USE CASES

BEFORE YOU GET STARTED

The Web Enabled Drilling Simulator of OpenLab is based on NORCE's computer models of well flow and drillstring mechanics. With the Web Enabled Drilling Simulator, these computer models are made available through an intuitive interface. Although the computer models behind the simulator are among the world's most detailed models of their kind, we have tried to create an environment where you can easily run simulations and handle the results.

We hope you will find this document useful. For more information about OpenLab Drilling, you are welcome to find more information on openlab.app.

CONTENTS

USE CASE 1: BACK PRESSURE MANAGED PRESSURE DRILLING

The back pressure Managed Pressure Drilling (MPD) method, sometimes called Constant Bottom Hole Pressure MPD, is expected to be known. Please read the literature on this topic if not. One relevant paper can be found [here](#).

Manual back pressure control using the Manual mode

<u>Part 1</u>

1. Create a new **configuration** named "My default MPD case" based on the offshore type of rig template "InclinedWell_2500m".

- Check that your open hole diameter is 8 ½ inch.
- Check that your mud density is 1.65 sg.

2. Create a new **simulation** named "My first manual BHP control". Under "Advanced settings" check that the bit depth is 2500 meter at start of simulation. Choose "No influx and loss". Press Create new simulation.

3. Start your simulation with Main pump flow rate set at 2000 L/min.

- Check that your Bottom Hole Pressure reaches approximately 360 bar when the circulation is steady at 2000 L/min.

4. Adjust your MPD choke opening so that your Bottom Hole Pressure increases to 380 ± 1 bar.

- Check that your MPD choke opening needs to be approximately 14.3 % to achieve 380 ± 1 bar when circulating at 2000 L/min.
- Observe that your ECD is very close to (or exceeding) the fracture pressure gradient.

5. Keep the simulation running until you have answered Question A below.

<u>QUESTIONS:</u>

<u>A:</u>

From the Add/remove graphs button, add the depth-based temperature profile. Then, simulate for a longer time (run in fast-forward mode), e.g. 15 minutes or longer. How does the dynamic temperature affect the Bottom Hole Pressure? What adjustments would you need to do on the MPD choke opening to maintain 380 bar?

B:

Why is the ECD high (and approaching infinity) in the upper part of the well?

Part 2

1. Go to the Geopressure Editor in your current configuration ("My default MPD case").

● Check that the pore pressure gradient is varying between 1.3 and 1.4 sg in the open hole section.

2. Go to the Fluid Editor and change the fluid Density to 1.3 sg.

3. Create a new **simulation** named "My second manual BHP control" with the bit at 2500 meter depth. Choose "No influx and loss".

4. Start your simulation with Main pump flow rate set to 2000 L/min.

● Check that your Bottom Hole Pressure is reaching approximately 286 bar when circulating at 2000 L/min.
● Observe from the ECD graph that your ECD is now below the pore pressure (underbalanced) in parts of the open hole section.

5. Change your MPD choke opening to 15%.

● Check that your ECD now exceeds the pore pressure in the entire open hole section.

QUESTIONS:

A:

What would have happened in real life when the well was underbalanced and if the formation in the open hole contained hydrocarbons?

B:

What characteristics of the formation would have affected the severity of the situation?

Part 3
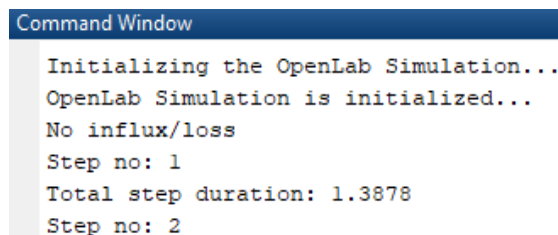
Automated back-pressure MPD using Matlab

1. Create a new **configuration** named "My Matlab MPD case" based on the offshore type of rig template "InclinedWell_2500m".
2. Start Matlab (ver 2016b or later). Download the Matlab package from the OpenLab web page (at the bottom of the page), unzip on your computer and set this folder as your working directory in Matlab.
3. Open the file "`GetLoginData.m`". Enter your user name for the variable "`username`" and paste the "api-key". The API key is found in the Settings menu in the Web client.

