

# MeshIt – Practical User Guide

In this document the **MeshIt** standard workflow is shown for the generation of a three-dimensional tetrahedral mesh of a natural reservoir with example data containing a discrete and a fully represented fault as well as two geological layers and a well. For more technical information about **MeshIt** please refer to Blöcher & Cacace (2015).

## Table of Contents

<b>1.</b>	<b><i>Input Data</i></b> .....	<b>1</b>
<b>2.</b>	<b><i>Loading Data and Visualisation</i></b> .....	<b>2</b>
<b>3.</b>	<b><i>Refinement and Surface Meshing</i></b> .....	<b>3</b>
<b>4.</b>	<b><i>Setting Material Points</i></b> .....	<b>6</b>
<b>5.</b>	<b><i>Selection of Relevant Surface Intersections</i></b> .....	<b>8</b>
<b>6.</b>	<b><i>Advanced Selection via Material Points</i></b> .....	<b>10</b>
<b>7.</b>	<b><i>3D Meshing</i></b> .....	<b>11</b>
<b>8.</b>	<b><i>3D Mesh Export</i></b> .....	<b>12</b>
<b>9.</b>	<b><i>References</i></b> .....	<b>13</b>

## 1. Input Data

To start the mesh generation process input data in the form of scattered data points for all surfaces defining the homogeneous areas of the reservoir (boundaries for geological units, fault zone surfaces, well paths) is necessary. A hard requirement for the surfaces is that in those locations where they touch each other (e.g. two boundary surfaces in a model corner or a fault surface hitting the top of the model, ...) a true overlap is necessary for the correct calculation of the corresponding intersections (see **Selection of Relevant Surface**

**Intersections).** All surfaces need to be prepared outside of **MeshIt** meeting all the requirements. Each surface should then be saved as an individual input file.

Input Data Requirements:

- **every surface in a single file!**
- **.txt .csv ...**
- **x, y, z**
- **no empty lines**
- **no header**
- **decimal separator .**
- **true overlap between input surfaces**

Example (x,y,z) input:

```
300.00;-1500.00;-500.00
300.00;-1465.00;-496.00
300.00;-1430.00;-492.00
300.00;-1395.00;-488.00
300.00;-1360.00;-484.00
300.00;-1325.00;-480.00
300.00;-1290.00;-476.00
300.00;-1255.00;-472.00
300.00;-1220.00;-468.00
300.00;-1185.00;-464.00
300.00;-1150.00;-460.00
300.00;-1115.00;-456.00
300.00;-1080.00;-452.00
300.00;-1045.00;-448.00
300.00;-1010.00;-444.00
```

...

#### **Example Data:**

To get familiar with the mesh generation process example input data is provided:

Border: east\_border.csv, north\_border.csv, south\_border.csv, west\_border.csv

Faults: main\_fault\_1.csv, main\_fault\_2.csv, secondary\_discrete\_fault.csv

Units: top.csv, middle.csv, bottom.csv

Well: production\_well.csv

## **2. Loading Data and Visualisation**

Input files can be loaded into **MeshIt** depending on the surface type they represent using the Menu: **File → Add → unit/fault/border/well**

*Tipp: multiple input files of the same type can be selected simultaneously holding down "shift"*

To check if the data was loaded correctly a visualization of the scattered data points is recommended. For that use the **View Panel** → choose the data types (e.g. **Units**) and the particular surface (e.g. **all** or individually) → check **Scattered Data Points** (see Figure 1)

*Tipp: In case the loaded data is not viewed correctly (e.g. only one small data point is visible or none at all) check input data again for correct formatting.*

*Tipp: **Surface** and **Intersections** visualization becomes available after **PreMesh** has been executed.*

General visualization options:

- it is possible to change the rotation center which can help in certain situations with a high zoom degree (model, viewport, object)
- there is also a slider to adjust the rotation speed

#### Summary:

- Add all individual input files via **File** → **Add** → **unit/fault/border/well**
- Use the **View Panel** to check if the data was loaded correctly (see Figure 1)

#### Example Data:

After loading and visualization of the 11 example input files as scattered data points the 3D view should look like as shown in Figure 1.

