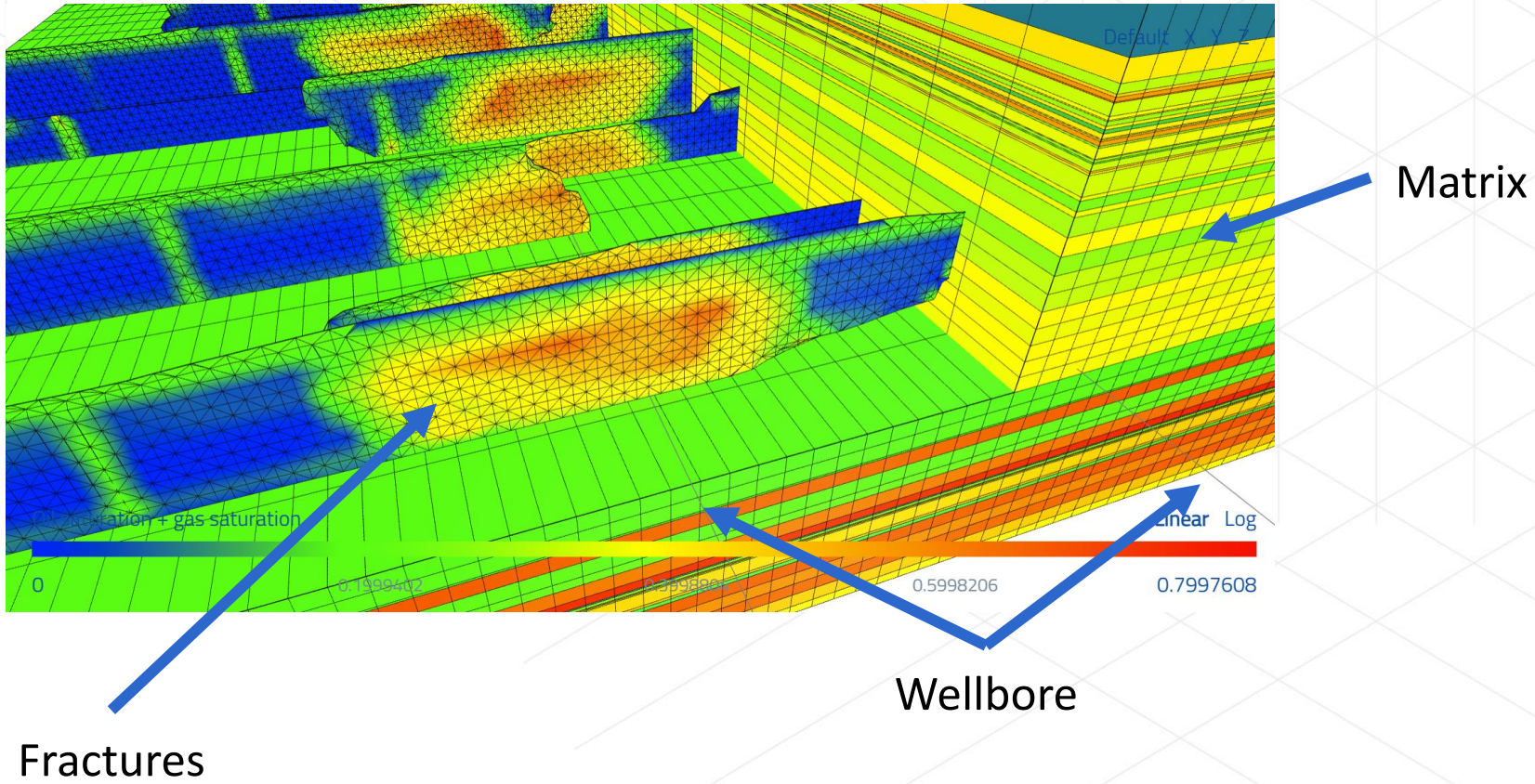
ResFrac



Meshing in ResFrac

Overview

ResFrac simulations fully mesh the matrix, fractures, and the wellbore. The wellbore is meshed to the surface. The wellbore elements are line segments, the fracture elements are rectangles, and the matrix elements are cuboids. In every timestep, the simulator solves the governing equations in every element. The mesh is nonconforming. That is, the edges of the matrix elements do not have to align with the edges of the fracture elements or the well elements.

