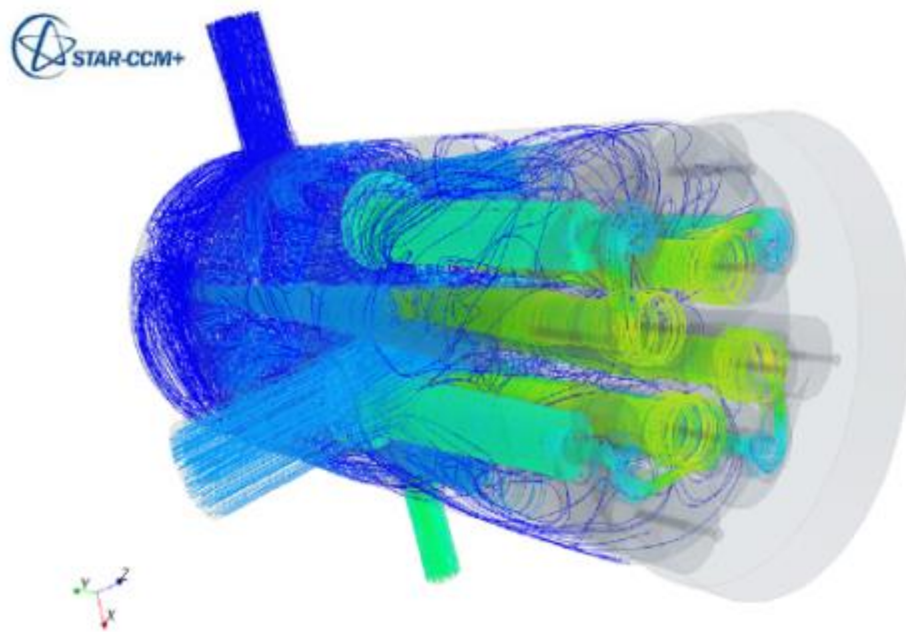
Modelling of multiphase oil/water cyclonic separator



Abdullah Rehman, *Caltec*
Dr Steve Howell, *Abercus*

STAR Global Conference, Vienna
March 2014

CALTEC
www.caltec.com

www.abercus.com

Introductions

Caltec synopsis

- Caltec is a world leader in Surface Jet Pump (SJP) and compact separation systems for upstream oil and gas production enhancement. It offers application identification, design optimisation, and the provision of solutions based upon proprietary SJP and compact separation technologies. Caltec has extensive track record in adding value to major IOC and NOC clients worldwide.

Abercus synopsis

- Abercus is a consultancy specialising in advanced engineering simulation, specifically computational fluid dynamics (CFD) and finite element analysis (FEA), within the energy industries.

Caltec's Wx technology

- Wx – Water eXtraction is Caltec's patented compact in-line cyclonic device for bulk oil/water separation for deployment onshore/offshore/subsea
- High 'g' forces generated within patented 'I-SEP'
- Recovers water from an oil-water mixture
- Gas tolerant with no moving parts
- Oil in water levels better than 1000ppm
- Debottlenecking existing separator
- 65% or more of produced water recovered from stream

Wx-4 Tested Onshore/Offshore Norway

CALTEC
www.caltec.com

abercus
www.abercus.com

Aim of CFD Model

Aim of the CFD model was to improve the Wx design by:

- Improving the **amount of water** that can be recovered
- Improving the **oil in water content** of the produced water
- **Decreasing the pressure drop** across the device
- **Increasing flow handling capacity**

Wx development using CFD

In order to get a deeper understanding of the flow field and distributions within the Wx unit Caltec engaged Abercus to provide specialist input

The study was undertaken in two parts:

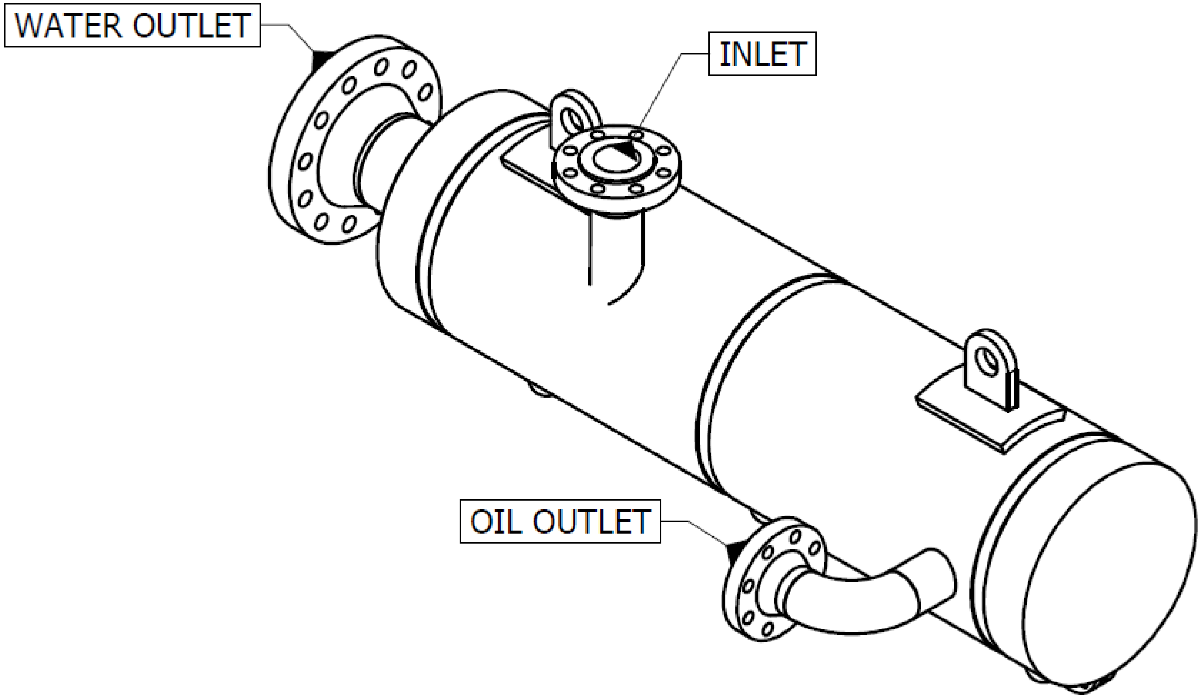
1. Build a CFD model and compare results with experimental data
2. Use the verified CFD model to predict Wx performance for modified designs

Methodology:

- CFD model was based around the Wx-4 separator, currently being proposed for field trials in Malaysia

- **Used real data** obtained from Onshore/Offshore trials in Norway, to validate the model

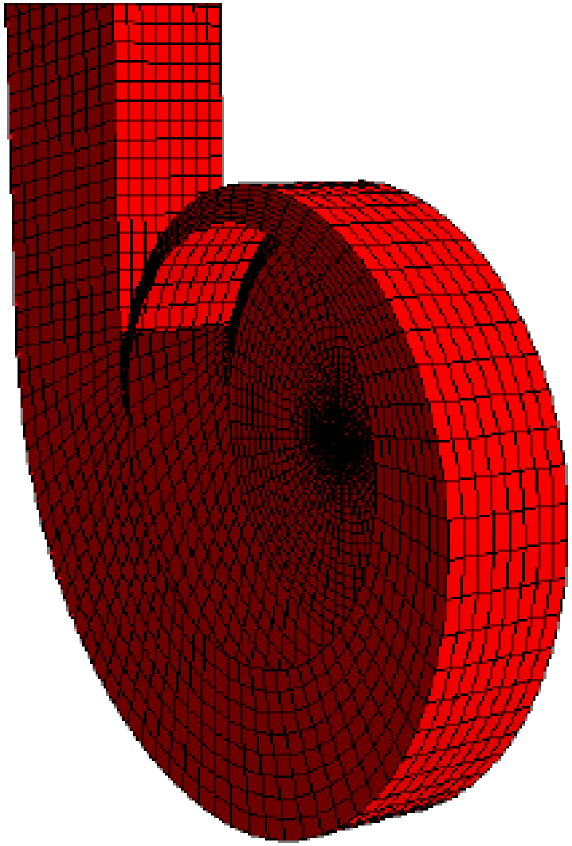
Cases considered



CASE	Inlet Flow Rates (m^3/hr)			Water Recovery (%)	Pressure drop relative to Oil Outlet (bar)		Oil in Water (ppm)
	Water	Oil	Gas		Inlet	Water Outlet	
A	20	4.9	-	60.2	2.2	1.1	590
B	22.5	2.5	-	64.4	2.5	1.15	977
C	20.5	5.0	10.9	57.8	3.7	1.1	804