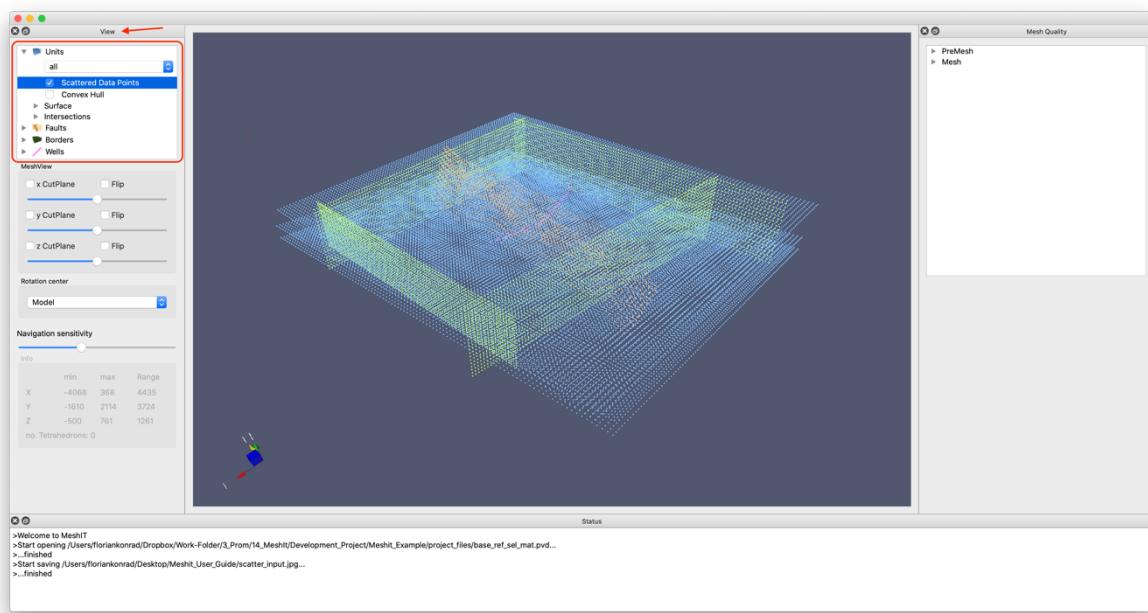


Figure 1 Visualization of scattered input data

### 3. Refinement and Surface Meshing

To construct the surface information from the input scattered data points a triangular meshing for each surface has to be done. Based on this information the interactions between all input-surfaces (intersections) can be calculated. This step is called **PreMesh** and the user can find all necessary options inside the **Mesh Quality Panel**. Here the refinement, meaning the edge size of the triangles, for each input surface has to be specified. The user has to decide on reasonable values (depending on the size the models geological units and elements) here

since the triangle-size on each surface forms the basis for the 3D tetrahedral element generation and therefore sets the spatial discretization in the resulting 3D mesh. Inside the **Premesh** submenu an interpolation type can be set for the interpolation between input data points. It is recommended to choose **kriging** here. After clicking >**Execute PreMesh**< surface information becomes available inside the **View Panel**.

#### Summary:

- choose refinement values for each surface (see Figure 2)
- set the interpolation type (see Figure 2)
- >**Execute PreMesh**<
- Visualize surfaces via **View Panel** → **Surface** → check **Faces** and **Edges** (see Figure 4)

#### Example Data:

Reasonable refinement values for the example data is shown in the following table.

Surface	Refinement
<b>bottom</b>	50
<b>top</b>	50
<b>middle</b>	50
<b>main_fault_1</b>	15
<b>main_fault_2</b>	15
<b>secondary_discrete_fault</b>	20
<b>north_border</b>	40
<b>east_border</b>	40
<b>south_border</b>	40
<b>west_border</b>	40
<b>production_well</b>	6