4. Open the file "FlowSweep_BHPControl.m". Change "`ConfigurationName`" to "My Matlab MPD case". At "`SimulationName`" you can enter a name of your choice or keep the default.

To initialize a simulation, you need to set the initial parameters which are the same as when you create a new simulation in the web client. We have already done this for you in the example files in Matlab. In FlowSweep_BHPControl.m . It looks something like this :

```
[username, api_key, license_guid] = GetLoginData();

ConfigurationName = 'My Matlab MPD case';
SimulationName = 'Flow sweep with bhp control (Matlab)';
InitialBitDepth = 2500; % [m]
```

You are now ready to simulate from Matlab. The case we are using in this part is a flow sweep (flow is ramped down and up) and a PI control algorithm is used to maintain a constant bottom hole pressure.

5. Press the Run button in Matlab or enter "FlowSweep_BHPControl" in the command window in Matlab. A simulation should now start, and you should see this in the command window.


```
Command Window
  Initializing the OpenLab Simulation...
  OpenLab Simulation is initialized...
  No influx/loss
  Step no: 1
  Total step duration: 1.3878
  Step no: 2
```

● Observe from the "Bottom Hole Pressure" graph that the bottom hole pressure becomes stable at 380 bar throughout the automated BHP control sequence (which in this case starts at 100 seconds).

You can run until the simulation stops (defined by the variable "*MaxTimeSteps*" in FlowSweep_BHPControl.m). If you want to terminate the simulation, you can enter Ctrl+c in the Matlab command window.

6. Go back to OpenLab in your web browser and open the configuration named "My Matlab MPD case". On the list of simulations, you should now see a new simulation with the name that you have specified from Matlab. The results can be viewed and downloaded from OpenLab, or you can use Matlab to save, plot and present your results.

QUESTIONS:

A:

What is the minimum MPD choke opening required to maintain 380 bar bottom hole pressure during the flow sweep?

B:

In Matlab, in the file FlowSweep_BHPControl.m, change the reference pressure (named ReferenceBHPPressure) from 380 bar to 390 bar, and run the flow sweep again. What is the minimum MPD choke opening required to maintain 390 bar bottom hole pressure during the flow sweep?

<u>**C:**</u>

In Matlab, in the file FlowSweep_BHPControl.m, change the reference pressure (named ReferenceBHPPressure) back to 380 bar. Change the Proportional gain, Kp, from (-0.015) to (-0.005). Run the flow sweep again. What is the effect on the MPD choke opening during the sequence?

D:

Keep the reference pressure (ReferenceBHPPressure) at 380 bar. Change the Proportional gain, Kp, from (-0.015) to (-0.025). Run the flow sweep again.

How well does the PI controller perform now compared to the original settings?

USE CASE 2: CIRCULATING A KICK

<u>**Part 1**</u>

1. Create a new **configuration** named "My default influx case" based on the offshore type of rig template "InclinedWell_2500m".

●   Check from the Geopressure pressure gradient that the pore pressure is varying between 1.3 and 1.4 sg in the open hole section.

2. Go to the Fluid Editor and change the density of Fluid 1 to 1.36 sg.

3. Create a new **simulation** named "My first kick simulation". Choose Advanced Settings and check that the bit depth is 2500 meter. Under the "Influx and loss", **Choose "Based on geopressure"**. Create your simulation.

4. Without changing any of the setpoints, start the simulation. Run the simulation until time has reached approximately 2 minutes (use fast-forward) and press the Pause button. Open the "Gas volume" graph and observe that there is gas in the lower part of the well: Open the "Add/remove graphs" modal and make your selection: Remember to save graph selection as default before closing the modal window.

