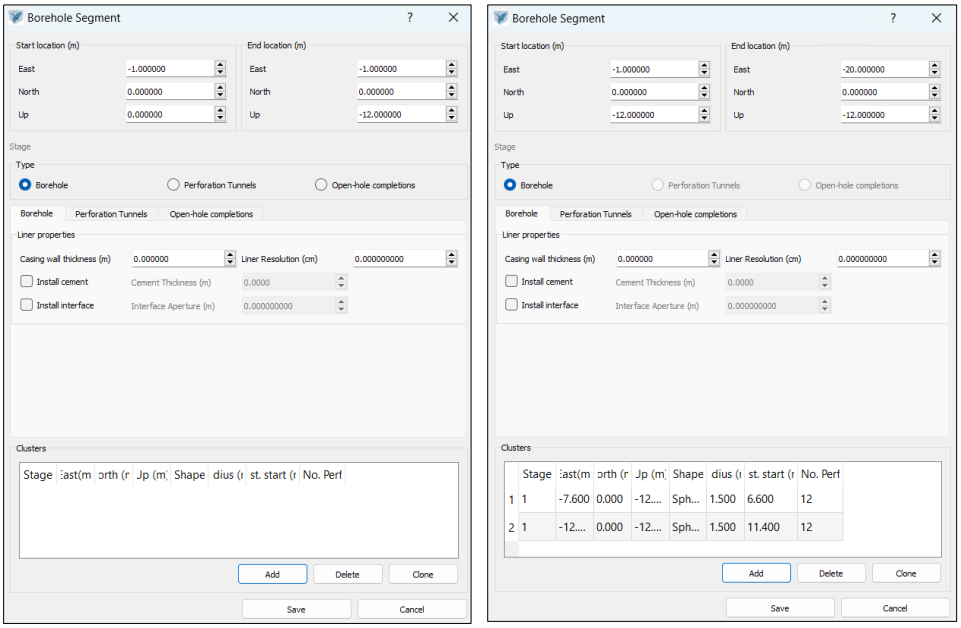
5. The borehole can be defined using the **Hydraulic Fracturing** tab shown below and clicking on the **Add** button.



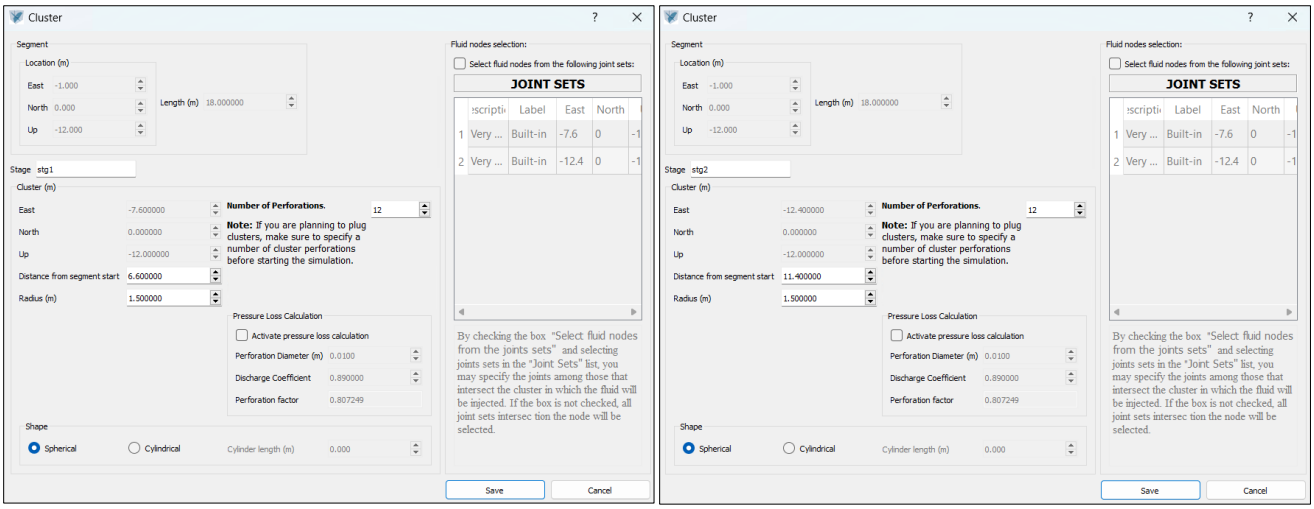
6. The borehole dialog will appear, and the radius can be entered. Note that injection rate for this example will be specified later at the clusters and should not be entered in this dialog, as this is the injection rate at the start of the borehole. To define the horizontal and vertical segments of the borehole, click the **Add** button highlighted blue below.