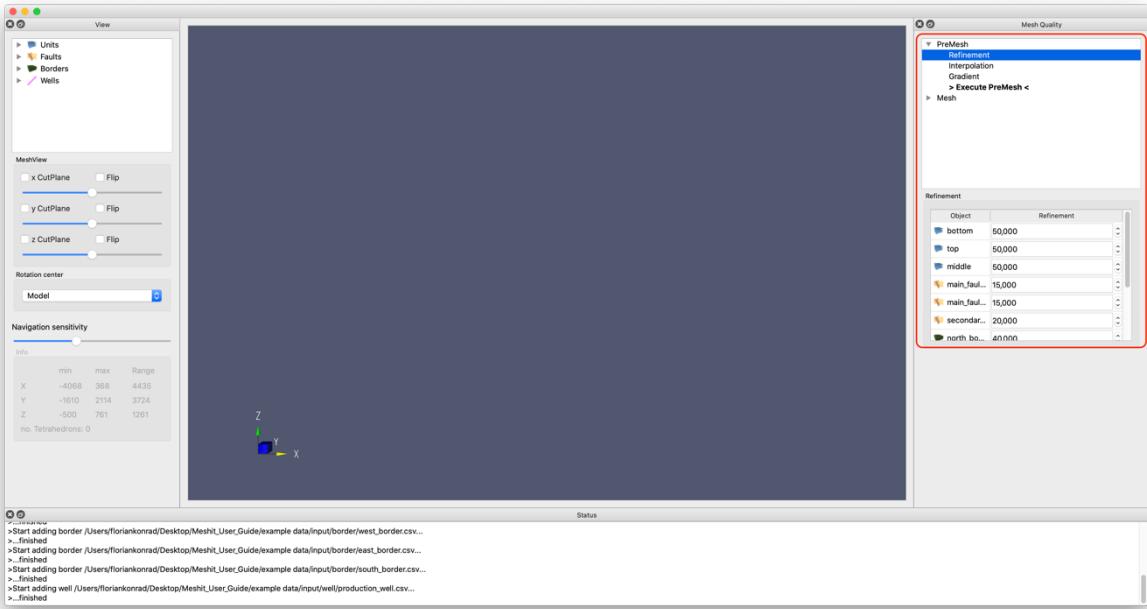


Figure 2 Refinement menu for the PreMesh stage

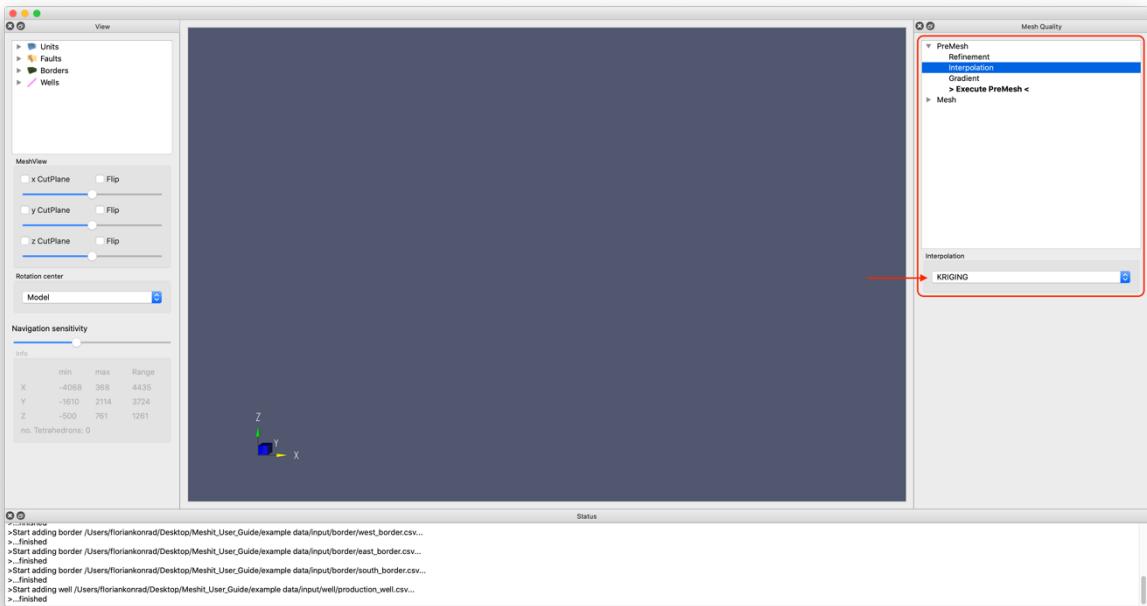


Figure 3 Interpolation type menu for the PreMesh stage

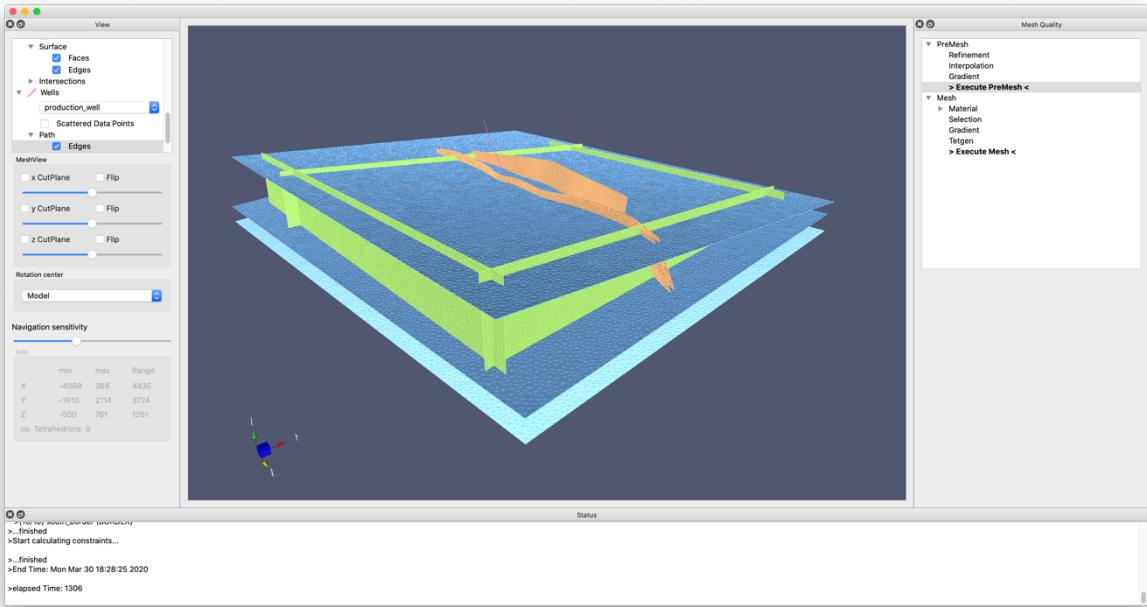


Figure 4 Visualization of surface meshes after PreMesh execution

## 4. Setting Material Points

For all three-dimensional subdomains that are defined by the surrounding input surfaces the user has to specify how they will be parametrized in the final modelling software (see an example of a subdomain in Figure 6). Meaning, should each subdomain get an individual object ID or should they be grouped into higher order IDs through which the physical properties will get applied to all tetrahedral elements inside that subdomain/group of subdomains in the final numerical model. To define this the user needs to set material IDs for discrete elements (wells and discrete faults) as well as for all 3D subdomains using marking points via the **Mesh Quality Panel → Mesh → Material**.

- 1D and 2D materials can be set by checking the boxes corresponding to the desired wells or 2D faults
- 3D materials can be added via **+/- buttons** below the **add/remove Material** and the **add/remove Location Panel**. One 3D Material can have multiple locations. (material points will assign the same material ID to all tetrahedral elements inside its current subdomain)
- **Every subdomain needs to get an Material ID assigned!**

*Tipp: Adjust rotation center and sensitivity to precisely set material points inside narrow or small subdomains and zoom in very far to visually confirm that the material points indeed are located inside the desired area.*

### Summary:

- Set wells as 1D materials by checking corresponding box in **Quality Panel → Mesh → Material → 1D**
- Set 2D discrete surfaces like faults by checking corresponding box in **Quality Panel → Mesh → Material → 2D**

- Set 3D materials by moving location points into each subdomain of the model area:  
**Quality Panel → Mesh → Material → 3D → Add Material → Add Location → Move point with x,y,z slider or input exact coordinates** (see Figure 5 and Figure 7)

