Rapid Reservoir Modelling

This help and tutorial document consists of five sections.

A. Short overview of workflow

B. Recommendations for using RRM

C. Detailed workflow from sketching to flow diagnostics

D. Key and Mouse commands

E. Screenshot tutorial

Sections A and C are a general and detailed overview of the workflow and the options available in RRM. Section B lists a number of recommendations for using RRM that are related to ongoing developments. Section D lists all mouse and keyboard shortcuts. It is recommended to have a quick look through section A and B. Section E consists of a tutorial using screenshots of the different steps involved and the highlights the associated buttons. Some tutorial videos are also available on the RRM website.

A. Short overview of workflow

In general the workflows starts by sketching a model in the sketch window and then transfer it into the flow diagnostics module and visualise results (Figure 1).

1) Make a sketch in the sketch window;

2) Insert sketch into Flow Diagnostics module;

3) Drag-and-drop region IDs to all volumes created in the sketch;

4) Assign petrophysical properties to every region ID;

5) Generate a cornerpoint or unstructured mesh;

6) Compute flow diagnostics;

7) Visualize results;

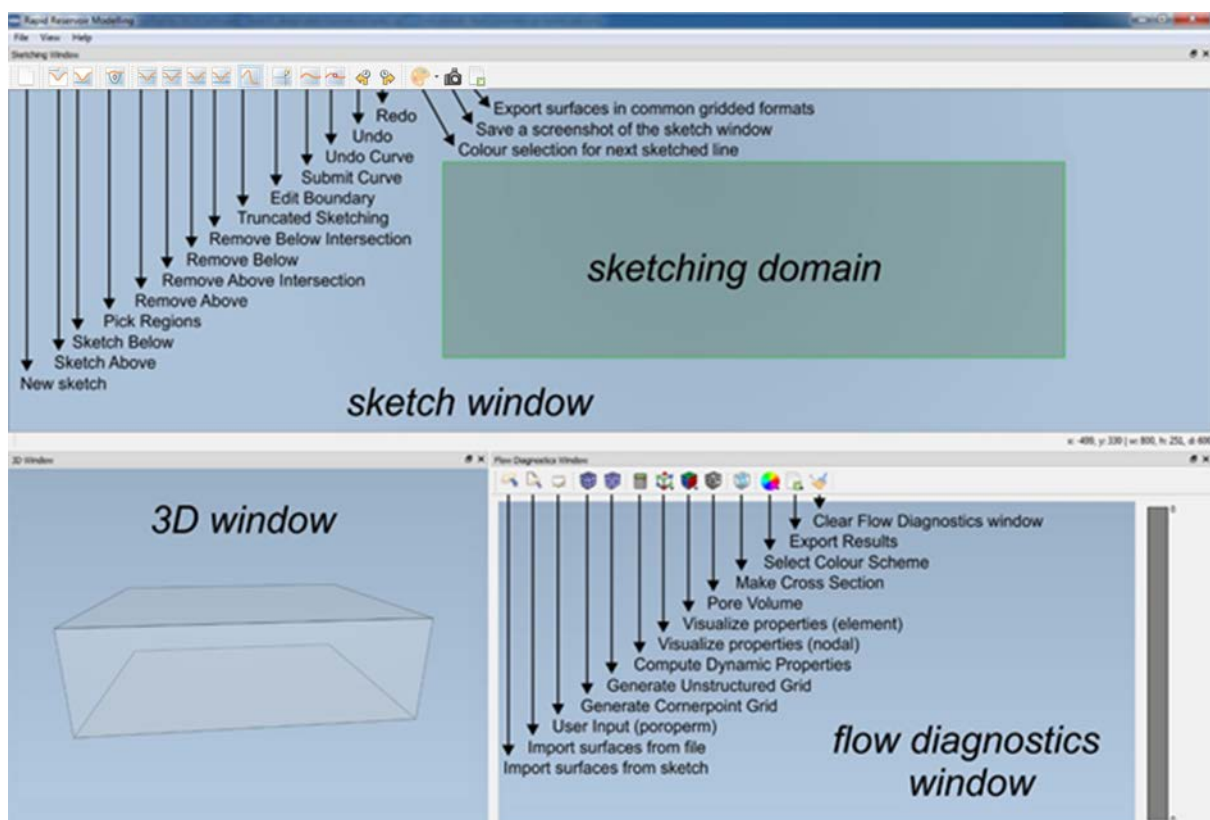8) Optional: Export sketched surfaces and/or flow diagnostics results;

Figure 1. Overview of the Rapid Reservoir Modelling software.