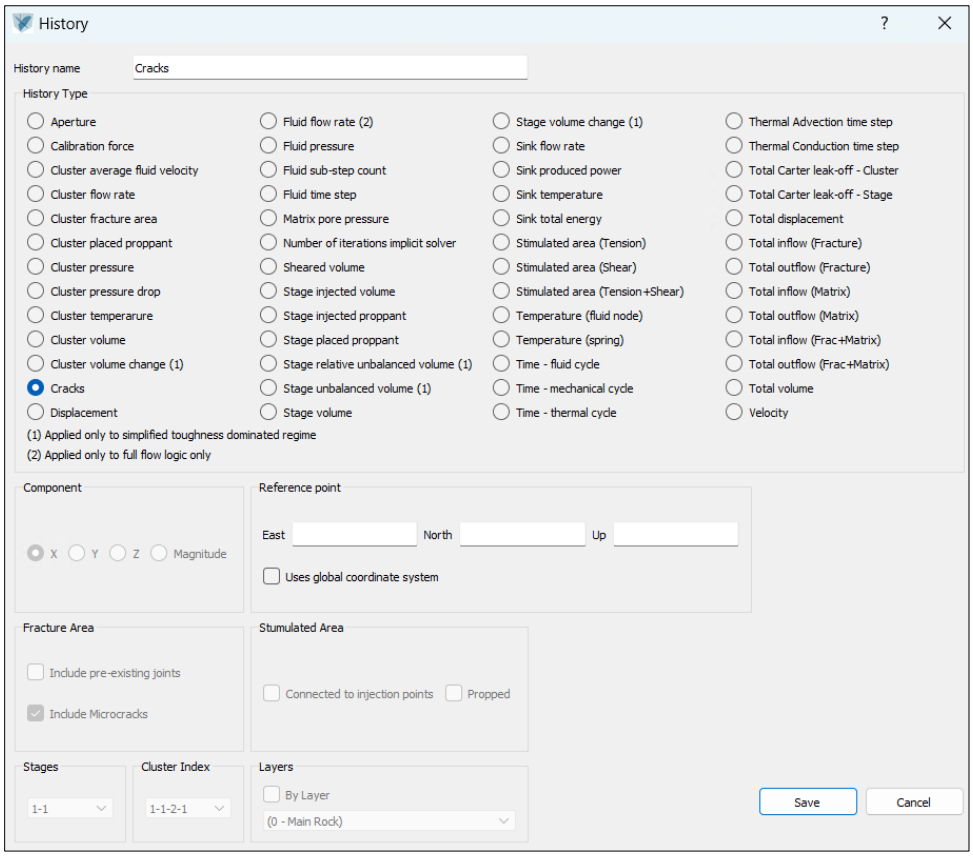
7. The borehole segment dialog will appear, allowing the start- and end- locations of the segment to be defined. The vertical segment is shown below on the left-hand side, the horizontal segment on the right-hand side. In order to add the injection clusters along the horizontal segment, click the **Add** button highlighted blue in the right-hand side shown. This is not required for the vertical segment, as it does not contain any clusters.