With a conventional finite volume approach, you would need a highly refined mesh near the fracture elements in order to accurately calculate fluid exchange between the fracture elements and matrix elements (leakoff and production). However, ResFrac uses the 1D submesh method (McClure, 2017), which enables accurate calculations even if the mesh is coarse. Internal to the code, the method creates a finer mesh within each element, and then transforms it into a series of 1D problems that can be solved rapidly.

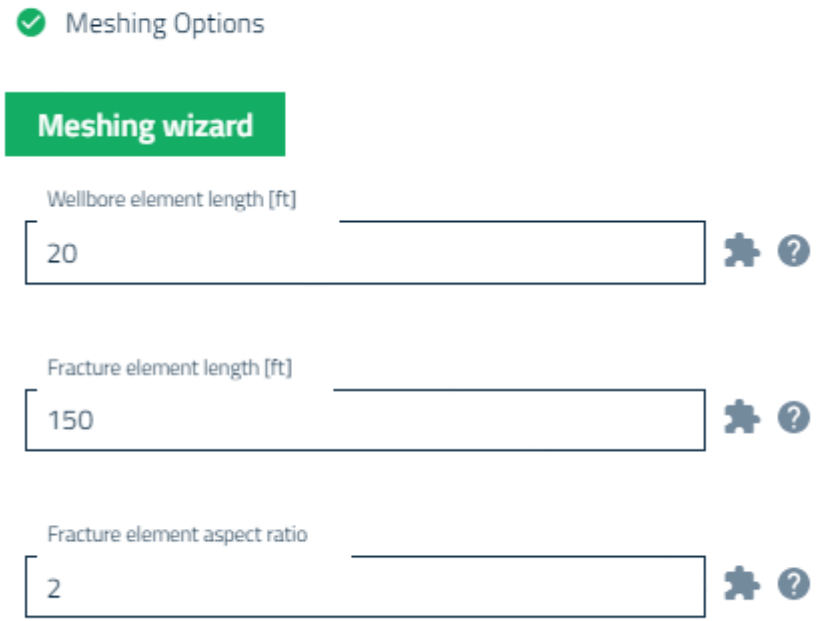
Because you can calculate fracture/matrix flow with a relatively coarse, nonconforming mesh, it is not necessary to remesh as the fractures propagate and the overall number of elements in the model is greatly reduced.

For setting up the well mesh, you specify a 'wellbore element length'. The well elements have the same length throughout the simulation, except that they may deviate somewhat in length in order to terminate at the defined well vertices. The wellbore element length should be no bigger than your cluster spacing. This ensures that you have at least one well element per cluster.

For setting up the fracture mesh, you specify 'fracture element length' and 'fracture element aspect ratio'. The fracture elements are always the same size throughout the simulation. Your selection of 'fracture element length' will have a huge impact on runtime. The simulation rumtime scales with the number of fracture elements. You generally want to be running simulations with 1000s of fracture elements, usually not more than 5000. If you run a simulation and find that it is taking too long, then the surest way to speed up the simulation is to increase fracture element length. We have done sensitivity analysis on fracture element size, and found that the results are quite robust to coarsening the mesh. We usually use values of fracture element length of 50-100 ft. For 'aspect ratio', we recommend either 1 or 2. Aspect ratio of 1 means that the fracture elements are square. Aspect ratio of 2 means that the element length is double the element height. In order to ensure numerical accuracy, you cannot use aspect ratio values greater than 2.

When setting up the simulation mesh and wellbores, it is important to keep in mind tradeoffs between model scale and detail. ResFrac has highly detailed physics, but as a consequence, does not run fast enough to be used on full wellbores. Usually, we run simulations with 1-5 stages in 1-10 wells. In simulations with multiple wells, we cut a cross-section through the formation (in the direction of SHmax) to include one or more stages in each of the wells. Right now, we use a Cartesian mesh, which neglects structural complexity (such as dip, offset at faults, or bed thinning). In the future, we'll support importing of a corner point mesh from Petrel. But for now, the Cartesian mesh is fine because at the scale of a model (100s-1000s ft), there usually is not a huge amount of structural complexity in most shale plays.