### Example Data:

- Set 1D material for production\_well
- Set 2D material for secondary\_discrete\_fault
- Set six location points totally for covering all subdomains resulting from the input data (3 Materials with 2 locations each → upper geological layer will be ID 0, lower layer ID 1 and fully represented fault zone gets ID 2)

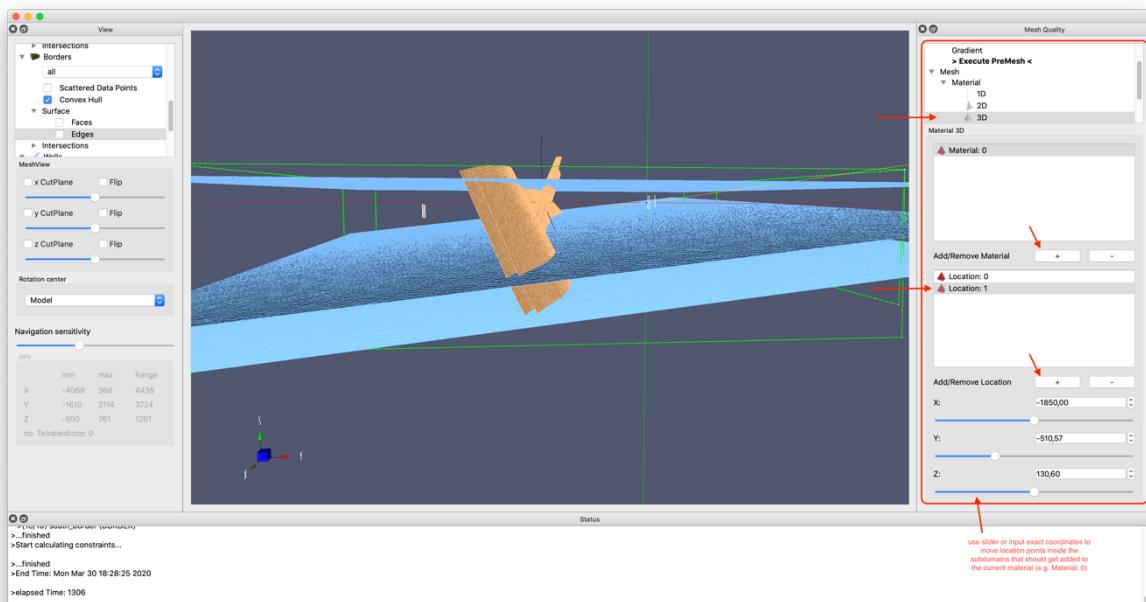


Figure 5 3D material menu

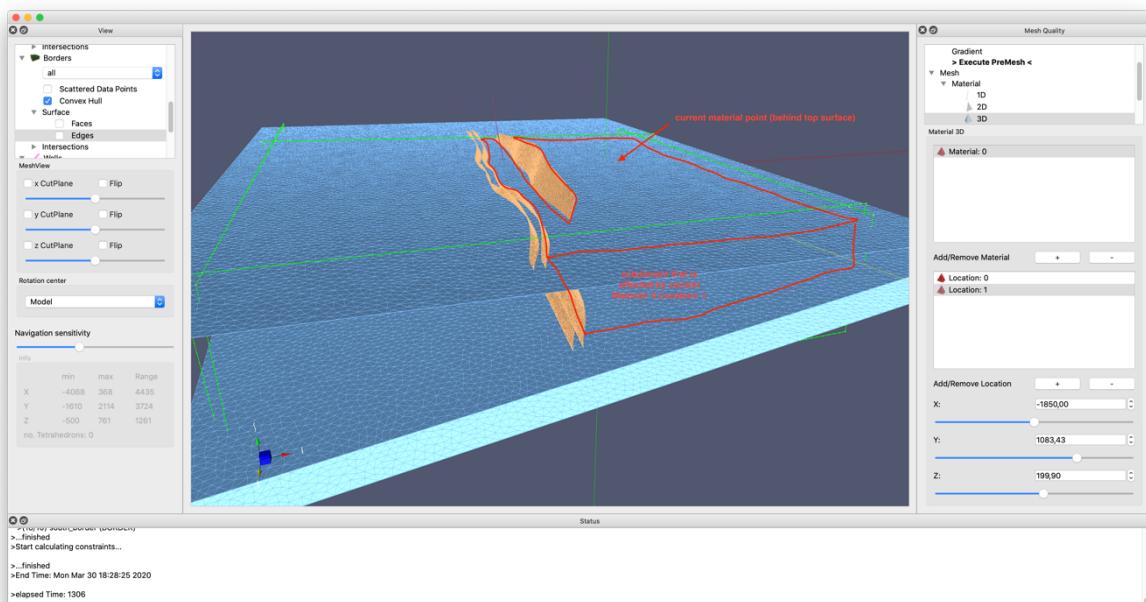


Figure 6 Example for a subdomain which is affected by a location point

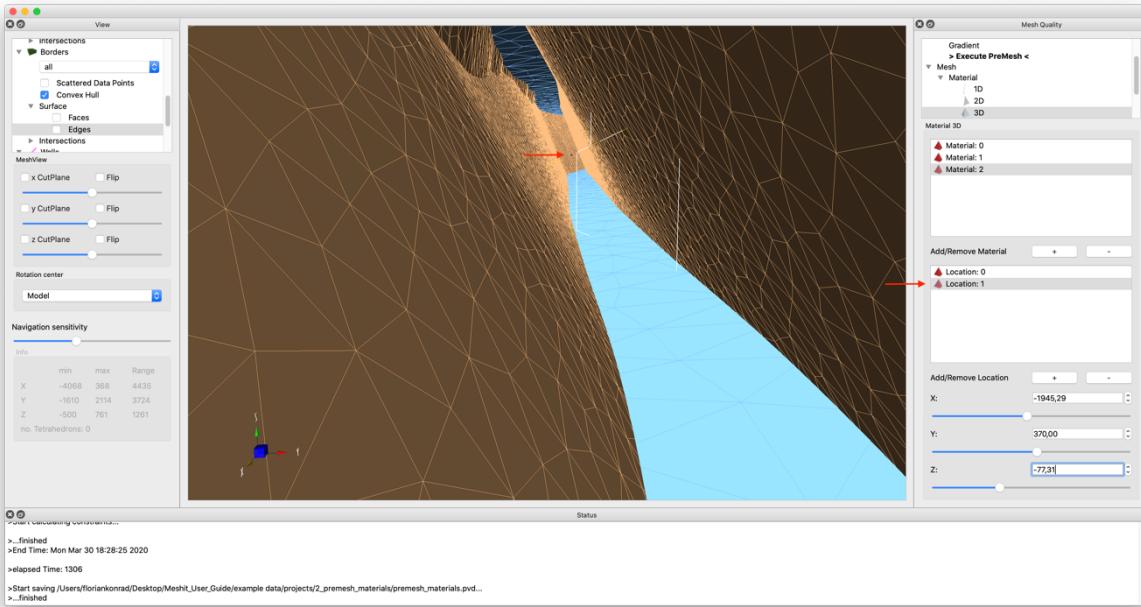


Figure 7 Example for small subdomain inside fault zone and its material location point

## 5. Selection of Relevant Surface Intersections

In a next step the user has to “cut out” the parts of interest for the final model from the overlapping input data. This is done by manually selecting the relevant intersections between all the input surfaces via the **Quality Panel → Mesh → Selection**. The user has to work his/her way through all surfaces that can be selected via the dropdown menu. There are different tools available (**single**, **multi**, **all**, **bucket**, **polygon**) to select/deselect (**mark/unmark**) intersection lines or to mark them for exclusion during meshing (**hole**).

*Tipp: It is important to spatially understand how the overlapping input surfaces interact with each other. For that it can be helpful to visualize different surfaces via the **View Panel**. Or use the **highlight all unmarked** option from the dropdown menu (see Figure 10).*

### Summary:

- Go over each surface in the **Selection Panel** using the dropdown menu
- Select the relevant intersections that define the area which will be meshed (definition of a piecewise linear complex, see Blöcher & Cacace (2015)) using the provided selection tools (see Figure 8)

### Example Data:

The surface **secondary\_discrete\_fault** is supposed to end at the front of the main fault (**main\_fault\_1**) in the final mesh. This is a design property specific to this example data to show that **secondary\_discrete\_fault** surface is intersecting additionally with the surfaces **main\_fault\_2** and **middle** (see Figure 9). This results in a number of intersection lines that can be confusing for a first time user. To select only the correct lines without skipping any necessary ones the user needs to spatially understand the model (using surfaces visualization).