Approach

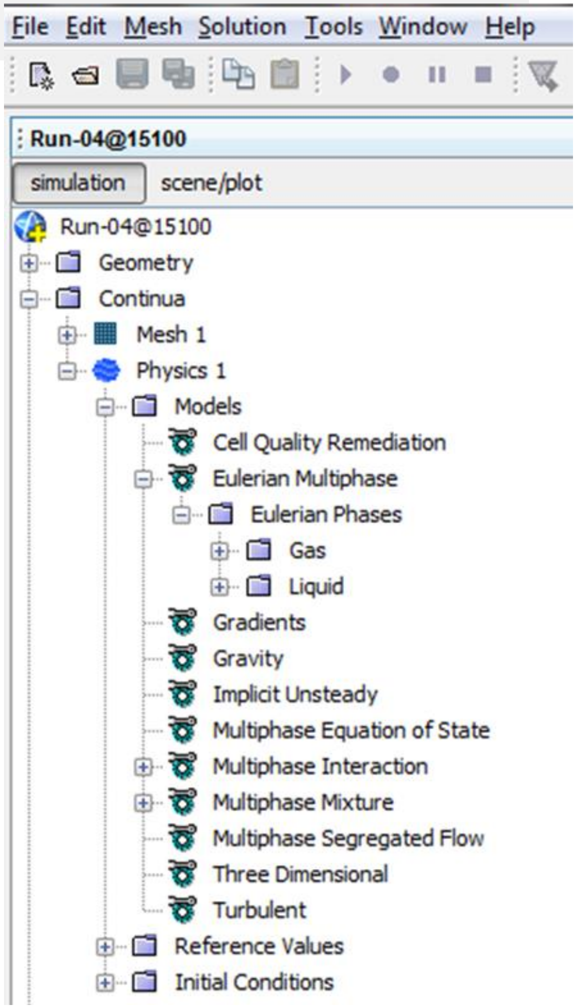
- The STAR-CCM+ v7.02 platform used throughout the study
- Manually generated hex-mesh vs auto-generated mesh
  - Automated trimmer mesh = 2,160,000 cells,
  - Structured hexahedral mesh count = 950,000 cells
- For larger more complex cyclones, this could be the difference between running on a desktop workstation or a requirement for cluster computing
- CD-adapco implemented the new interactive meshing feature 'Directed Meshing' in its very next release v7.04