B. Recommendations for using RRM

Because RRM is still under development, some available features are not fully implemented yet. The list below consists of recommendations to avoid running into problems. This list will change for every release as some features will be completed while others are being developed.

B.1. Sketching related recommendations

- When sketching a new surface it is best practice to sketch across the whole sketch domain. This is to avoid unwanted extrapolations generated by RRM in the sketch window and helps stratigraphic rules to make correct truncations.

- Overhanging surfaces are not implemented in the current version or RRM.

- It is advised to finish every model by sketching a continuous lower and upper boundary for the model. This will ensure correct placement of wells for flow diagnostics.

B.2. Flow Diagnostics related recommendations

- Corner-point grids are suitable for very simple geometries, whereas unstructured grids are advised for more complex geometries and intersections. Some surface interactions will cause problems for corner-point grids and produce incorrect results. Besides, for corner-point grids, the region IDs should always be given from bottom to top.

- The discretisation scheme is control volume finite element method for unstructured grids and finite volume method for corner-point grids. Thus the results obtained using unstructured grids should be more accurate.

- In this release, single-phase incompressible flow is assumed for flow diagnostics. Pressure, velocity, TOF and tracer results are all of steady-state.

C. Detailed workflow from sketching to flow diagnostics

Below is a detailed chronological step-by-step description of the workflow, covering all aspects and buttons integrated in RRM.

C.1. Sketching a model in the sketching window

C.1.1 Select the sketching boundary

After opening RRM, a sketching domain is automatically created ready to start. Two options are available to change the default boundary.

Select button to draw a new boundary. Click-and-drag your cursor across the blue sketch window to create a new rectangular boundary.

For sketching on top of an image, drag an image file (*.png, *.jpg) into the sketch window. A new boundary will be created around the image.

A completely new sketch can be started by selecting button or using the 'n'-key. A new sketching domain will be created and all surfaces are deleted.

The left and right boundaries of the sketching domain correspond to the simulation model's boundaries. The bottom and top of the simulation model exists of the lowermost and uppermost sketched surfaces.

C.1.2 Sketching

Sketching starts by clicking-and-dragging the cursor across the sketching domain. It ends by releasing the cursor and a 'right-click' or 'i'-key. The 'right-click' or 'i'-key confirms the new sketched line and generates the corresponding surface in 3D. It will then automatically be displayed in the 3D window.

A sketched line can be cancelled by clicking button before confirming. The new line will be removed and a new line can be sketched.

Before sketching a line, an appropriate geological rule can be selected, depending on the next surface to be drawn. Five rules are available in RRM:

- 'Sketch' (default)

This rule is used to simply draw a line between two boundaries. The new sketched line will be truncated by previously existing lines, or the sketching domain. This is the default sketching rule. The new line will start from the first line/boundary it intersects, and end at the last line/boundary it intersects.

- 'Remove Above'

This rule allows to sketch a line across the sketching domain and will remove all previously sketched surfaces that lie above the new line.

- 'Remove Above Intersection'

This rule allows to sketch a line across the sketching domain and will cut all surfaces that it intersects. It then removes the portions that lie above the new line.

- 'Remove Below'

This rule allows to sketch a line across the sketching domain and will remove all previously sketched surfaces that lie below the new line.

- 'Remove Below Intersection'

This rule allows to sketch a line across the sketching domain and will cut all surfaces that it intersects. It then removes the portions that lie below the new line.

Two additional stratigraphic rules are available that allow to specify smaller sketching domains by selecting previously sketched lines as upper and lower boundaries.

- 'Sketch Above'

Selecting 'Sketch Above' allows to select a lower boundary for the next lines to be sketched. All available lines will be highlighted in yellow. Use the cursor to select one of these yellow lines. The selected line will colour red to indicate it acts a new lower boundary. It remains active until 'Sketch Above' is clicked again to deactivate it.

- 'Sketch Below'

Selecting 'Sketch Below' allows to select an upper boundary for the next lines to be sketched. All available lines will be highlighted in yellow. Use the cursor to select one of these yellow lines. The selected line will colour red to indicate it acts a new upper boundary. It remains active until 'Sketch Below' is clicked again to deactivate it.