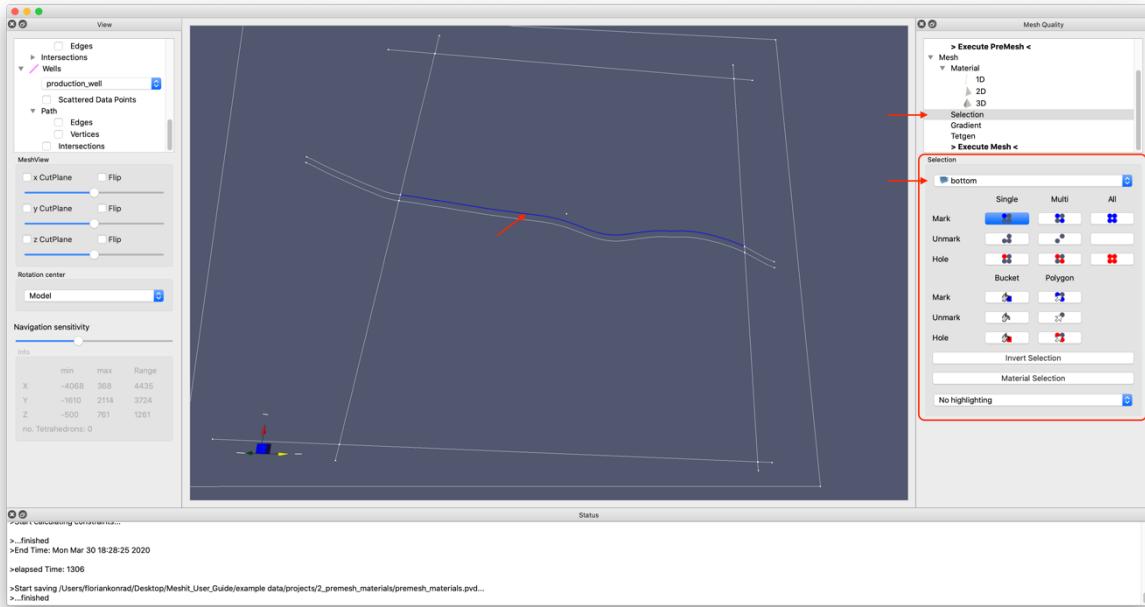


Figure 8 Selection menu and example for one selected (blue) intersection

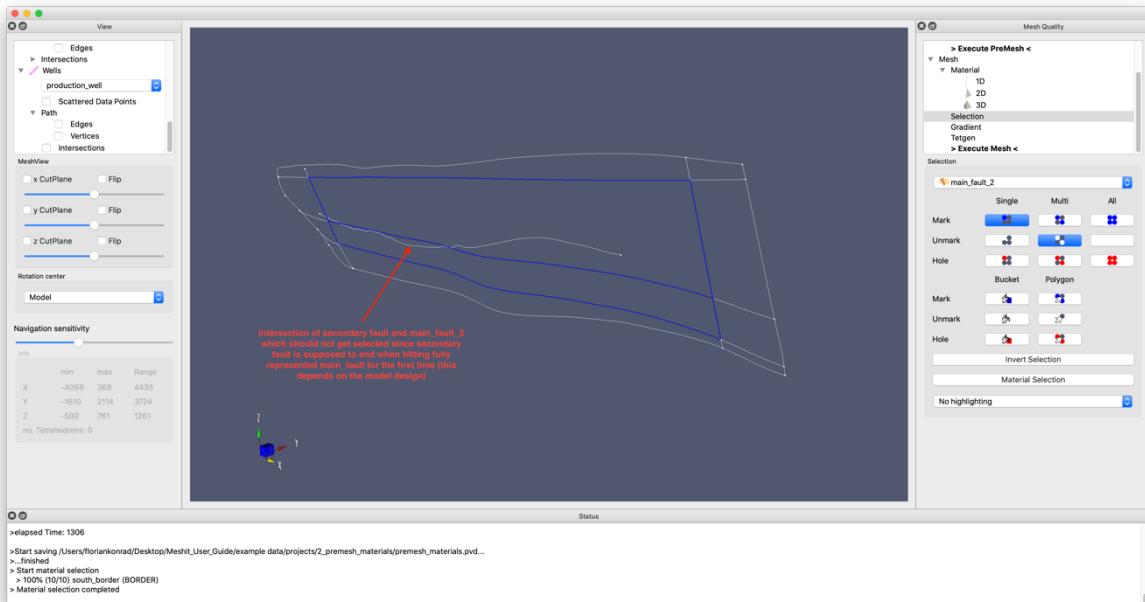


Figure 9 Example for complex selection of intersections

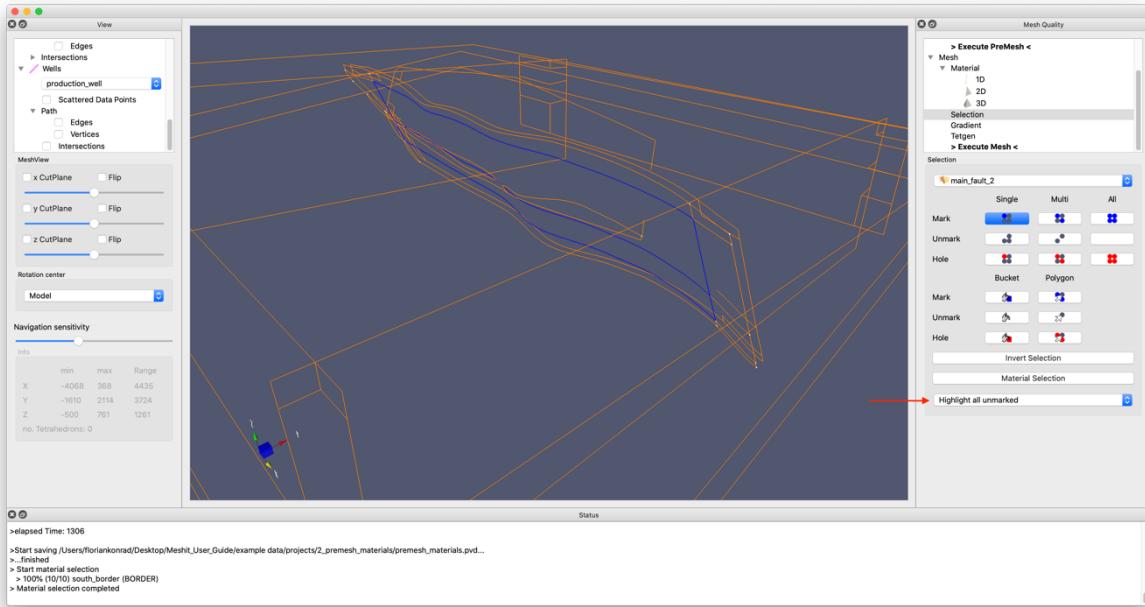


Figure 10 Highlight all unmarked option

## 6. Advanced Selection via Material Points

If the materials points of the previous workflow step have been set correctly they can be used to select intersections automatically. Use the **Material Selection** button for this. This automatic selection cannot account for model specific characteristics like one fault ending at another one. Therefore, in complex geometries the automatic selection will most likely select more intersections than desired. These selections need in a second step be manually deselected again using the previously presented tools.

### Summary:

- Use the **Material Selection** tool to select intersections via previously set material points
- Check and manually adjust the automatically set selections for each surface

### Example Data:

If choosing this workflow instead of selecting everything manually the selections have to be at least adjusted for **secondary\_discrete\_fault** surface. Use the **Unmark** tools to deselect unnecessary selected intersections (see Figure 11).