Meshing in ResFrac



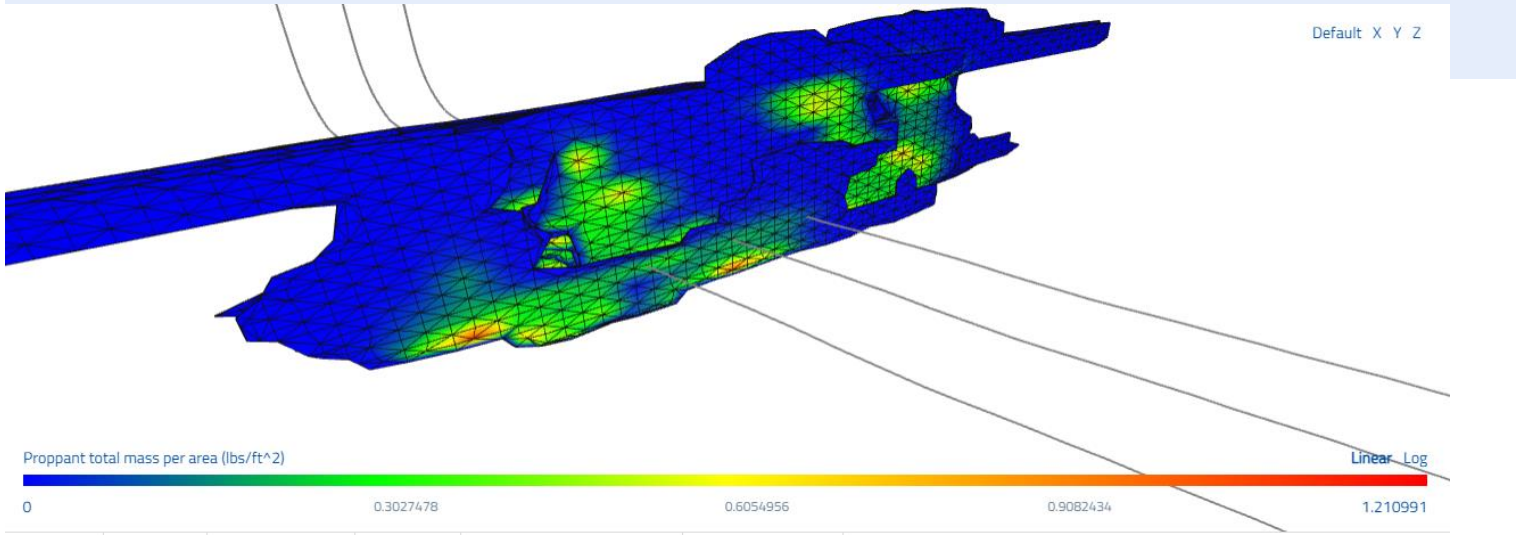
Use the meshing wizard to automatically set up the meshing of the matrix. Alternatively, the matrix meshing can also be set up independently.

The wellbore matrix length is set to the value specified and meshed to the surface. The wellbore element size should not be greater than the cluster spacing so that at least one wellbore element is between each cluster.

The fracture element size will affect the run time significantly. Keeping the number of fracture elements less than 5000 is optimal. The comments file, found in the results folder, outputs the number of open and closed fracture elements.