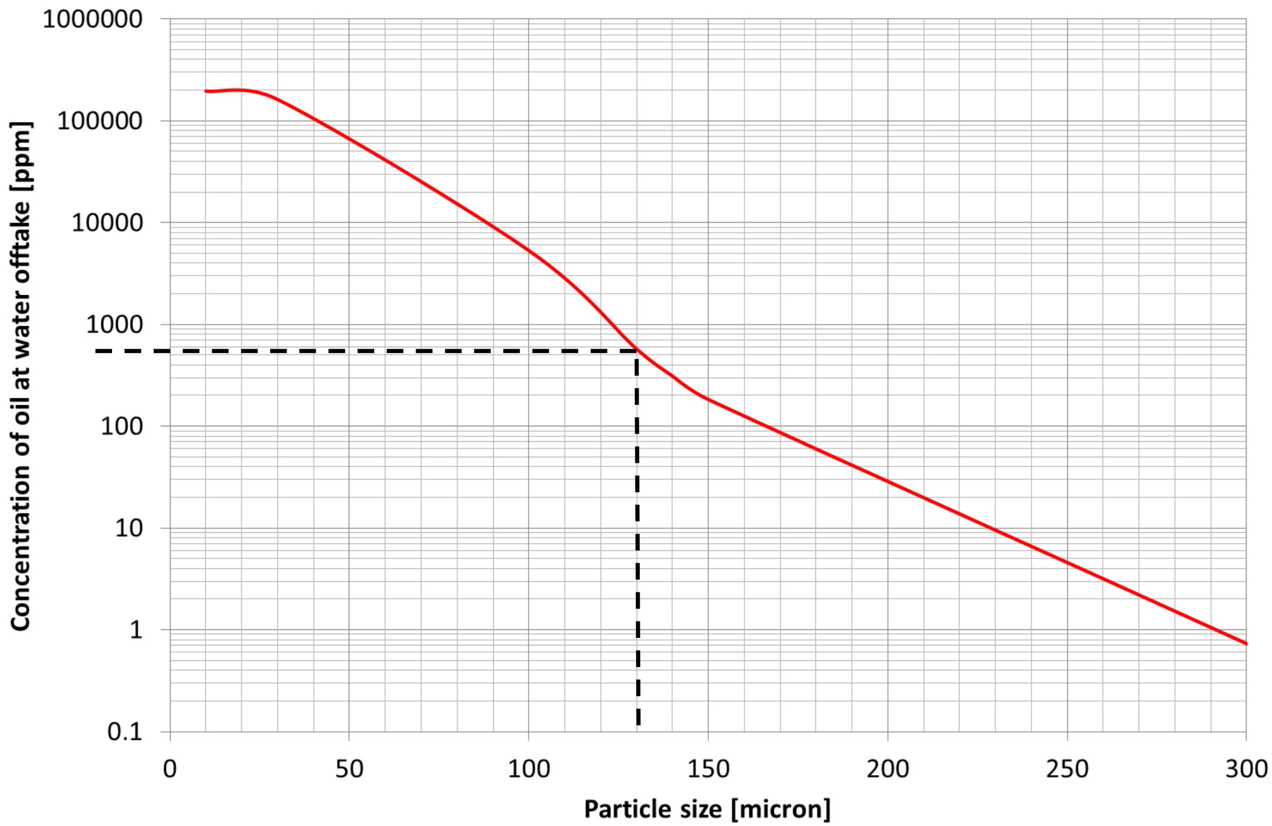
Physics

- Steady state
- Eulerian Multiphase model (EMP)
- Multiphase segregated flow method
- Isotropic standard k-ε model used –

  anisotropic Reynolds Stress Turbulence Model (RSTM) was not yet implemented in STAR-CCM+ v7.02
- Water density and viscosity: 1035 kg/m3 & 0.001 Pa.s
- Oil density and viscosity: 797 kg/m³ & 0.0015 Pa.s
- Schiller-Naumann drag correlation method used
- Water specified as continuous and oil as dispersed phase
- Wall surfaces assumed to be smooth
- Oil droplets size unknown - sensitivities undertaken