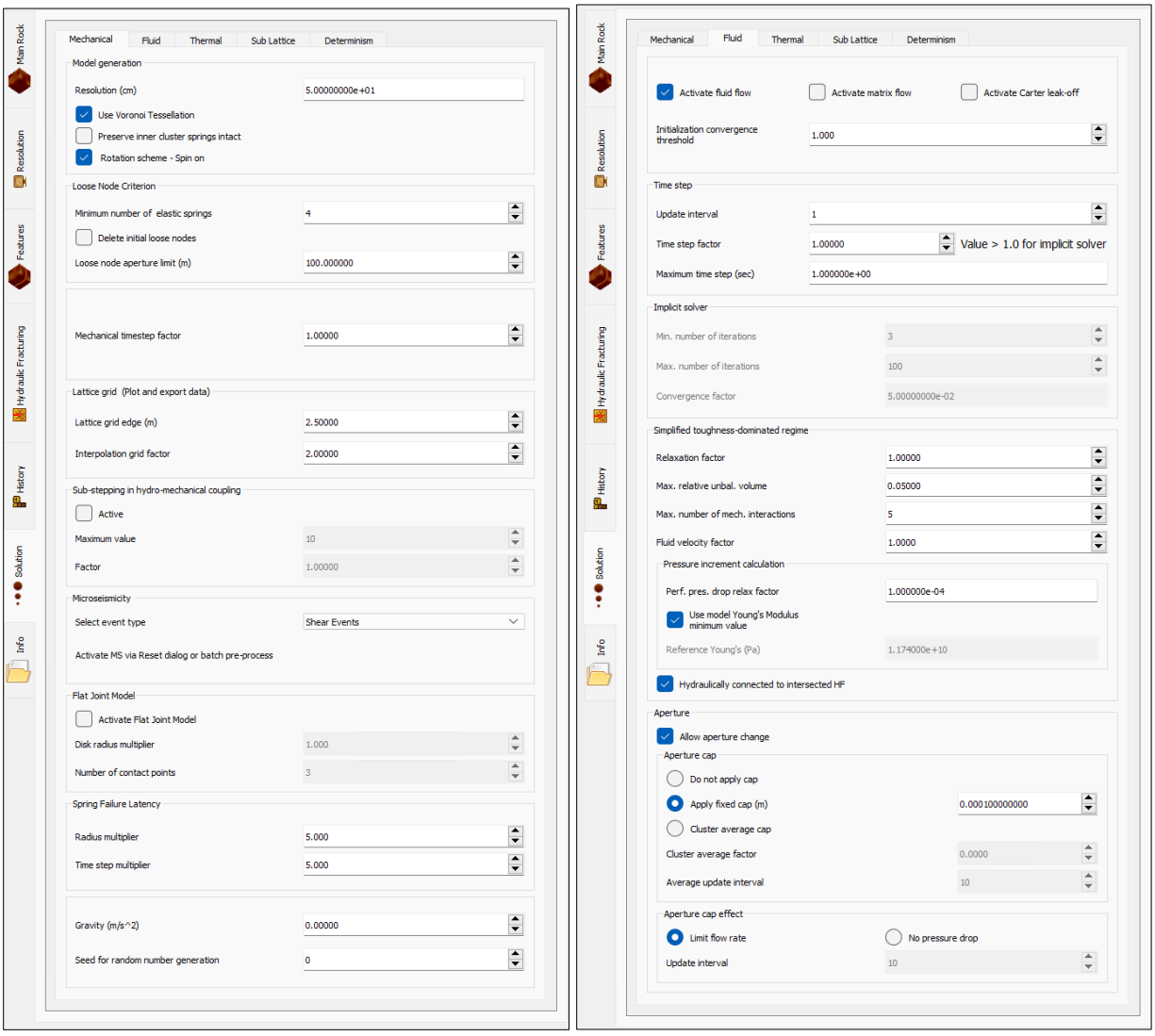
8. A cluster dialog will appear, and the clusters can be defined, one at a time, as shown below. The *relative* position of the cluster (with respect to the start of the borehole segment) can be specified along with the cluster radius and the injection rate at the cluster.