Both 'Sketch Above' and 'Sketch Below' can be used in conjunction to sketch in specific stratigraphic domains. Either rules will also affect the lines available for its complement. For example when 'Sketch Above' is active, only lines above the selected lower boundary will be available for 'Sketch Below'.

'Undo' and 'Redo' is available for surfaces that have been confirmed and are already constructed in 3D.

Line colours are assigned randomly by default. Colours can be set. It can be deactivated by clicking the button again.

C.1.3 Screenshot

A screenshot of the sketching domain can be saved as an image file by clicking the camera button.

C.1.4 Surface export

The sketched surfaces can be exported as gridded surfaces. The surfaces can be loaded into conventional modelling packages.

C.2. Calculating flow diagnostics

Calculation of flow diagnostics consists of three steps, (1) Import the sketch from sketch window, (2) Assign IDs to different regions and set petrophysical properties and (3) Compute the flow diagnostics.

C.2.1 Import sketched model

The sketched model is then imported into the flow diagnostics model. This button will load the surfaces into the flow diagnostics window.

Models can also be imported from files.

C.2.2 Assign region IDs and petrophysical properties

After the sketch is imported into flow diagnostics, the input panel can be opened to assign the number of geological regions and set their attributes.

1. Open the input panel for region attributes

2. Set the number of regions in the input panel

3. In the sketching window, activate 'Region Picking' or use the 'r'-key. A series of numbered diamonds will appear in the upper left corner of the sketching domain (see below). The number represents the region-ID. Drag-and-drop one diamond into each region of the sketch. Every region should have 1 diamond/region-ID.

4. Go through the list of regions and assign the corresponding porosity and permeability using the sliders.
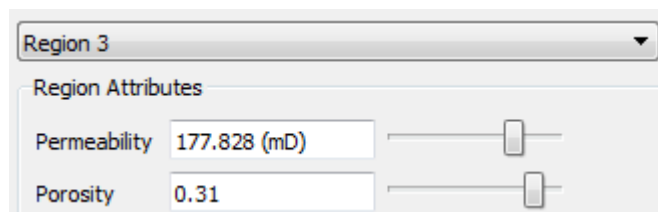
C.2.3 Generating a mesh

Two types of meshes are available in RRM, cornerpoint grids and unstructured grids. Once every region is assigned petrophysical properties, the model is ready to be gridded. The grid will appear in the flow diagnostics window.

 Select mesh button to generate a cornerpoint grid. Cornerpoint grids should only be used for simple geometries (see 'Recommendations').

 Select mesh button to generate an unstructured grid. Unstructured grids are suitable for more complex model geometries.

C.2.4 Computation of flow diagnostics

 Once the model is gridded, flow diagnostics can be calculated by clicking the calculator button.
For now, calculation is based on two vertical wells that are automatically assigned to the model. An injector in the southwest corner of the model and a producer in the northeast corner. Their respective bottom hole pressure is 400 and 300 bar. Because the model has only 2 wells, values for tracers are 0.

C.3. Visualisation and exporting results

C.3.1 Visualise results

Results of flow diagnostics calculation can be visualised on the 3D model.

 The nodal properties button shows the following properties:

- pressure,
- forward time-of-flight,
- backward time-of-flight,
- total time-of-flight,
- max. backward tracer
- max. forward tracer

 The region properties button shows the following properties:

- permeability
- velocity magnitude
- velocity in X direction
- velocity in Y direction
- velocity in Z direction

For now, region properties are visualised as interpolated values between nodes. This will change to a more correct element-based visualisation in next releases.

Show the pore volume for each region in the model.

Different colour schemes are available to visualize the results.

C.3.2 Cross section view

A cross section through the model can be generated. An input panel will show up to enter X, Y and Z components of the normal direction to the cross section. The cross section can be turned off by clicking the same button.

C.3.3 Exporting meshes and results

Different export options are available for the meshes and flow diagnostics results. Select the export-format in the drop-down menu, and choose a filename.

C.4. Clear flow diagnostics module

To clear the flow diagnostics module. This will remove the model from the flow diagnostics to allow a new model to be imported after for example some changes are made to the sketched model.