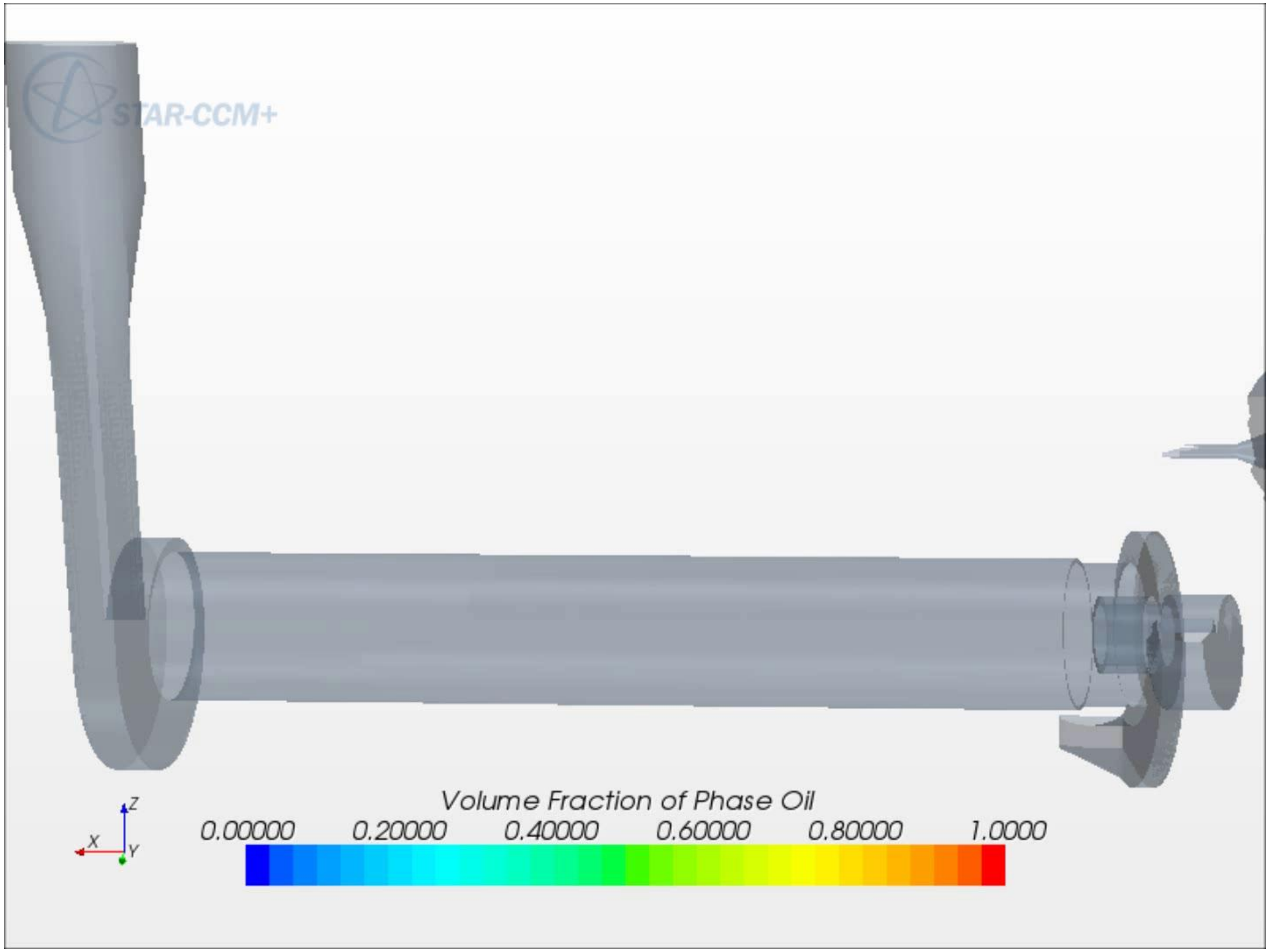


CFD Inlet Droplet Sensitivity Analysis



An oil droplet diameter of 130 microns corresponds to the water quality of ~590 ppm, as observed in the experiments

