From: Khanna, Samir
Sent: Fri May 07 16:00:57 2010
To: Lockett, Tim; Austin, Julian; Hill, Trevor
Cc: Fairhurst, Paul
Subject: RE: Plumes
Importance: Normal
Attachments: Plume40000.jpg; Velocity40000.jpg
Attachments: Plume40000.jpg; Velocity40000.jpg

Tim,

I have done many sensitivity studies on my end, including adding current and a larger domain of the sea above the trench, much higher resolution. But the core picture remains the same. I have even tried a simple Boussinesq-type Dimensional analysis that gives the diatance L at which the plume turns as rho * V^2/ (Delta rho * g), showing that at these velocities Buoyancy dominates. One issue here might be what I am plotting. Take a look at the following 2 pictures from the same calculations for 40k bbl. One is showing just oil-gas presence (the picture I sent earlier), the other shows the velocity at the plane of symmetry.

I think what you are seeing is just the movement of gas-oil-water in that region. Does the second image look anything like what you see?

Another issue I can think of is that the opening might be restricted and hence the jet comes out at a higher velocity, or our density estimates are low.

I'll try using 2 scalars to add slip between gas and oil and see if that makes any difference.

Based on what you are observing, can you suggest what I should be plotting?

Thanks.

Samir

From: Lockett, Tim
Sent: Friday, May 07, 2010 3:24 AM
To: Khanna, Samir; Austin, Julian; Hill, Trevor
Cc: Fairhurst, Paul
Subject: RE: Plumes

Samir

I was commenting on the angle at which the plume rises. On the video the plume appears to go across the crater/trench away from the inlet (riser discharge point) while rising. With the plots you have, the plume is tending to rise straight out of the inlet (particularly at the lower flowrates).

The only way I can think this is working is that in your simulation the plume is staying 'coherent' so that the density is low, whereas in practice I therefore assume the plume is entraining a lot more water so that the average density is brought up to be closer to that of water, hence the bouyancy term is reduced. This is a function of the turbulence model. I guess turbulent intensity is an input for the inlet boundary condition - this is a multiphae flow and so will have a higher level of turbulence than for a single phase flow at the same velocity.

10333
Exhibit No.
Worldwide Court Reporters, Inc.

Alternatively, or as part of the above, it is also possible that the plume is translating across the crater/trench further in practice because of gas-oil slip effects (gas moving faster, oil moving slower than modelled) compared with the homogeneous model.

For the other work...
In regard to the plume rise to surface CFD modelling work, I would think the knock-on effect is that in practice the origin of the plume is the equivalent of being more spread out at sea-bed and therefore presumable it will be more spread out at the surface than shown in the model. In terms of physics, the gas density decreasing with depth is very important. Equally, the oil phase gets more dense and releases gas. I presume the oil density change will mean that it rises slower as it gets heavier and hence will spread out more on the way up. I don't know whether the initial size of the 'pool' arriving at the sea surface was measured/photographed, but given these comments I would expect that the CFD is currently giving a smaller surface arrival 'pool'.

I think this is still useful - for instance in regard to giving a view of gas/oil rise time (sea bed to surface) and gas concentration in the vicinity of the Discoverer Enterprise if the collection riser system stops working (eg for process upset).

Best regards
Tim

From: Khanna, Samir
Sent: 06 May 2010 16:54
To: Lockett, Tim; Austin, Julian; Hill, Trevor
Cc: Fairhurst, Paul
Subject: RE: Plumes

Tim,
A few more comments/questions:
1) I am predicting the mean plume using Reynolds-averaged turbulence models, you are observing an instantaneous snapshot of the plume, based on turbulence intensity the two can differ quite a bit.
2) What do you mean by rising strongly…the location of the plume coming out of the trench, the width of the plume, or the rise velocity?
Thanks.
Samir

From: Lockett, Tim
Sent: Thursday, May 06, 2010 3:33 AM
To: Khanna, Samir; Austin, Julian; Hill, Trevor
Cc: Fairhurst, Paul; Webster, Simon
Subject: RE: Plumes

Samir

Simple answer is that none of these pictures are like the video footage, as the plume is rising too strongly, so something must be wrong with the physics we are trying to represent or model set-up.

A couple of questions/comments then:

- Somewhat surprised by the speed of turn around of these cases. Are these 3-D simulations or

2-D? If 2-D then I would expect the solution will set up a water circulation zone between the plume rising and the back wall of the box. This will prevent the plume coming over to the far wall. In 3-D the water entrainment can come down the sides, so I feel this is important.

- Influence of the box depth. What happens if we don't have a box and just introduce the plume at sea level, or maybe try a box of 2-D or 5-D depth?

- Influence of the shape of the back wall. As per my sketch, maybe the back wall isn't vertical but is sloped? Would this allow the plume to stay low and hit the back wall?

- Influence of sea current. If we have a current aligned with the direction of the plume? Probably wouldn't have a great effect with a 10-D depth, so more like try this with a shallow trench.