D. Key and Mouse commands

D.1. General commands

Key/Mouse	Action
n	New sketch
r	Activate/Deactivate 'Region Picking'
E	Export gridded surfaces
P	Save screenshot of sketch domain

D.2. Sketch window

| Key/Mouse	Action
i	Confirm and truncate sketched line
left-mouse button	Sketching line
right-mouse button	Confirm and truncate sketched line
mouse-wheel	Zoom in and out

D.3. Flow diagnostics window

Key/Mouse	Action
left-mouse button	Rotate model
right-mouse button	Visualization option
mouse-wheel	Zoom in and out

E. Screenshot tutorial

The following pictures show an overview of the RRM workflow. Different actions are number chronologically and the buttons involved in the different steps are highlighted.
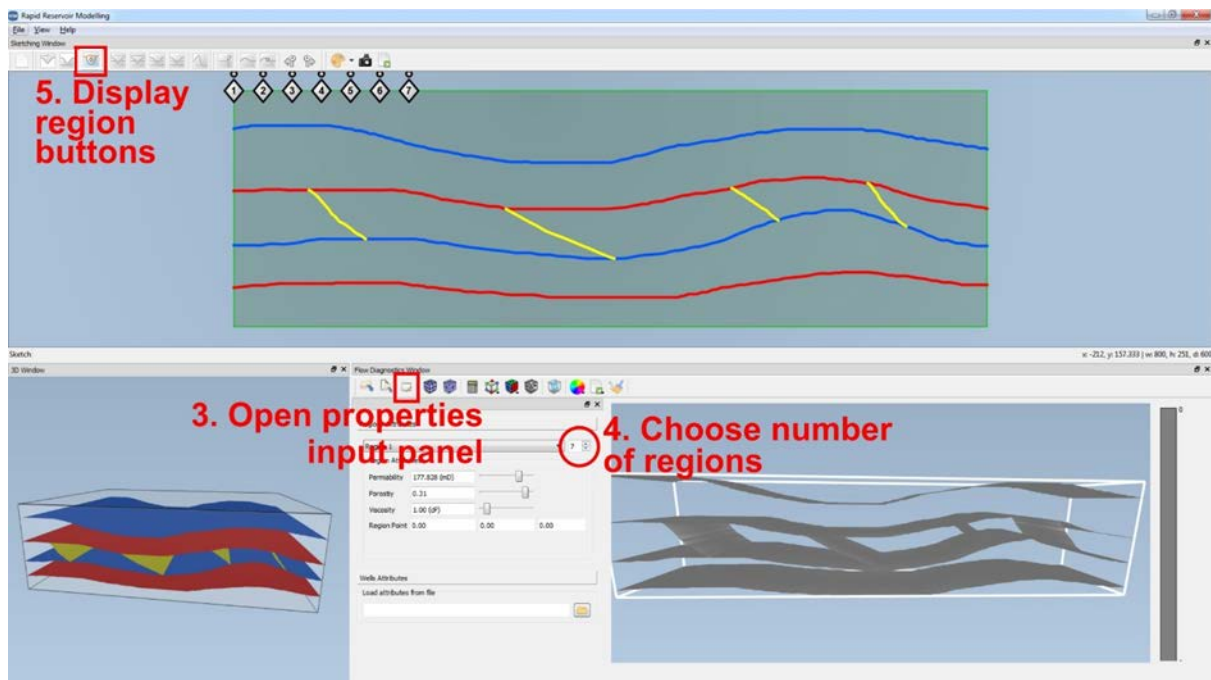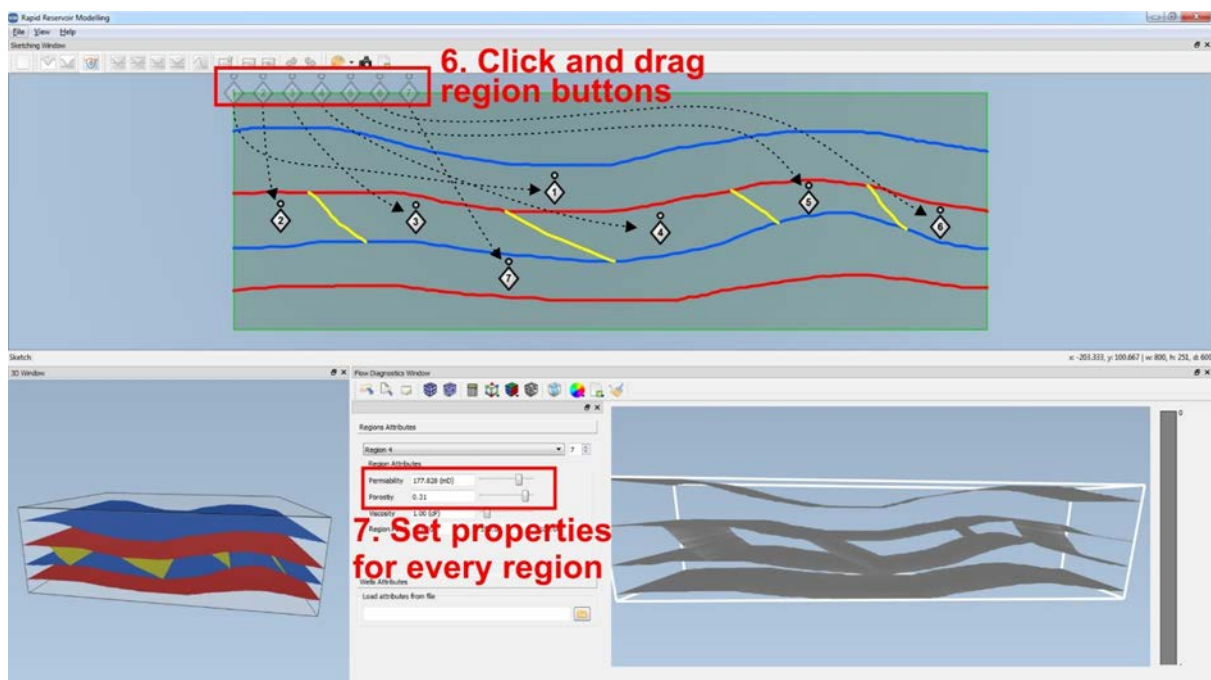
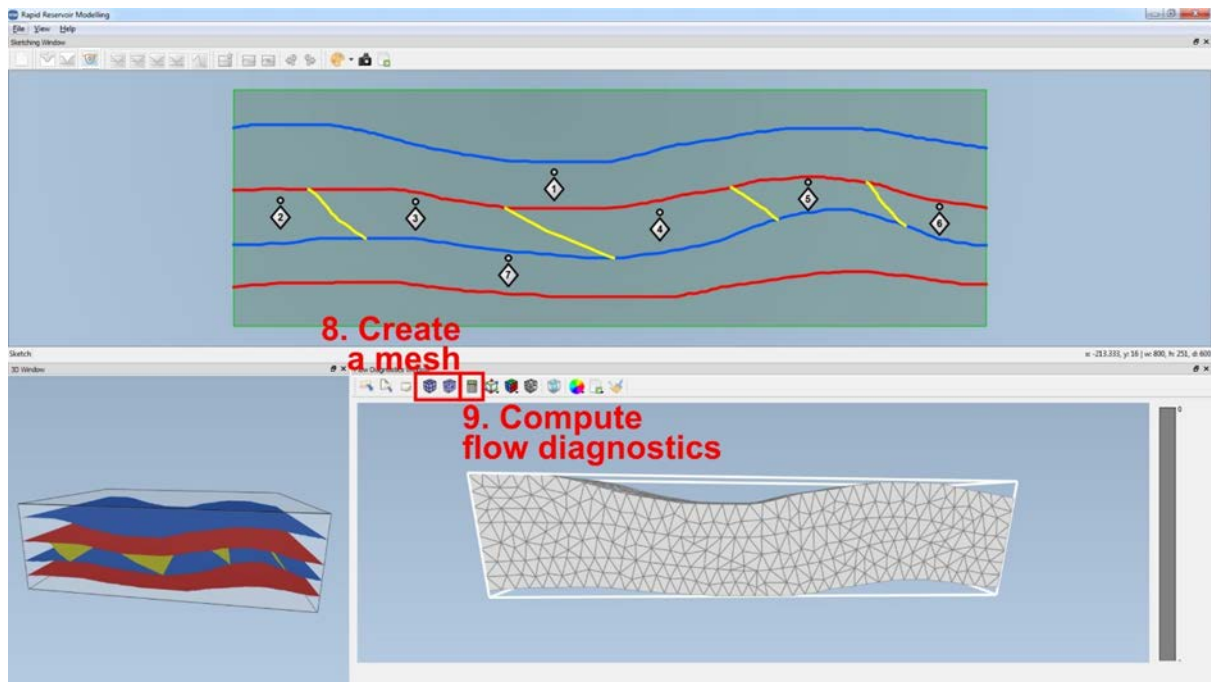## E.1. Sketching
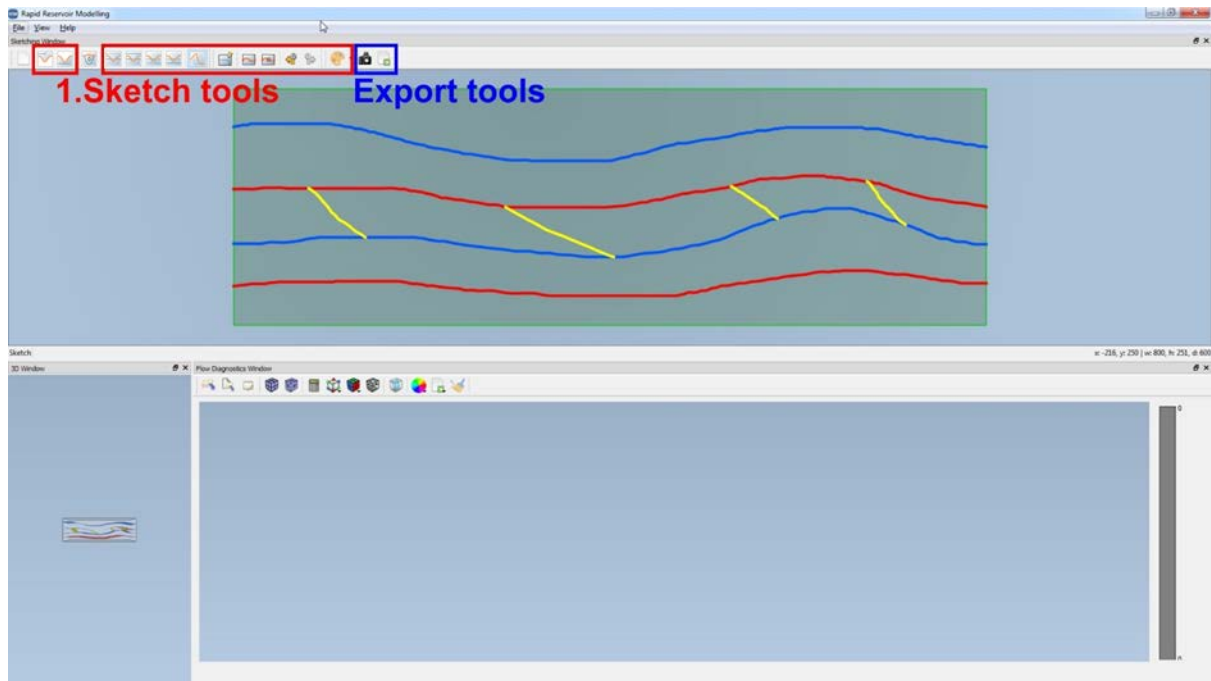
E.2. Load into flow diagnostics
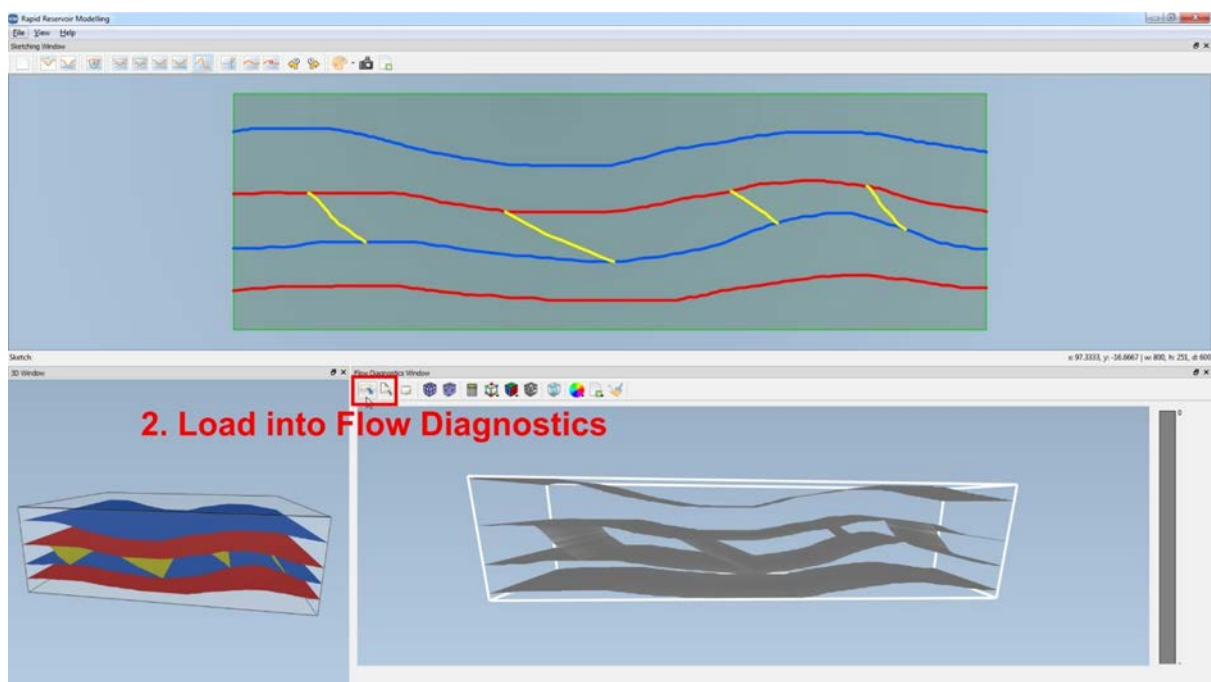