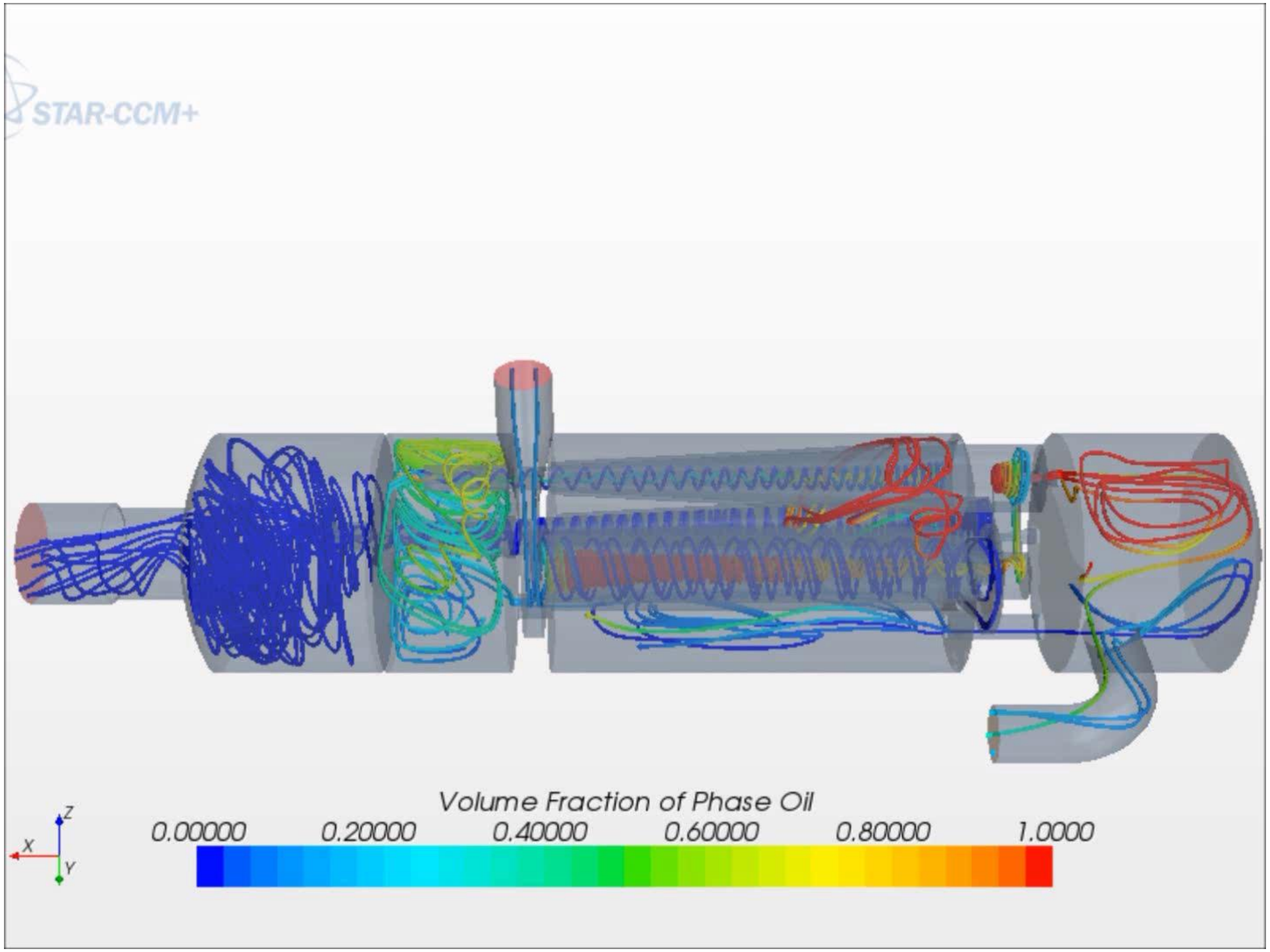
I-SEP Streamlines



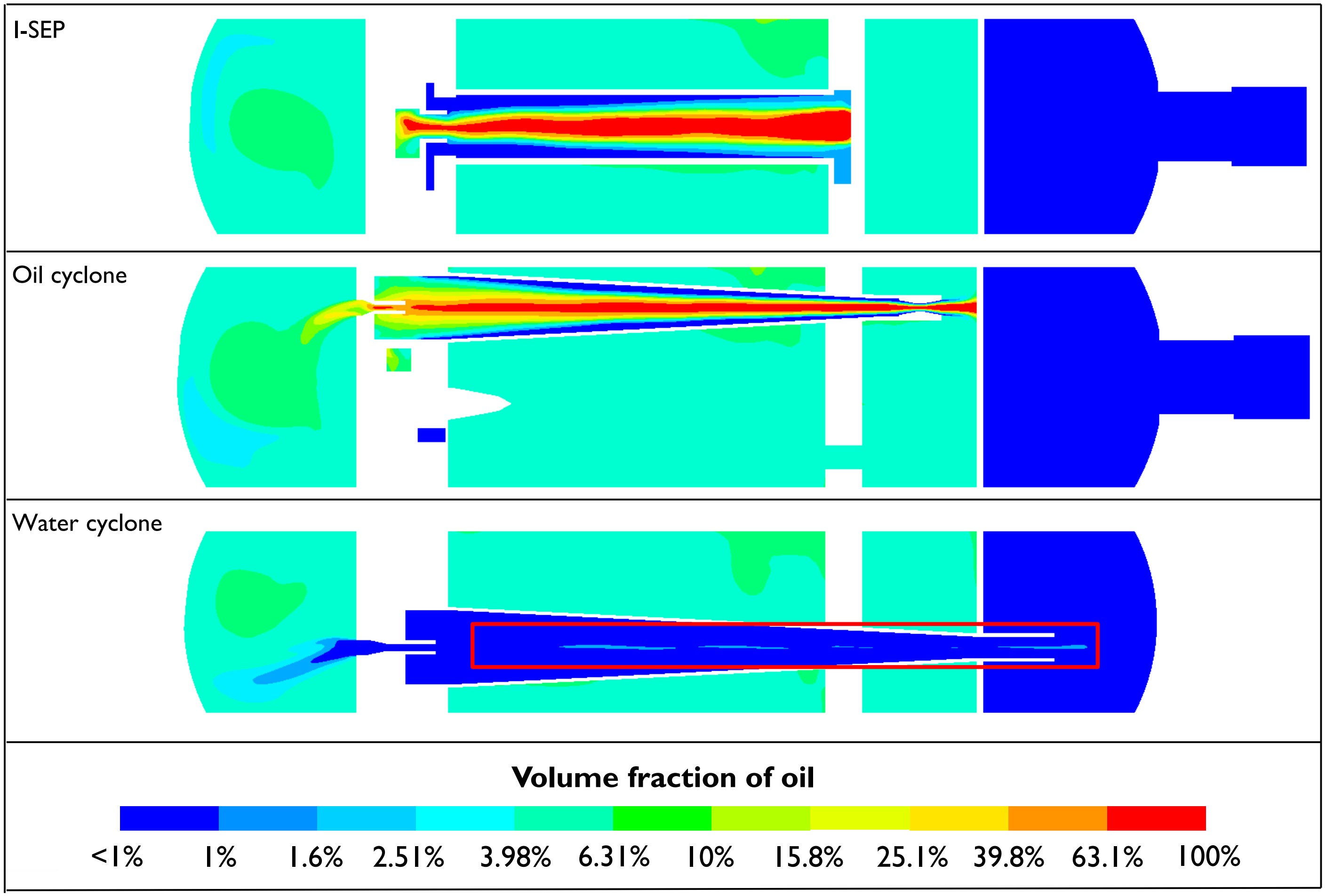
CALTEC
www.caltec.com

abercus
www.abercus.com

Wx4 Streamlines



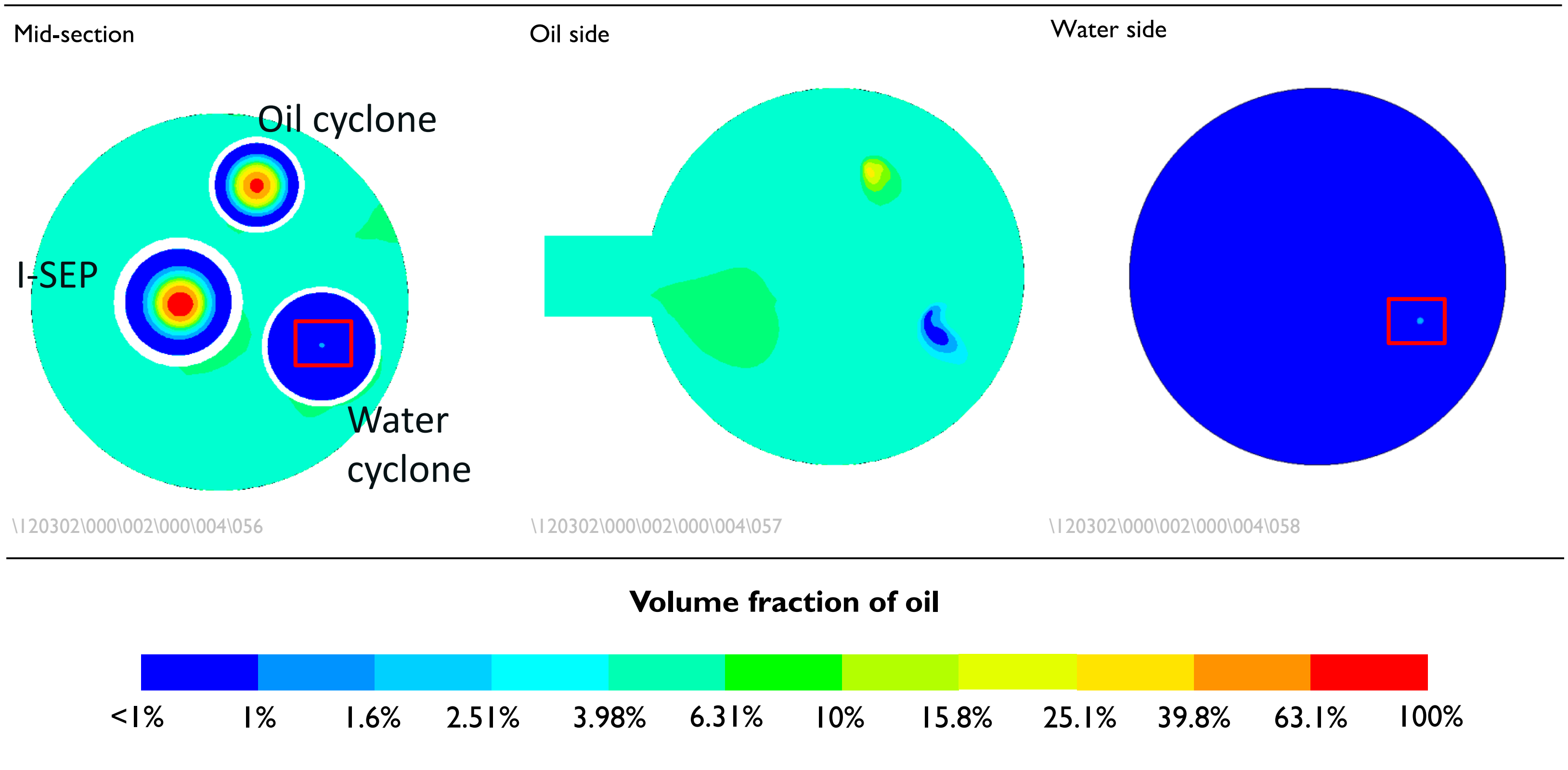
Typical contours with standard k-ε turbulence model



CALTEC
www.caltec.com

abercus
www.abercus.com

Typical contours with standard k-ε turbulence model



CALTEC
www.caltec.com

www.abercus.com

CFD Predictions

CASE	Experimental Pressure drop relative to Oil Outlet (bar)		***CFD Predicted Pressure drop relative to Oil Outlet (bar)***	
	Inlet	**Water Outlet**	**Inlet**	**Water Offtake**
A	2.2	1.1	1.2	0.3
B	2.5	1.15	3.3	2.2
C	3.7	1.1	1	5.1

- The pressure drop across the device were incorrectly predicted for all three cases
- It is thought that this is due to the use of the standard k-ε turbulence model
- These predictions were shared with CD-adapco and, as a direct consequence of this work, the Reynolds Stress Turbulence Model was implemented for use with the Eulerian multiphase framework (released in v8.06).

