12. Defining Fluid Contacts

12.1 Creating a New Contact Set

After having built a Petrel 3D grid and prior to running the volume calculation, the various contacts should be defined in the *Make Contacts* process. Several sets of contacts can be defined and each Contact Set can contain a number of different contact types. All Contact Sets will be stored under a folder called *Fluid Contacts* in the Petrel Explorer Models tab. The Contact Set can be created based on a constant depth value or a surface. If a surface is used as an input for the contact, it has to exist in the Petrel Explorer window. Any type of surface can be used as an input. The user has the option of using the same contact for all zones and segments, different contacts for each segment and/or different contacts for each zone.

To create a new fluid contact:

2. Create a Gas Oil Contact by selecting it in the *Make Contacts* dialog box so that it is highlighted in gray. In the text box below the label “All Segments”, type a value of -1880 as shown in Fig. 12.1 (left).
3. Create an Oil Water Contact by selecting it in the *Make Contacts* dialog box so that it is highlighted in gray. In the text box below the label “All Segments”, type a value of -2010 as shown in Fig. 12.1 (right).
4. Click OK and the Fluid Contacts folder with the new set of contacts will appear in the model tab below the segment filter.

![Fig.12.1: The Make Contacts dialog box](image-url)
12.2 Visualizing the Contacts on a Surface

Visualizing the contacts on surfaces in both 2D and 3D is illustrative for showing the extent of the oil/gas zones.

To visualize contacts on a surface, follow the steps:

1. Expand the Fluid Contacts and the Contact Set folders under the Models tab.
2. Open the Horizons folder under the active 3D grid and display the Top Tarbert horizon in a 3D display window, see Fig. 12.2.
3. Display the oil/water and gas/oil contacts.
4. Open Settings > Style tab for the Fluid Contacts folder. Toggle on Show fill on horizons to drape the contacts over the horizons, see Fig. 12.3.
5. Select the Show contact surfaces to also see the contacts as surfaces. See Fig. 12.4 and 12.5.

Fig. 12.2: Top Tarbert and Fluid Contacts displayed in a 3D window
Fig. 12.3: Fluid Contacts displayed in a 3D window with Show fill on horizons checked
Fig. 12.4: Fluid Contacts displayed in a 3D window with *Show contact surfaces* checked
Fig. 12.5: Fluid Contacts displayed in a 3D window with Show fill on horizons and Show contact surfaces checked
12.3 Visualizing the Contact in Map View

Map view is a window especially designed for creating scaled plots. The contacts can be visualized in this window (and printed from this window as well, if desired).

To visualize contacts in map view, follow the steps:

1. Open a New Map Window from the Window menu.
2. Display one horizon from the 3D grid, say Top Ness as shown in Fig. 12.6.
3. Display the set of contacts to be visualized on top of this horizon, see Fig. 12.7.

Fig. 12.6: Top Ness horizon displayed in a map view window
Fig. 12.7: Top Ness horizon displayed in a map view window with the Gas Oil Contact displayed
Fig. 12.8: Top Ness horizon displayed in a map view window with both Gas Oil and Oil Water Contacts displayed
12.4 Visualizing the Contact as Property in 3D

This operation allows the user to create a property where the cells are given a facies code according to their position related to the hydrocarbon contacts. This may be a useful way of displaying the contacts. The generated contact property can also be used as input for other property calculation, for instance in the Property Calculation, or for filtering.

To visualize contacts in 3D, follow the steps:

1. Right click on one Contact set.
2. Select Settings.
3. In the Settings window, select the **Operations** tab, as shown in Fig. 12.9

![Fig. 12.9: Settings for 'Contact Set' dialog box](image)

4. In the “**code above the highest contact**” drop down box, select Gas Zone,
5. In the column **Facies value below contact**, specify: Oil Zone below the Gas Oil Contact, and Water Zone below the Oil Water Contact.
6. Click **Make Property**. A new property model called **Contacts** will be added inside the Properties folder in the **Models** tab. Click the **OK** button.
7. Uncheck the **Contact Set** under Fluid Contacts and check the **Contacts** property. See Fig. 12. 10 and 12.11.
Fig. 12.10: The Contacts property displayed in a 3D window
Fig. 12.11: The Contacts property along with Contact Set displayed in a 3D window
To get a cross-sectional view of the model, visualize the property on a General Intersection Plane and clip the plane in front or behind.

To visualize the property on a General Intersection Plane, follow the steps:

1. Right click the **Intersections** item in the **Models** tab,
2. Select the **Insert General Intersection** item from the drop down menu. See Fig. 12.12.
3. Play the property using the available options in the General Intersection Player toolbar with and without clipping the plane.

Now spend some time playing with the different options on the General Intersection Player toolbar to get you familiar to using such capabilities.

**Fig. 12.12:** The **Contacts** property is visualized on a General Intersection Plane
13. Volume Calculation

Volumes are most commonly calculated in the Volume Calculation Process step. Volumes can be calculated exactly within zones, segments and user defined boundaries (e.g. License boundaries). The Contacts defined in the previous process (Make Contact) are used as input to the Volume Calculation process. Volume calculations can be performed using several hypotheses in one operation. Each hypothesis is called a run. The user has the option to include an uncertainty range for the contact level and create distribution functions based on this uncertainty range.