5. Without changing any of the setpoints, continue the simulation. Run the simulation until time has reached approximately 5 minutes (use fast-forward) and press the Pause button.

●   Observe from the "Reservoir flow rate" graph that the "Total influx mass" has reached a value between 50 and 60 kg (115 to 120 lbs).

You have now created a small gas [kick](#). We will use this case for Part 2. Do not Stop the simulation but keep it paused before Part 2.

<u>**Part 2**</u>

Use your **simulation** named "My first kick simulation" that you created in Part 1.

●   Observe from the "Gas volume" graph that there is gas in the lower part of the well.

1. Continue your simulation but with "Main pump flow rate" set to 1000 L/min. You are now transporting the gas up in the annulus.

2. Pause the simulation after it has reached 10 minutes (use fast-forward).

<u>QUESTIONS:</u>

<u>A:</u>

When you started the pump at 1000 L/min, how did it affect the Reservoir flow rate?

<u>B:</u>

Your ECD is still very close to the pore pressure. What do you think will happen if you continue to circulate out the gas keeping the Main pump flow rate at 1000 L/min and not changing any of the other setpoints?

Part 3

In the current version of the simulator, we do not allow circulation through a [choke line](#). Instead we use the MPD choke to add back-pressure and allow formation gas to be circulated through the MPD choke. In real life a kick would normally be circulated through the choke line, dependent on the size of the influx. However, the basics around the circulation procedure will be the same. The following example is therefore the MPD version of [the Wait & Weight method](#).

1. Use your existing **configuration** named "My default influx case" based on the offshore type of rig template "InclinedWell_2500m".
2. Go to the Fluid Editor and change the density of Fluid 1 to 1.33 sg (based on Generic OBM 1)
3. Create a new **simulation** named "My MPD kick circulation" with the bit at 2500 meter depth. Under the "Influx and loss", Choose "Based on geopressure". Create your simulation.
4. Start your simulation with Main pump flow rate at 2500 L/min. Run the simulation until time has reached approximately 1 minute (use fast-forward if you want) and press the Pause button.

- Observe from the "ECD" graph that your well pressure is balanced against the formation pressure.

5. Continue your simulation but change the Main pump flow rate to 1000 L/min. Run the simulation until time has reached approximately 2 minutes (use fast-forward) and press the Pause button.

- Observe from the "Gas volume" graph that there is gas in the lower part of the well. Reducing flow rate from 2500 to 1000 L/min has reduced the ECD, causing influx to the well.
- Observe from the "Stand Pipe Pressure" graph the pump pressure at 1000 L/min. This is going to be your slow-rate-friction-pressure, P_slow, in the circulation procedure. In this case you should get in the region 31 bar.

You have now created a gas kick. We will circulate it out using the MPD choke and replace the mud with heavier mud to regain overbalance.

6. Change your "Main pump flow" rate to 0 L/min.
7. Close the BOP and run the simulation until time has reached approximately 3 minutes and press the Pause button.

- Observe from the "Reservoir flow rate" graph that the gas influx has stopped.

During a well shut-in, the pressure in the wellbore stabilizes, and the formation pressure equals the hydrostatic pressure, plus the added pressure needed at the top of the hole (shut-in drill pipe and casing) to avoid influx. The drill pipe at this time is full of known-density fluid. Therefore, the formation pressure can be calculated using the shut-in drill pipe pressure (SIDPP).

8. Change your Main pump flow rate to 200 L/min. Run the simulation until time has reached approximately 3 ½ minutes and then press the Pause button. This is to open the non-return flow valve on the BHA.
9. Change your Main pump flow rate to 0 L/min. Run the simulation until time has reached approximately 4 minutes and then press the Pause button.