For these three, rather than running lots of cases, may be better to persue the option of getting more data.

In relation to the other discussion with Chris Matice, have you agreed that the length scales you are interested in are quite different as I presume his modelling is the plume rise and spread over the 'far-field'.

Best regards

Tim

From: Khanna, Samir
Sent: 06 May 2010 04:23
To: Lockett, Tim; Austin, Julian; Hill, Trevor
Cc: Fairhurst, Paul; Webster, Simon
Subject: Plumes

Attached are 3 pictures of the predicted plume, at 10k, 40k, 100k stbblpd, using only 1 scalar. The calculations were quick and dirty, I can fine-tune the numerics further, but they show some interesting information. The trench dimension is based on Tim's sketch. I contacted Dave for Metcon data but haven't received anything yet, so have not included any current. However, the effect of the current will only be felt above the trench. I haven't seen the video so don't know what the obervations have been. Please let me know if these calculations are helpful and I can pursue further.
Thanks.
Samir

<< File: Plume100000.jpg >> << File: Plume10000.jpg >> << File: Plume40000.jpg >>

From: Lockett, Tim
Sent: Wednesday, May 05, 2010 9:57 AM
To: Khanna, Samir; Austin, Julian; Hill, Trevor
Cc: Fairhurst, Paul
Subject: RE: CFD effort in Houston

Samir

Trench:
I have now seen video footage of the plume coming from the riser. I don't know who over there has a copy but maybe Trevor can help with access to this. You need to see it. Sorry I tried to grab some

screen shots but that doesn't work.

As a general description, imagine the overall domain for the sea with the bottom surface at the sea bed level, now join on a box (crater) which goes below sea bed level with walls on four sides and a wall on the bottom of the box, and the plume coming from the 19.5 inch riser end which is on the side wall at the bottom of the box. Attached is a very rough sketch.
<< File: Trench-rough-outline.ppt >>
The sides look pretty steep on the face where the riser is and the left and right sides. No visibility on the side opposite the riser where the plume is 'hitting' but I would expect this to have been worn away into an angle by the plume so that it is not steep sided.

For dimensions, I wonder if the ROV guys could help with this. Maybe they have measured the crater? However, in the absence of any other numbers, I would describe the crater as being 3-D across on the face where the riser is discharging, 6-D coming away from the riser, and set maybe 10-D depth below seabed although this is very difficult to judge, where D is the riser.

Overall I am concerned about how the CFD result will be influenced by the description of the trench!

For Metocean data please contact Dave Driver in GOM. I understand the data on sea currents has been provided to Mike Tognarelli for the riser design. I don't have this data.

Re: boundary conditions for S1, S2 I was referring to an imposed sea current having an inflow boundary (specified velocity). If you are using pressure boundaries then that wouldn't apply.

Tim

From: Khanna, Samir
Sent: 05 May 2010 14:54
To: Lockett, Tim; Austin, Julian; Hill, Trevor
Cc: Fairhurst, Paul
Subject: RE: CFD effort in Houston

Tim,
This can be done, might be a bit more challenging numerically, but I'll setup both ways (1 and 2 scalars). For domain boundaries, I am using pressure boundary conditions, I can not set S1 = 1, S2 = 0, as the HC will leave the domain at some place (so S2 can't be 0 everywhere). Can you please send me the trench geometry and the location of riser relative to the trench, and also the information on sea current?
Thanks.
Samir

From: Lockett, Tim
Sent: Wednesday, May 05, 2010 7:22 AM
To: Khanna, Samir; Austin, Julian; Hill, Trevor
Cc: Fairhurst, Paul
Subject: RE: CFD effort in Houston

Samir

The flow regime in the riser is expected to be stratified flow.

Just a question: can you use two scalar tracers and set density & viscosity on the basis of these two numbers?

I was thinking that one scalar, S1, would be the water fraction, and the second, S2, would be the gas fraction, and then the density would be the combination of these two....

RhoHC = S2 * RhoG + (1-S2) * RhoL
Rho effective at any x, y, z = S1 * RhoW + (1-S1) * RhoHC

Likewise for Effective viscosity using the same S1, S2.

For the boundary condition on the pipe outlet / inlet into the domain, set S1=0,S2 = 1 for the top 30% of the area, and S1=0,S2=0 for the bottom 70% of the area to represent a stratified flow condition, and S1=1, S2=0 for the boundaries of the domain to represent the sea. You can then also set the velocity in the top 30% (gas) to be higher than that in the bottom 70% (oil) to represented slip between the phases at the outlet of the riser.

Assuming you can do this, the attached xls is an enhancement of the one I sent last night where I now deal with the phases separately on the inlet boundary to give two velocities.
<< File: CFD-inputs-v2.xls >>
Whether you use one scalar or two:
- What sea current velocity are you using? I understand MetOcean have some numbers for this for the area and depth, and they should be able to say how that is oriented relative to the riser-pipe-trench alignment.