The procedures for defining the set-up used for volume calculations are straightforward but require that the user thoroughly check all the volume menus. All settings related to the Volume Calculation process are found under the Volume Calculation process window. The user will specify the type of output to generate (3D properties, report, distribution functions) and which input to be used. Uncertainty analysis can be done based on an uncertainty in the hydrocarbon contact level. If you want to see the effect of different property models, then you should define several Volume Runs, each using different inputs (such as property, contacts or boundary). A report defined by the user will be created after the process has been run, listing all the volumes per zone, per segment and/or per facies. The user also has the option of creating a volume height maps (typically a HCPV or STOJIP map).

13.1 Calculate a bulk volume above a contact

To calculate the bulk volume, follows the steps:

1. Open the Volume Calculation process, located in the Process Diagram.
2. Create a new case by clicking in the Create a new case and name it for example HAH as shown in Fig. 13.1.
3. Specify Common Settings:
   - Toggle on the Results tab to create and define which parameters to be included in the report under Output tab. Toggle on Bulk volume both in the Make property column as shown in Fig. 13.1 and in the in report column which you can displayed in the Report Setting… as shown in Fig. 13.2,
Fig. 13.1: Volume Calculation dialog show Make property tab under Results tab

Fig. 13.2: Report column which displayed from Report Setting
• Specify Report settings. The Detail level defines the order of reporting. If zone is level 1 and segment level 2 that means that the volume will be reported for each segment in each zone as shown in Fig. 13.2,

• Define the Units of the reported numbers in the Volume format tab as shown in Fig. 13.3,

![Fig. 13.3: The Volume format for change the Units](image)

• Contacts tab: Select ‘Hydrocarbon Interval(s): Oil and Gas’ and enter the gas-oil contact and the oil-water contacts by first selecting them in the Fluid Contacts folder and then clicking on the blue arrow as shown in Fig. 13.4,

![Fig. 13.4: Contacts tab to select OWC&GOC](image)
• Under the Gen.Properties enter the general properties to be used. If you do not have a N/G property in your project then use a constant value of 0.8 as shown below,

[Image of Gen.Properties]

• Under the Oil Zone Properties choose which Water saturation (Sw) property to be used and enter the Bo 1.21. If you do not have a Sw properties in your project then use a constant value of 0.3 as shown below,

[Image of Oil Zone Properties]

• Under the Gas Zone Properties choose to use a constant Water Saturation and enter Bg=0.0009 as shown below.

[Image of Gas Zone Properties]
4. Press OK to calculate the volumes.
5. A report will be created by pressing Run. Copy the entire report and then click on Copy Open Excel or another editor and select Paste from the Edit menu as shown in Fig. 13.5.

Fig. 13.5: Report results of bulk volume property

6. Also you can display the results as chart in program like Excel as shown in Fig. 13.6.

Fig. 13.6: Chart from Excel shown the bulk volume for all Segment
13.2 Creating a STOIIP Map

A hydrocarbon column height map is the sum of all the values in the same X, Y position. For instance, a STOIIP map will show the sum of STOIIP for every X, Y position in the entire grid. It will therefore show you where to expect the highest concentration of oil.

How you can create a STOIIP map:

- Close the report window,
- Open the Volume Calculation process. Use the same settings as define previously but toggle on STOIIP as the Volume Height map to be created. If you don’t want to regenerate the Bulk volume, you should toggle this option off. You can toggle on the option “Overwrite existing properties” (in the lower left part of the window) to not create all the same properties over again as shown in Fig. 13.7.

![Fig. 13.7: Dialog box shown where you should toggle to create a STOIIP map](image)

- After running the process, the map will be placed in a folder in the bottom of the Petrel Explorer Input tab as shown in Fig. 13.8.
Fig. 13.8: The STOIIP MAP Displayed in a 3D Window
13.3 Draping the STOIIP map on a depth surface

To draping the STOIIP map on a depth surface, you need to do the following steps:

- Open the settings for the Top Tarbert horizon, found under the Horizons folder under 3D Grid (DC) in the Petrel Explorer Models tab. Under the Output tab, select ‘Make Surface’. The generated surface will be stored under the Input tab in the Petrel Explorer as shown in Fig. 13.9,

![Fig. 13.9: The Settings dialog box of Top Tarbert under Horizons folder](image)

- Open the Settings for the generated Top Tarbert depth surface. Under Style tab >Solid> Color choose 'Textured'. Select the STOIIP map by clicking on it in the Petrel Explorer input tab (make it bold) and enter it into the 'Surface/image/Seismic' position by clicking on the blue arrow 1 as shown in Fig. 13.10,
• Press OK,
• Display the Top Tarbert depth surface to see the STOIIP map draped over it. Remember that you are now draping the STOIIP map for all zones on to the Top Tarbert surface as shown in Fig. 13.11.