- We will now find the shut-in drill pipe pressure. In this case it cannot be measured directly since we have circulated after we closed the BOP, in order to open the non-return valve. This is how you find the shut-in drill pipe pressure in this case:
    - Read the drill pipe pressure from the "Stand pipe" graph. This will depend on the amount of influx and how much the BOP pressure has increased during the shut-in when you opened the non-return valve. If you have followed the steps carefully, you should have approximately 17 bar stand pipe pressure now.
    - However, to find the shut-in drill pipe pressure you need to extract the increase in BOP pressure during the opening of the non-return valve. The BOP pressure increased from approximately 38 to 45 bar during this time. Therefore, the amount of pressure to extract from the stand pipe pressure is 7 bar.
    - The shut-in drill pipe pressure is therefore 16 bar minus 7 bar, equals 10 bar.

Calculate your kill mud weight δ_{kill} by using the formula:

$$\delta\_kill = \ \delta\_0 + P\_SIDP \ / \ (9.81 * Vertical \ Depth \ ) + \delta\_Safety$$

Here, $\delta_{\_0}$ is your initial fluid density (1330 kg/m3 in SI units in this case) and $\delta_{\_Safety}$ is a safety margin. In this case we can use $\delta_{\_Safety} = 20\frac{kg}{m3}$, (0.02 s.g). The Vertical Depth can be found from either the Geopressure Editor or the Wellpath Editor. In this case, the depth of the influx zone is around 2000 meter Vertical Depth.

In this case we get:

$$\delta\_kill = \ 1330 + (10 * 10^5) \ / \ (9.81 * 2000 \ ) + 20 = 1401 \ kg/m3$$

10. In your simulation view, activate the "Control active pit" and set the "Active pit density" to 1401 kg/m3. (1,40 s.g.).
11. Set "Main pump flow rate" to 1000 L/min. This is your kill rate. In this example you should have read approximately 31 bar stand pipe pressure when circulating at 1000 L/min.
12. Set the MPD choke opening to 9%.
13. Start the simulation and run 10 seconds and press pause. This is to avoid the bottom hole pressure to decrease before circulation through the MPD choke is obtained.
14. Open the BOP.
15. Start the simulation and adjust the MPD choke opening carefully so that your Stand pipe pressure is equal to $P_{slow} + P\_SIDP$. In this case 10 bar + 31 bar = 41 bar. Hint: Instead of using the slider to adjust the choke opening, use the up/down arrows, or scroll with your mouse. In this example you may start with choke opening at 9% and then adjust choke opening manually until you reach 41 bar on your Stand pipe pressure.
16. Continue to circulate with 1000 L/min while carefully adjusting the choke opening so that Stand pipe pressure is following a (more or less) linear curve from 41 bar to 31 bar when the kill mud passes the bit. This should take about 25 minutes (based on the inner volume of the drill string and the flow rate).