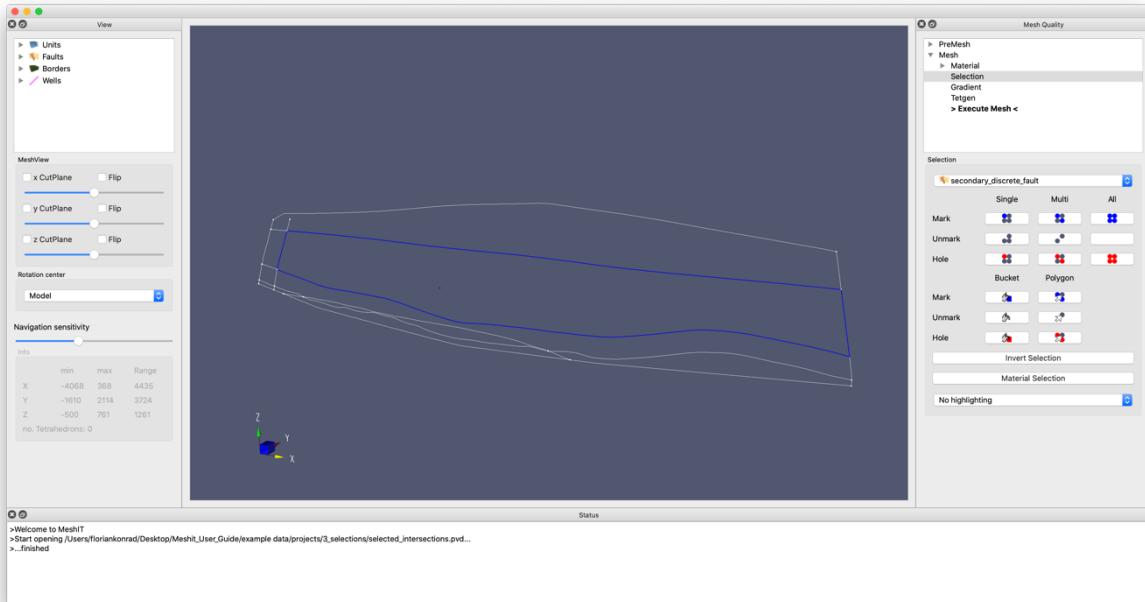


Figure 11 Selection for example data surface ***secondary\_discrete\_fault***

## 7. 3D Meshing

The last options that are available before generating the final 3D tetrahedral mesh are **Gradient** (**Mesh Quality Panel** → **Mesh** → **Gradient**) and **Tetgen** (**Mesh Quality Panel** → **Mesh** → **Tetgen**).

**Gradient** allows the user to define a gradient between elements of different sizes.

The **Tetgen** option opens MeshIt up for the command line switches that are available for Tetgen for the final meshing process (Si, 2015).

After clicking **> Execute Mesh <** the 3D mesh can be visualized via the **View Panel** where an additional **Material** menu becomes available. **MeshIt** is not meant to visualize complex meshes with many elements. This view option is only a rudimentary option to quickly confirm the mesh generation. For a proper check of the resulting mesh an export and a visualization in Paraview is recommended (see **3D Mesh Export** for different export options).

*Tipp: The number of resulting elements of the 3D mesh is shown inside the info box below spatial model information (see Figure 12).*

### Summary:

- Change the gradient between different sized elements via **Mesh Quality Panel** → **Mesh** → **Gradient**
- Optionally adjust the Tetgen command line switches via **Mesh Quality Panel** → **Mesh** → **Tetgen**
- **> Execute Mesh <** to generate the final tetrahedral mesh
- Quickly visualize the 3D Mesh via **View Panel** → **Materials** → **all** → check **Surfaces** and **Edges** (see Figure 12)

### Example Data:

- **Gradient** and **Tetgen** options don't need to be adjusted for the example
- Click **> Execute Mesh <**

- Quickly visualize the 3D Mesh via **View Panel** → **Materials** → all → check **Surfaces** and **Edges**