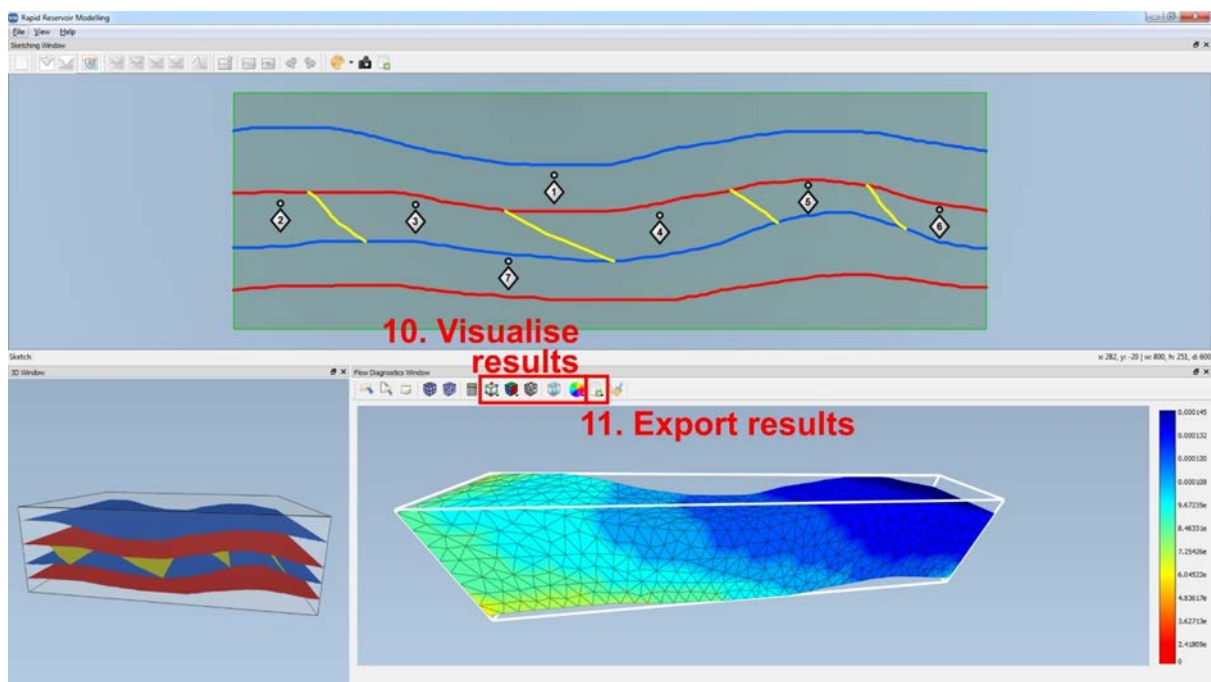## E.3. Open properties panel and select number of regions

E.4. Assign region IDs and set petrophysical properties

E.5. Create mesh and compute flow diagnostics

E.6. Visualise and export results from flow diagnostic

E.7. Load exported surfaces into other reservoir modelling packages