17. Then adjust your MPD choke to keep 31 bar Stand pipe pressure as gas is circulated out of the well and the kill mud has filled the annulus.
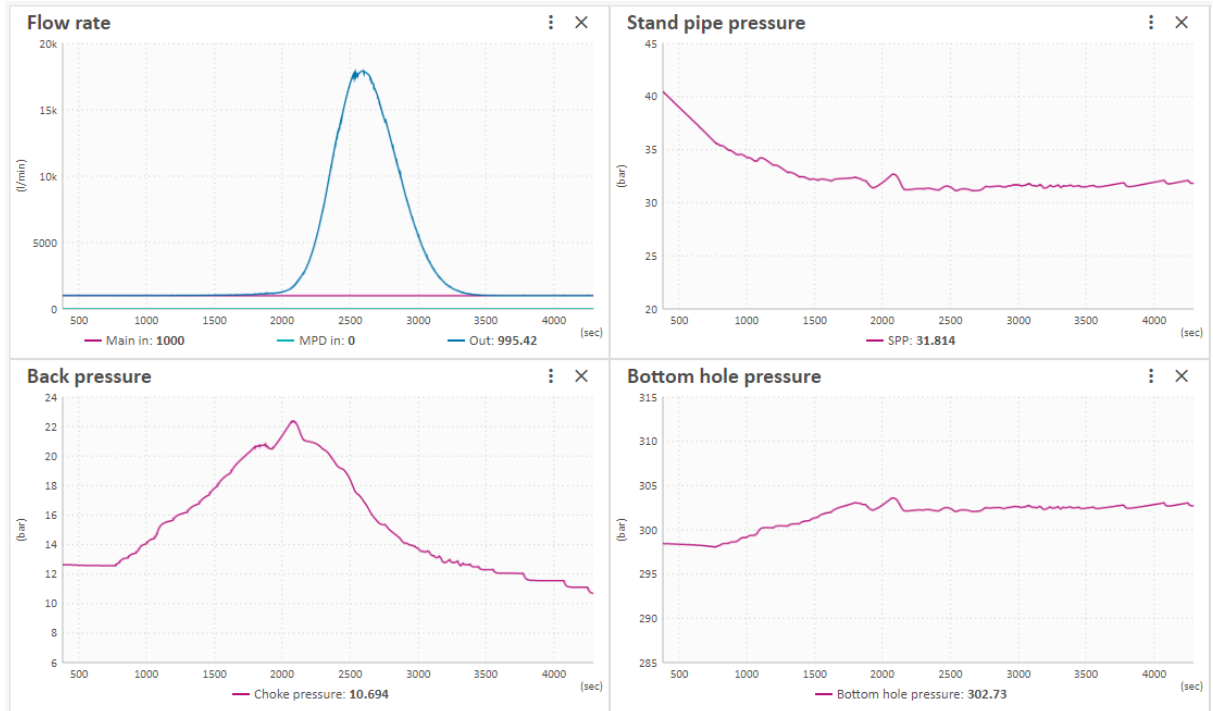
Note: Dependent on your user account and simulation limit, the maximum number of time-steps may not allow a full well control circulation.
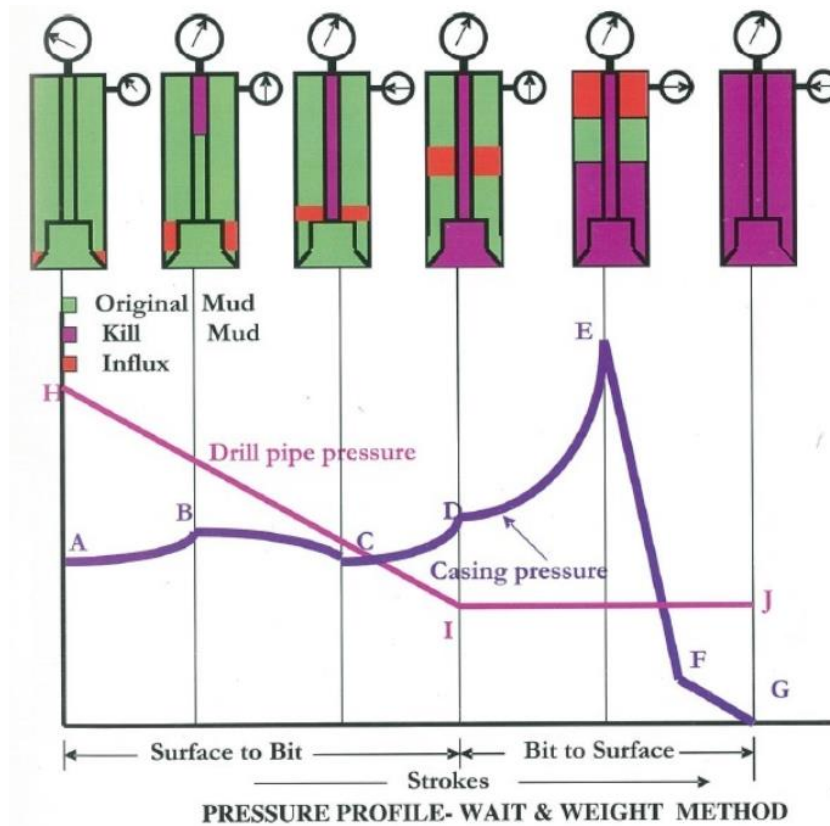

The Wait and weight method: Source ONGC Well (http://www.ongcindia.com)

11

USE CASE 3: CUTTINGS TRANSPORT

To learn about how to start drilling please open the User guide and Chapter 3.3 under the section "Example on how to start drilling".