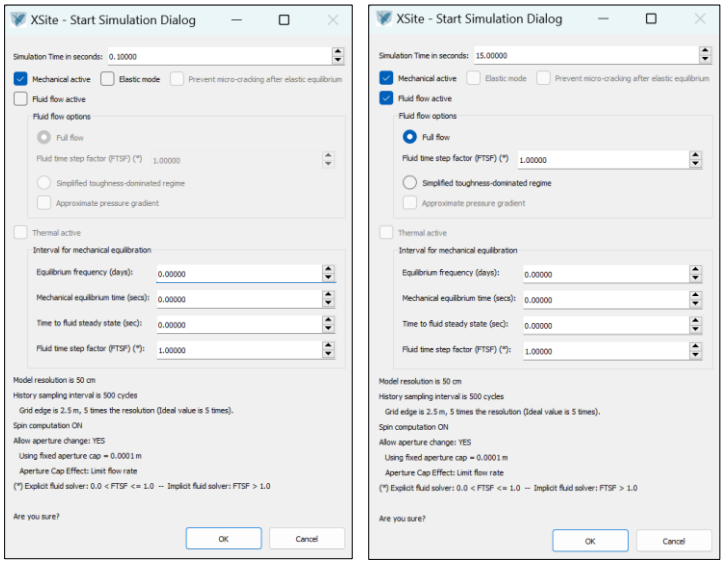
9. Because it is of interest to estimate the damage, caused by fluid injection, to the rock matrix, the crack tracking tool should be activated as shown in the History dialog, as follows. Although not shown, an alias for any history item can be specified. Click **Save** for each dialog; **Cancel** will clear any changes made.



10. Finally, use the Solution tab to specify the model resolution, rotation scheme, fluid flow and acoustic emissions settings as shown below.



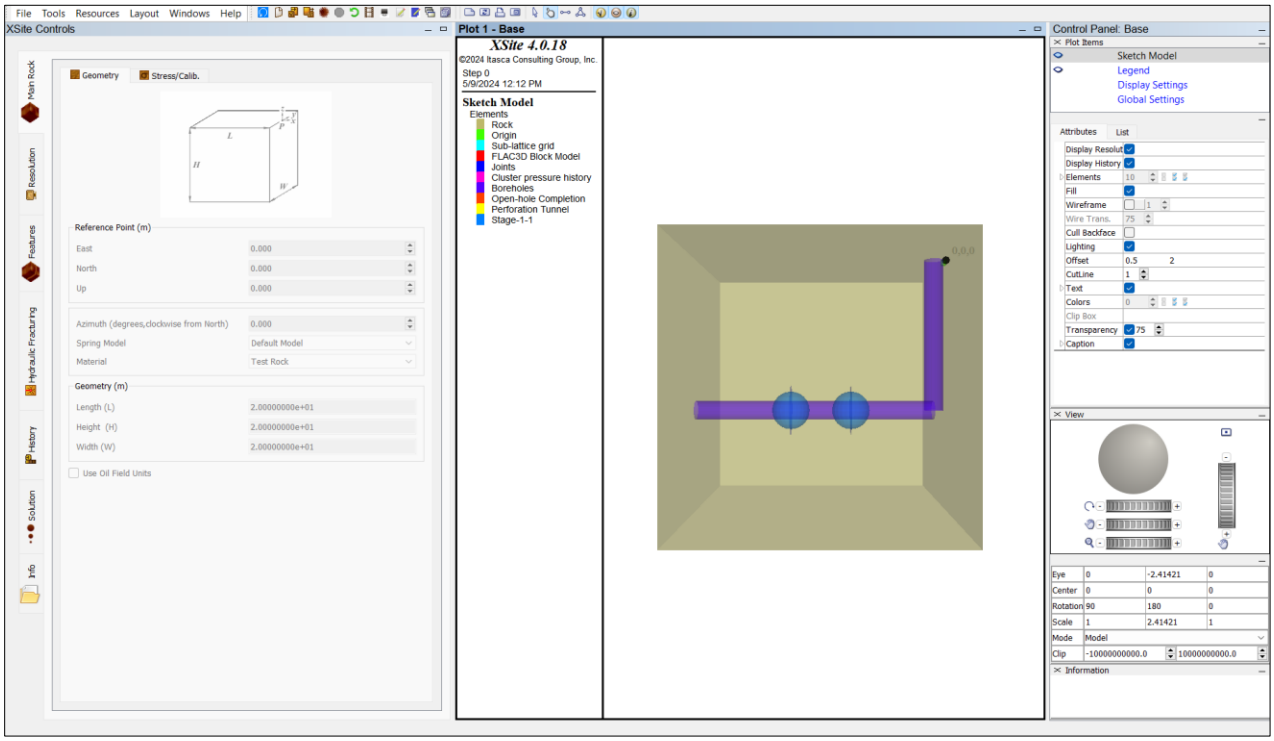
11. To start the simulation, click on the **Simulate** icon . If this icon is not visible, click anywhere on the Simulation control panel to change the contextual menu from plotting to simulation mode. Run an initial mechanical simulation for 0.1 second, followed by a coupled hydro-mechanical simulation for 15 seconds more. Be sure to check **Fluid active** for the second simulation.

