Q&A Report: 3D hydraulic modelling essentials

#	Question	Answer
1	Ray, can you model air entrainment in open foam?	Hi , yes. it can solve air entrainment, like my last slide shown. It requires higher computational resources. The main multiphase solver is interFoam which can model air and water together, although doesn't do all of the air bulking you'd see in the froth of a hydraulic jump
2	If yes, what are the air entrainment and transport models used?	Hi, for better air capture, I think LES turbulence model do a better job. The eddies would carry airs into the water, and breaking into small air bubbles by turbulent forces
3	It appears that USACE is underway to develop a GUI and a more "user friendly" open source CFD platform (https://www.erdcwerx.org/computational-fluid-dynamics-methods-for-water-resource-development/). Are there any other similar efforts underway?	Thanks for sharing, I think DualSPHysics is a good, free alternative too. I prefer modelling in windows so makes life easier. https://dual.sphysics.org/ https://dual.sphysics.org/gui/
4	It appears that USACE is underway to develop a GUI and a more "user friendly" open source CFD platform (https://www.erdcwerx.org/computational-fluid-dynamics-methods-for-water-resource-development/). Are there any other similar efforts underway?	Note, it's based around the SPH (smoothed particle hydrodynamics) meshless method. This method is getting more commonly adopted by modellers and commercial software like FLOW3D are including it as an option.
5	Will he create a simple model for clarification in this webinar?	There will be some examples of the model output, but the model creation will be covered in a future course
6	Can you please provide or comment some tips, guidance of some sort or share your experience in using cloud computing services (AWS, etc) to perform such simulations. thanks	I've used Azure virtual machines on 120 core machines, which has worked okay. You'll need to install the software on blank Linux OS when you turn the machine on, but parallel computing with high performance computing is one of the key things for larger CFD models (which requires access to faster machines).
7	Is there any user manual for Openfoam?	Yes, please refer to OpenFOAM https://doc.cfd.direct/openfoam/user-guide-v12/index
8	Hi Ray, have you made focused wave generation on the inlet boundary? If so, can you please discuss how. Thanks	Hi, I did a case on wave generation. It can be achieved, basically, you need to define the wave functions for the inlet
6	Using C codes you mean? Or have you coupled a utility to do that? I can do regular waves but am interested in focused waves	Yes, I did the regular waves, sorry not sure on focused waves.
7	can openform be coupled with groundwater model?	groundwater solvers exist (ie porousMultiphaseFoam), you'll need to investigate their equations and if it's appropriate for your application. A big part of CFD is the investigation and reading literature associated with the solver.
8	You can simulate vertical fluid motion in 1D and 2D by solving the Serre equations.	My understanding is the Serre equations are primarily for modelling waves, I dont think it would be suitable for modelling turbulence at structures and complex eddies
9	All good, thanks for that. Do you know if there is a two phase compressible solver for air similar to interFoam?	Yes, there is module for compressible solver. But I never used before.
10	Is SALOME a good option for dealing with "terrestial" geometries/bathymetries?	Yes, SALOME is a great tool.
11	I realise the topic here is primarily water, but could you use OF for wind pressures on buildings?	Yes, it is the another big module for openFOAM
12	Do you recommend the use of BlueCFD for Windows?	Sorry, never used before.
13	What is the accuracy of OpenFOAM compared to other commercial software, such as Flow-3D?	OpenFOAM is widely used in research as well, so accuracy is very high.
14	Hi is Open Foam a freeware? or have a trial period? Also, is open foam can only be utilized for condition "with structures" only?	Completely free.
15	Thank you, Ray. Does a formal benchmark exist for OpenFOAM?	No, but you can find in all kind of papers
16	Is there any tutorial on Open Foam for beginners?	Please refer to the OpenFOAM officical website for the tutorials
17	Can it be applied to coastal models?	Do you mean two-dimensional Shallow-water equations ?
18	if you become profficient in openfoam, could you also expect to also be able to easily adopt commercial packages such as Flow3D?	From my point, learning OpenFOAM is a journal to learn the fundamentials as well. With solid theortical background, it would be easier to master other packages.
19	Is the data inputs needed for OpenFoam are quite similar to the inputs with HECRAS?	Both need geometry and flow data (boundary condition), but the format is different
20	Hi Kyle, thanks for the discussion, have the scour models been built using a dynamic mesh you said?	
21	What criterias are commonly used for the convergence of hydraulic (weir) structures? I was asked once about residual erros but I had no sensitiveness about acceptable errors. Could you recommend any publications about it?	The criteria would be the mesh convergance test. So the mesh is very important for these structures with complex geometries.
22	Used in the definition of the mesh, such as GCI... Are you going to cover it in the course? After defining the proper mesh, and running the model, are any advice for pressure residual erros or other parameters?	
23	Can a 1D network be integrated with an OpenFOAM model?	Not easily. It can do 2D, but not 1D like you can with 1D/2D TUFLOW model for example, or culvert in 1D/2D HEC-RAS. OpenFOAM is better for object scale problems, while 1D/2D is better for landscape scale
24	would OpenFoam be useful for modeling intense rain falling on engineered structures such as landfill covers?	The input of rainfall would be challenging in OpenFOAM
25	SPH is on GPU!	SPH is very diffusive and generally first-order accurate, not recommended. May be good option for simple gometry. LBM on GPU might be a better option.