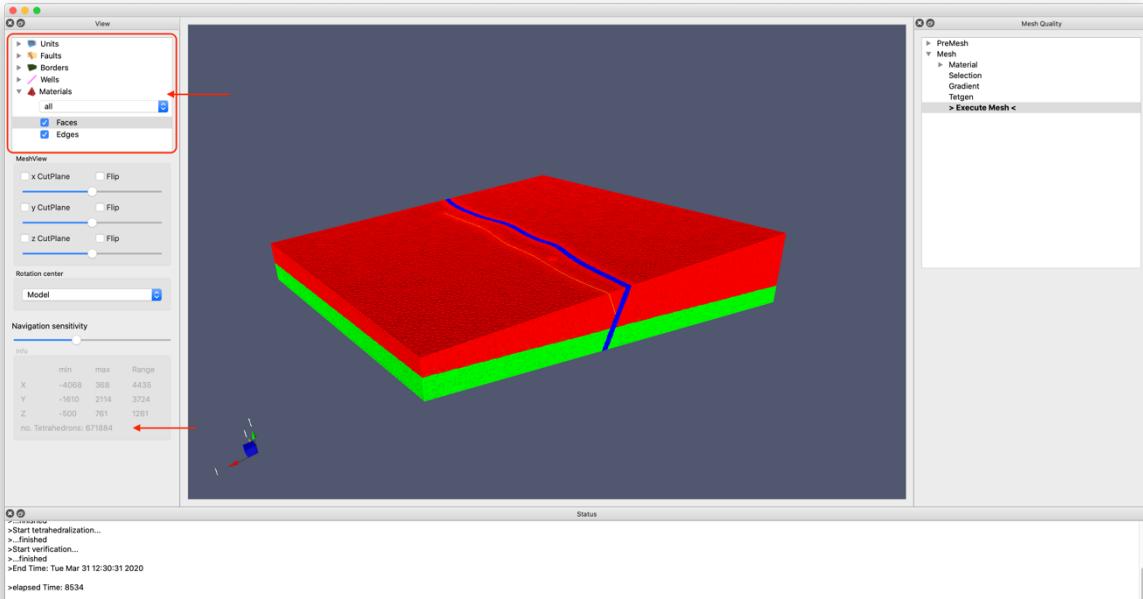


Figure 12 Quick visualization of a generated 3D mesh

## 8. 3D Mesh Export

To export the final 3D mesh use **File** → **Export as ...** → **3D mesh** → choose a format. Different options are available for different simulators (FeFlow, OpenGeoSys, Comsol, Abaqus, Moose) and additionally for Paraview. Surface meshes can also be exported via **File** → **Export as ...** → **2D mesh** (see Figure 13).

Moose export is only available if Moose is installed on the system and **MeshIt** was build enabling the exodus option.

To export the 3D mesh for Moose or Abaqus the user is prompted with an additional window before specifying a storage location. In this window the model boundaries need to be selected (additionally the user can rotate the generated mesh in the coordinate space here by a certain degree counterclockwise).

### Summary:

- **File** → **Export as ...** → **3D mesh** → choose a format
- For Moose/Abaqus: Select boundary surfaces in the additional prompt
- Save!

### Example Data:

- **File** → **Export as ...** → **3D mesh** → **Moose**
- Select north\_border, west\_border, south\_border, east\_border → OK
- Specify a location and save!

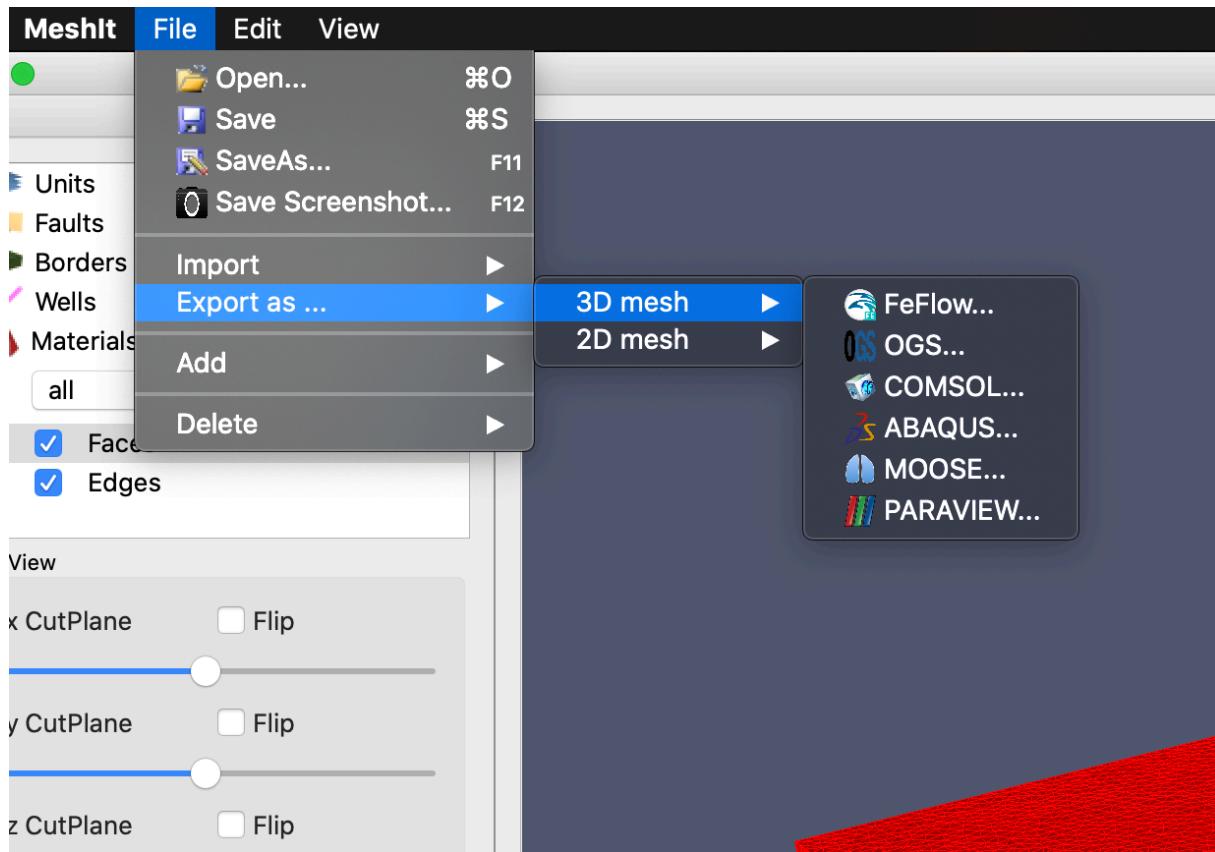


Figure 13 Export options

## 9. References

Blöcher, G., & Cacace, M. (2015). MeshIt—a software for three dimensional volumetric meshing of complex faulted reservoirs. *Computers and Geosciences*, 74(6), 5191–5209. <https://doi.org/10.1007/s12665-015-4537-x>

Si, H. (2015). *TetGen Command Line Switches*. <https://wias-berlin.de/software/tetgen/switches.html>