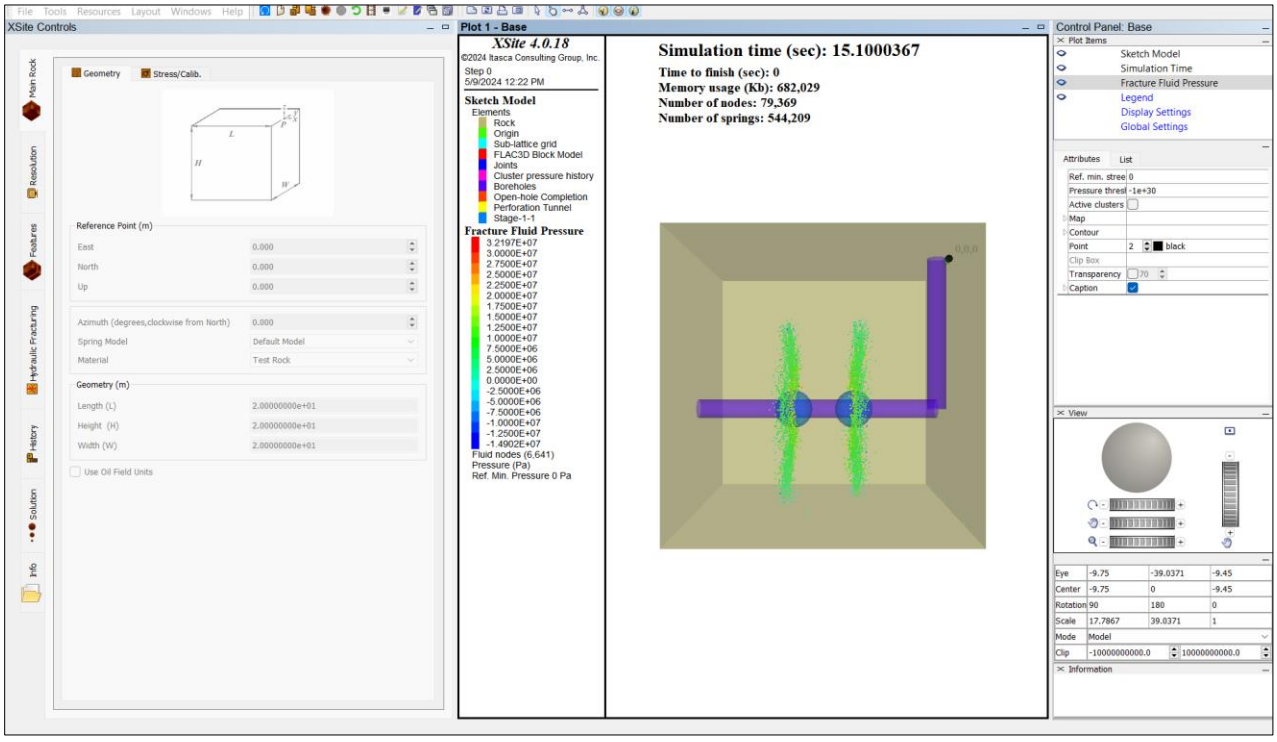


12. The dialog box on the left-hand side should appear briefly as the lattice model is generated, followed by the dialog box on the right-hand side, which will remain visible and indicate simulation status for the duration of the run. To stop the simulation, click the **Stop Simulation** icon .

13. Any plot item can be enabled and modified after, or during, the simulation. For example, click on the **Sketch Model** plot item to display a plot of the main elements of Example 1 (below); then click on the **Fluid** plot-item to reveal a number of items. Select the Fracture Fluid Pressure plot item to display the following final figure.