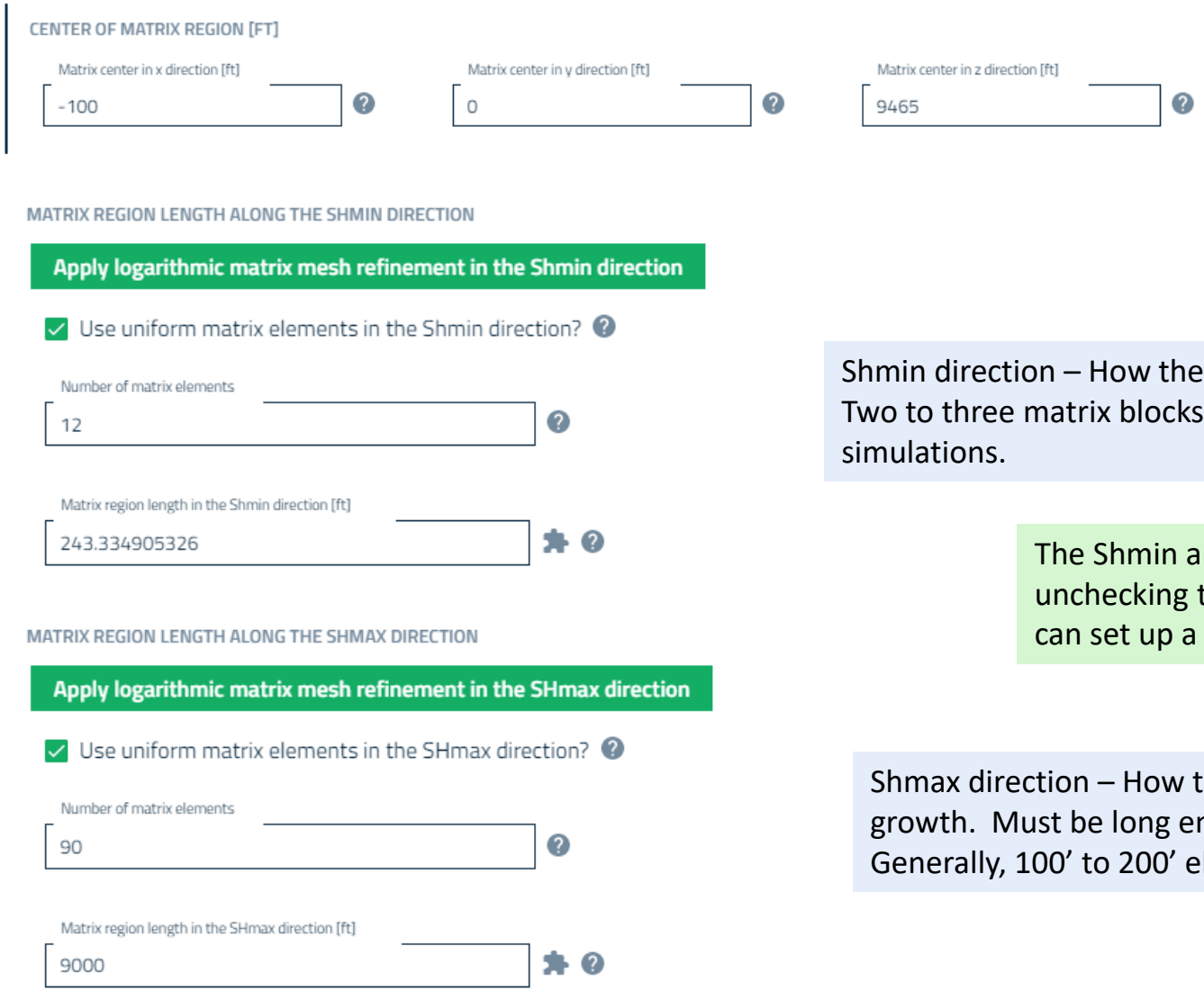
Additionally, the aggregate number of fractures in one direction should be approximately 3. For instance, if you have a contained 150’ zone, then the fracture height should be no more than 50’. Using an aspect ratio of 2 and setting the fracture length to 100’ would also accomplish this goal.

- **There are three meshes used in ResFrac**
  - **Wellbore**
  - **Fracture Elements**
  - **Matrix**



ResFrac

Matrix Meshing in ResFrac, Shmin and Shmax Directions



If the inline method was used for setting up the stages, then this will be set for you.

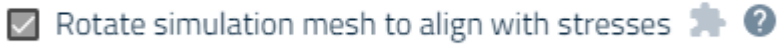
Shmin direction – How the matrix mesh is subdivided along the wellbore. Two to three matrix blocks between each perforation is advisable for most simulations.

The Shmin and Shmax meshing can be subdivided uniformly, or by unchecking the box a custom length can be defined. Additionally, ResFrac can set up a logarithmically distributed mesh.

Shmax direction – How the matrix mesh in the direction of the fracture growth. Must be long enough to contain the length of the fractures. Generally, 100’ to 200’ element sizes are OK here.

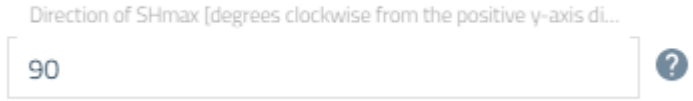
Meshing relative to the x, y and z coordinates

Startup Tab



Automatically orients the meshing to align with Shmax.

Static Model and Initial Conditions Tab



Specifies the orientation of Shmax measured from the positive y axis.