CFD Predictions

- For case C, which was three-phase (water, oil and gas) the oil carryover was significantly over-predicted using CFD
- The gas core within each cyclone displaces the oil outwards, which is manifested as a deterioration in the separator performance
- Contrary to this, it is observed experimentally that separation with gas is improved, due to adhesion of individual oil droplets to gas bubbles – this mechanism is called **gas flotation**
- The gas flotation mechanism is complex and, to our knowledge, remains beyond the current state-of-the-art for CFD

Best practice CFD

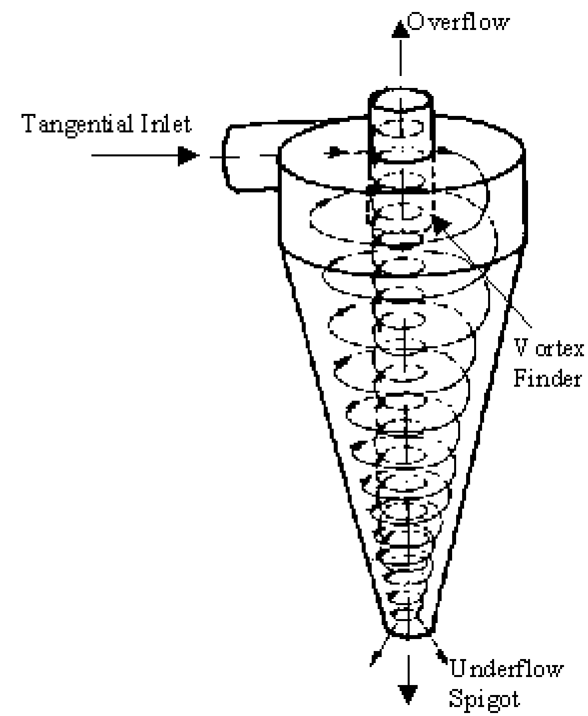
- Abercus is a member of NAFEMS and ERCOFTAC, the two leading organisations for validation cases for CFD and FEA

- Abercus has previous experience of modelling cyclonic flows in-line with ERCOFTAC best practice guidance

Best practice CFD

- ERCOFTAC application case AC3-03 considers flow within a cono-cylindrical cyclone.

- There are three main points to draw from this case:
  - Mesh: in order to minimise unwanted numerical diffusion effects, the best type of mesh is a hexahedral type with the mesh elements aligned with the circumference of the cyclone
  - Turbulence model: an anisotropic turbulence model is required to correctly capture the free to forced vortex transition that occurs in cyclonic flows. Standard k-ε models and other models based on assumptions of isotropic turbulence are not suitable as they tend to over predict the turbulent viscosity and exaggerate the forced vortex
  - Boundary layer resolution: wall resolution is not critical as the turbulence is generated in the main flow



Conclusion

- For cyclone applications it is important to invest effort in the mesh build to construct an efficient hexahedral mesh with cells aligned to the flow around the circumference of each cyclone
- CFD validation against empirical data is essential
- The CFD predictions for oil carryover at the water offtake are sensitive to the diameter of the oil droplets
- The pressure drop across the device was under predicted using CFD
- It is thought that this is due to the use of the standard k-ε turbulence model

Conclusion

- Caltec and Abercus were able to share the results of this study with CD-adapco and, as a direct consequence, the Reynolds Stress Turbulence Model was implemented for use with the Eulerian multiphase framework (released in v8.06)

- It is impressive how CD-adapco responded to implementing this improvement

CD-adapco is a truly adaptive and client-driven software company

Future areas for investigation

- Break-up and coalescence modelling
  - The oil carryover is sensitive to the diameter specified at the inlet, which is assumed to remain constant throughout the device. In reality this will vary and will depend upon break-up and coalescence within the device. Modelling this will remove the sensitivity to the user-defined droplet diameter at the inlet
  - The droplet size is difficult to measure experimentally, so CFD predictions could provide useful additional information
- Gas flotation
  - This is where oil droplets adhere to gas bubbles within the cyclone, which is manifested as a significant improvement in terms of separation efficiency. To our knowledge, this is beyond the current state-of-the-art for CFD