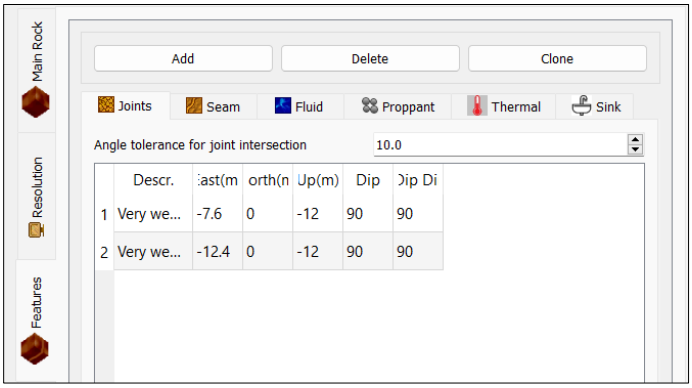
Once the simulation is complete, be sure to save your project, data file and/or model state.


File Tools Resources Layout Windows Help
XSite Controls
Main Rock
Resolution
Features
Hydraulic Fracturing
History
Solution
Info
Geometry
Stress/Calib.
Reference Point (m)
East 0.000
North 0.000
Up 0.000
Azimuth (degrees,clockwise from North) 0.000
Spring Model Default Model
Material Test Rock
Geometry (m)
Length (L) 2.00000000e+01
Height (H) 2.00000000e+01
Width (W) 2.00000000e+01
Use Oil Field Units
Plot 1 - Base
XSite 4.0.18
©2024 Itasca Consulting Group, Inc.
Step 0
5/9/2024 12:12 PM
Sketch Model
Elements
Rock
Origin
Sub-lattice grid
FLAC3D Block Model
Joints
Cluster pressure history
Boreholes
Open-hole Completion
Perforation Tunnel
Stage-1-1
Control Panel: Base
Plot Items
Sketch Model
Legend
Display Settings
Global Settings
Attributes
List
Display Resolut
Display History
Elements 10
Fill
Wireframe 1
Wire Trans. 75
Cull Backface
Lighting
Offset 0.5 2
CutLine 1
Text
Colors 0
Clip Box
Transparency 75
Caption
View
Eye 0 -2.41421 0
Center 0 0 0
Rotation 90 180 0
Scale 1 2.41421 1
Mode Model
Clip -10000000000.0 10000000000.0
Information