Part 1

1. Create a new **configuration** named "My default drilling case" based on the offshore type of rig template "InclinedWell_2500m".
2. Create a new **simulation** named "My first drilling simulation" with the bit at 2500 meter depth. Under the "Influx and loss", keep "No influx and loss". Create your simulation.
3. Open the "Add/remove graphs" modal and make the following selections: Remember to Save graph selection as default.
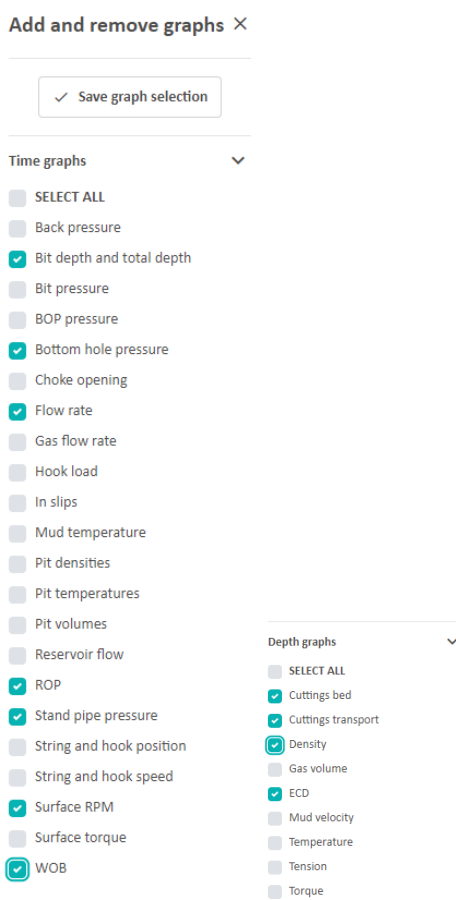
4. Close the "Add/remove graphs" modal.
5. Start your simulation with Main pump flow rate set to 2200 L/min and Surface RPM at 120 rpm. Run the simulation until time has reached approximately 1 minute (use fast-forward) and press the Pause button.

You then need to set bit on bottom to apply weight on bit.

6. Set Top of string speed to a positive value and run the simulation until "Bit depth" is identical to "Total well depth". Then press the Pause button.

Total well depth: 2500.06 m
Bit depth: 2500.06 m

■ ❙❙ ⊙ ▶❙ ▶▶

You are now ready to drill.

7. Apply weight on bit by setting ROP to 20 m/h (65 ft/h).
8. Continue your simulation with Main pump flow rate 2200 L/min, Surface RPM 120 rpm and ROP 20 m/hr. Run the simulation until time has reached approximately 10 minutes (use fast-forward) and press the Pause button.

● Observe from the "Cuttings transport" graph that there are cuttings in suspension from the bit up to 2000 meter.
● Observe from the "Cuttings bed" graph that there are cuttings accumulated in a cuttings bed around 2000 meter depth.

9. Change Main pump flow rate to 2500 L/min and run the simulation until time has reached approximately 11 minutes.

● Observe from the "Cuttings bed" and the "Cuttings transport" graph that the cuttings bed have been eroded and that cuttings are now transported.

10. Decrease Surface RPM from 120 to 100 and run the simulation until cuttings bed starts to accumulate again in cuttings bed. Stop the simulation.

QUESTIONS:

A:

Why is the cuttings bed building around 2000 meter depth?

B:

What are the most important drilling parameters that affect cuttings transport?