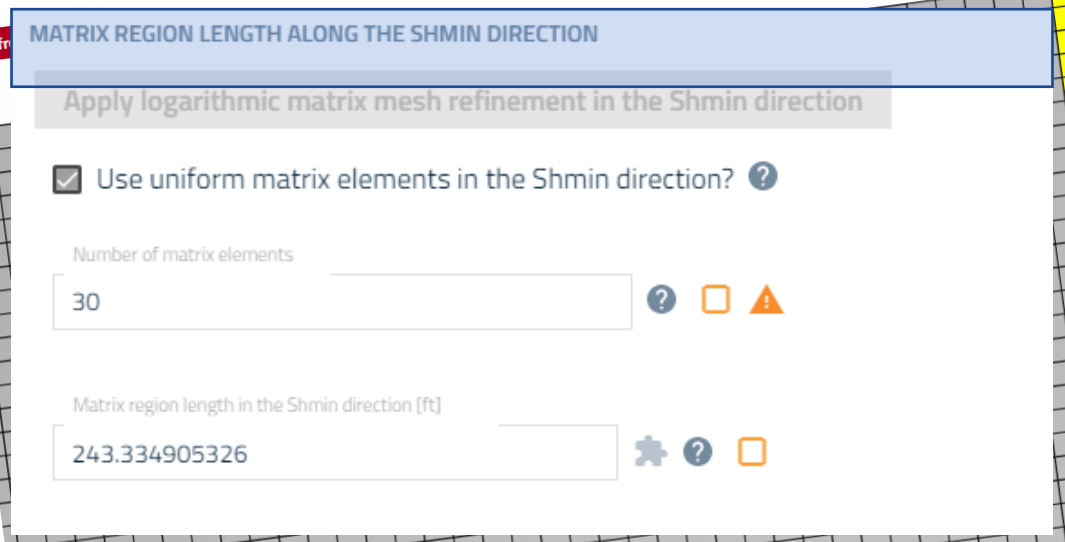
The meshing falls within the coordinate system, but you don't specify the length of the meshing based on the coordinates!

ResFrac

Meshing relative to the x, y and z coordinates

Shmin length – **not** coordinate dependent!

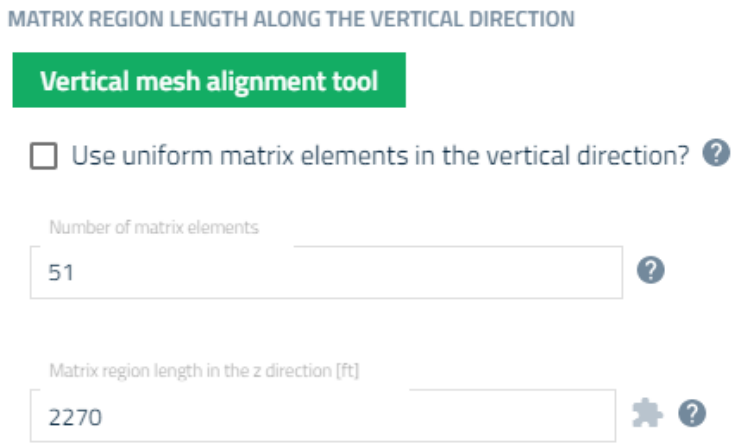
Meshing Tab



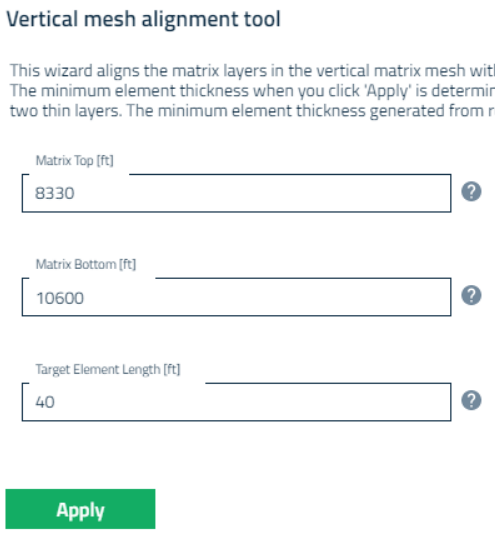
Specify the lengths and the direction in the Shmax direction and ResFrac will rotate the meshing, not the coordinates, and subdivide accordingly.

ResFrac

Matrix Meshing in ResFrac, Vertical Direction



In almost all cases, the vertical mesh alignment tool is used to set up the vertical meshing.



Specify the top and bottom far enough apart to ensure that the fractures are contained without hitting the edges. A fracture will be stopped at the edge of the matrix and cease to propagate in that direction. The energy of the frac will be directed elsewhere and may propagate the fracture laterally further than anticipated.

Specify the desired element length. The vertical meshing tool uses the static model to set up the meshing lengths. However, if the thickness of a layer in the static model is very large then it will attempt to subdivide it into smaller sections based on this value. For instance, Zone 1 in the static model is 10', the meshed length will be 10'. However, if Zone 1 was 120' then this Zone would be subdivided into three separate zones of 40' , all with the same exact properties.

Thank You!

Chris Ponners
cponners@resfrac.com

Last updated Feb 2026