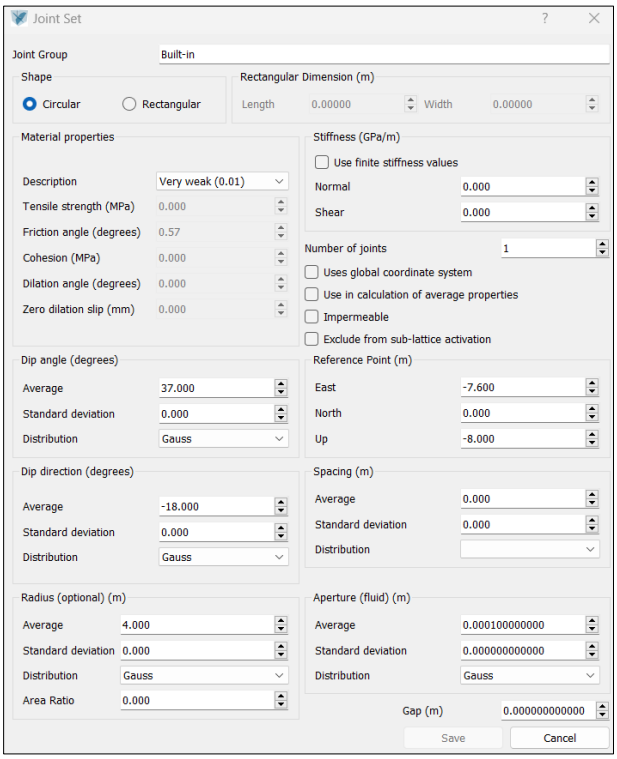

File Tools Resources Layout Windows Help
XSite Controls
Main Rock
Resolution
Features
Hydraulic Fracturing
History
Solution
Info
Geometry
Stress/Calib.
Reference Point (m)
East 0.000
North 0.000
Up 0.000
Azimuth (degrees,clockwise from North) 0.000
Spring Model Default Model
Material Test Rock
Geometry (m)
Length (L) 2.00000000e+01
Height (H) 2.00000000e+01
Width (W) 2.00000000e+01
Use Oil Field Units
Plot 1 - Base
XSite 4.0.18
©2024 Itasca Consulting Group, Inc.
Step 0
5/9/2024 12:22 PM
Simulation time (sec): 15.1000367
Time to finish (sec): 0
Memory usage (Kb): 682,029
Number of nodes: 79,369
Number of springs: 544,209
Sketch Model
Elements
Rock
Origin
Sub-lattice grid
FLAC3D Block Model
Joints
Cluster pressure history
Boreholes
Open-hole Completion
Perforation Tunnel
Stage-1-1
Fracture Fluid Pressure
3.2197E+07
3.0000E+07
2.7500E+07
2.5000E+07
2.2500E+07
2.0000E+07
1.7500E+07
1.5000E+07
1.2500E+07
1.0000E+07
7.5000E+06
5.0000E+06
2.5000E+06
0.0000E+00
-2.5000E+06
-5.0000E+06
-7.5000E+06
-1.0000E+07
-1.2500E+07
-1.4902E+07
Fluid nodes (6,641)
Pressure (Pa)
Ref. Min. Pressure 0 Pa
Control Panel: Base
Plot Items
Sketch Model
Simulation Time
Fracture Fluid Pressure
Legend
Display Settings
Global Settings
Attributes
List
Ref. min. stree 0
Pressure thresl -1e+30
Active clusters
Map
Contour
Point 2 black
Clip Box
Transparency 70
Caption
View
Eye -9.75 -39.0371 -9.45
Center -9.75 0 -9.45
Rotation 90 180 0
Scale 17.7867 39.0371 1
Mode Model
Clip -10000000000.0 10000000000.0
Information