Best regards

Tim

From: Khanna, Samir
Sent: 04 May 2010 23:42
To: Lockett, Tim; Austin, Julian; Hill, Trevor
Subject: RE: CFD effort in Houston

Tim,
Thanks for the information. We are modeling a horizontal discharge into seawater in a trench, right? What are the dimensions of the trench? Out of curiosity, what is the flow regime in the horizontal pipe? To account for multiphase physics here, I believe, CD-Adapco will have to use the Volume-Of-Fluid method, which could take a while, but might be good confirmation.
Samir

From: Lockett, Tim
Sent: Tuesday, May 04, 2010 3:56 PM
To: Khanna, Samir; Austin, Julian; Hill, Trevor
Subject: RE: CFD effort in Houston

Samir

Attached is an XLS which I believe gives you the input you need. I will talk you through it as it is late here... If you spot something you don't understand or don't agree with then please shout and I will look again tomorrow.
<< File: CFD-inputs.xls >>
The xls comprises 2 PVT flash calculations from the latest fluid model. The left hand one is at sea

bed P and T. The right hand one is at Stock Tank conditions.
To simulate a given case, I believe you need to input the density and viscosity of the 'no-slip' approximation to the gas-oil fluid. This comes from the left hand block.

You also need the velocity of the flow coming out of the end of the riser. I have assumed you will be setting a uniform velocity profile on a pipe with diameter 19.5 inches. The flowrate is calculated from the stock tank oil rate (eg 10000 sbbl/d) and is then turned into a total mass flowrate (gas+oil) at stock tank conditions. This is then divided by the density at sea bed conditions of the 'no slip' fliud, and then divided by the cross-sectional area of the riser to give the velocity of the fluid on that boundary.

I suggest you run a couple of flowrate cases (depends how long each takes and how many you can run in parallel). Change cell T9 to get the velocity input for another flowrate.

In the future, we will do multiphase CFD properly, but I agree that with the current requirements we do not have the time in the current context. However, if we want to challenge this view, or run in parallel, then I am willing to open dialogue with CD-Adapco about getting this case run by them as a multiphase physics case subject to them agreeing confidentiality etc.

By way of reference, the 'no slip' effective properties are calculated using the liquid volume fraction (cell E12), and the reference slides for this are in the 3-day flow assurance course, lecture on fundamentals of multiphase flow, slides 32 and 35 (copy attached for info/reference).
<< File: Microsoft PowerPoint - 02 FA Houston - Multiphase Fundamentals.ZIP >>

Best regards

Tim

From: Khanna, Samir
Sent: 04 May 2010 16:13
To: Austin, Julian; Hill, Trevor
Cc: Lockett, Tim
Subject: RE: CFD effort in Houston

Yes, I have.
Thanks.
Samir

From: Austin, Julian
Sent: Tuesday, May 04, 2010 10:09 AM
To: Khanna, Samir; Hill, Trevor
Cc: Lockett, Tim
Subject: RE: CFD effort in Houston

Trevor,

That's how I would have suggested also. Samir, I take it you have capacity to perform this analysis?

Kind regards,

Julian

From: Khanna, Samir
Sent: 04 May 2010 16:03
To: Hill, Trevor
Cc: Austin, Julian; Lockett, Tim
Subject: RE: CFD effort in Houston

Trevor,

The easiest approach would be if we can treat the oil-gas as a homogenous mixture with effective fluid density and viscosity, then I can do a single-phase CFD calculation by accounting for density/viscosity difference between sea-water and oil-gas mixture by using a tracing-scalar. I think this would be the best approach, unless Tim says otherwise. All we need is the relative gas to oil ratio and how it impacts the effective properties.

Thanks.
Samir

From: Hill, Trevor
Sent: Tuesday, May 04, 2010 9:31 AM
To: Khanna, Samir
Cc: Austin, Julian; Lockett, Tim
Subject: RE: CFD effort in Houston

Samir

Thanks for this... I have not yet caught up with Richard on what he is doing... please would you contact him directly (two in GAL) to offer help.

Samir/Julian

Would you give some thought to how we could model the two phase gas-oil flow coming out of the end of the horizontal 20 inch pipe discharging into seawater in a trench... if we can look at how quickly (how far from the pipe) the gas-oil mixture starts to move vertically, then we may be able to reduce the range of uncertainty in flowrate.

This would need an interface with Tim who can give the expected gas and liquid velocities in the pipe as a function of flowrate.

Please let me know what you think you and your contacts could provide... gas-liquid-liquid interaction...

Happy to have a phone call/conference if that would help clarify what we are after...

Regards
Trevor

From: Khanna, Samir
Sent: 03 May 2010 14:49
To: Hill, Trevor
Cc: Austin, Julian

Subject: CFD effort in Houston

Trevor,

My Fluent contact (CFD software vendor) called to let me know that Houston HPC bought many Fluent licenses over the weekend to help with the GoM effort. The BP contact was Richard Simpson. In case they need my help, please let me know, as I have good experience with Fluent. Thanks.

Samir Khanna
BP
(630) 961-7922 [Office]
(630) 605-7618 [Cell]