



Re: info for CFD simulations

Franklin Shaffer o James J Riley

06/01/2010 01:12 PM

pmbommer, "George Richards", "Madhava Syamlal", "Mehrdad  
Cc: Shahnam", savas, "pedro espina", Bill.Lehr, "Steven T. Wereley",  
aaliseda, "Juan Lasheras", "Marcia K McNutt"

---

\*\* High Priority \*\*

The quick answer is that we have several codes, including commercial codes like Fluent and our own in-house codes like MFIX. We run these codes on massive parallel computer systems with >760 CPU's and supercomputers in Pittsburgh and Oak Ridge.

I believe the first approach will be to start with simpler models and simulate just an oil jet under the conditions of the jets we are looking at. The simpler codes will be followed or run at the same time as LES level simulations. LES level simulations can take days to run.

I'm not running the CFD codes myself, but I will forward this email to the team of CFD experts and let them respond directly. We should coordinate our efforts and numbers with them.

Frank

>>> "James J Riley" <rileyj@u.washington.edu> 6/1/2010 12:29 PM >>>

Hi Frank,

Here are a couple of technical questions for the people doing CFD; perhaps, as you suggest, the CFD results might be useful in comparing and understanding the PIV results.

Do you know whether they are doing RANS or LES type of simulations? Either way, do you know what type of modeling they are using?

How do they determine the velocity (and composition) profile at the exit? Through simulating part of the pipe flow and the bend at the end, or assuming an outflow profile, or some other method?

How are they treating the large variable densities associated with the gas, oil and water? Low Mach number, fully compressible, or other?

Knowing this will help us to better interpret the results.

Thanks, Jim Riley

Franklin Shaffer wrote:

> \*\* High Priority \*\*

>

> Plume Analysis Team: Some of my DOE colleagues have started doing CFD  
> simulations of the oil leak jets. Now that we have more time to  
> estimate the maximum oil flow rate, they might be able to finish  
> their CFD simulations in time to aid our analysis. No promises, but  
> they will give it their best shot.

>  
> CFD will provide yet another independent estimation tool, and CFD  
> will tell us what is happening inside the jets -- we cannot see or  
> measure that now. By matching the CFD simulations with the BP videos  
> and our PIV measurements on the outside of the jets, CFD will be able  
> to tell use the velocity profile at the jet exit. At the exit we can  
> calculate oil flow rate without worrying about entrainment of water  
> into the jet.  
>  
> I believe they are starting by modeling an oil jet and a methane jet  
> separately, directed horizontally and vertically. They started CFD  
> simulations using properties of crude oil in water at 5000 ft. To do  
> the CFD simulations correctly, they need better numbers for the  
> following:  
>  
> \* oil properties: viscosity, density, temperature \* methane  
> properties: viscosity, density, temperature \* if possible, pressure  
> just upstream of the jet exit \* size of the jet exits \* anything else  
> you believe is pertinent  
>  
> Of course, we need this info for both the main jet at the  
> bend-of-pipe and the riser exit.  
>  
> I have cc'd my colleagues who are leading the CFD effort so you can  
> communicate with them directly. They are Drs. Madhava Syamlal,  
> Mehrdad Shahnam, and Geo Richards.  
>  
> If you have any of this info, please send it to use as soon as  
> possible!  
>  
> Thank you! Frank  
>  
>

-----  
James J. Riley, PACCAR Professor of Engineering

Department of Mechanical Engineering  
Box 352600  
University of Washington  
Seattle, WA 98195

Phone: (206) 543-5347  
FAX: (206) 685-8047  
email:  
rileyj@u.washington.edu

website: <http://faculty.washington.edu/rileyj/>  
-----