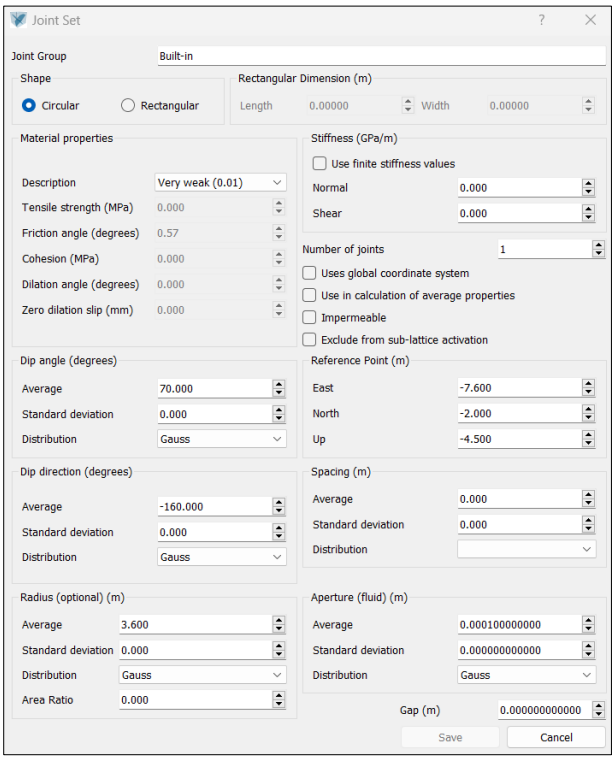
Example 2: Fractured Rock

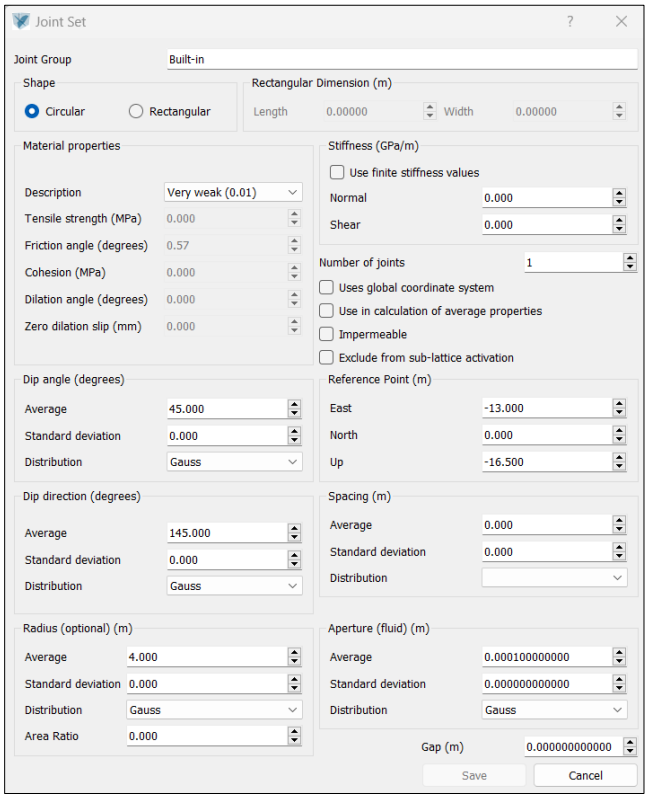
After saving Example 1, the second example can be set up simply by resetting the Example 1 model using the Reset Model icon and adding the three rock joints defined in Table 3. To insert the rock joints, click the Add button on the Joint Set tab under the Feature tab as shown highlighted blue below. Note that the previous cluster joints are still present; do not delete these.



Enter the following information for each joint, one-at-a-time, in order to specify the rock joints for Example 2.







As for Example 1, Start Simulation and run mechanically-only for 0.1 second, followed by an addition 15-second coupled hydro-mechanical simulation. Save Example 2 and compare with Example 1.

9.0 References

Crump, J.B., & Conway, M.W. (1988). Effects of perforation-entry friction on bottomhole treating analysis. *J. Pet. Technol. 40*(8). https://doi.org/10.2118/15474-PA.

Damjanac, B., Detournay, C., Cundall, P.A., Purvance, M., & Hazzard, J. (2011). XSite — Description of Formulation with Validation Examples, Revision 1. Itasca Consulting group, Inc., Report to BP ICG11-2466-15-05-43R1, July.

Hazzard, J.F., Young, R.P., & Oates, S. J. (2002). Numerical Modeling of Seismicity Induced by Fluid Injection in a Fractured Reservoir. In *NARMS-TAC 2002: Mining and Tunnelling Innovation and Opportunity*, Vol. 2, pp. 1023-1030. R. Hammah et al., Eds. Toronto: University of Toronto Press.

Itasca (2023) XSite, Description of Formulation. Itasca Consulting Group, Minneapolis, USA.

Keck, R.G., Nehmer, W., & Strumolo, G. (1992). A New Method for Predicting Friction Pressures and Rheology of Proppant-Laden Fracturing Fluids. *SPE Production Engineering*, *7*(01), 21–28.

Nolte, K.G. (1988). Fluid Flow Considerations in Hydraulic Fracturing. In Proceedings, *SPE Eastern Regional Meeting (Charleston, West Virginia, November, 1998)*. Society of Petroleum Engineers.

Pierce, M., Cundall, P., Potyondy, D., & Mas Ivars, D. (2007). A Synthetic Rock Mass Model for Jointed Rock. In *Rock Mechanics: Meeting Society's Challenges and Demands (1st Canada-U.S. Rock Mechanics Symposium, Vancouver, May 2007)*, Vol. 1: *Fundamentals, New Technologies & New Ideas*, pp. 341–349. E. Eberhardt et al., Ed. London: Taylor & Francis Group.

Schlumberger Ltd. (2011) Oilfield Glossary. https://glossary.oilfield.slb.com/

111 Third Avenue South, Suite 450
Minneapolis, MN 55401
phone: 612 371-4711
fax: 612 371-4717
email: software@itascacg.com
web: www.itascacg.com