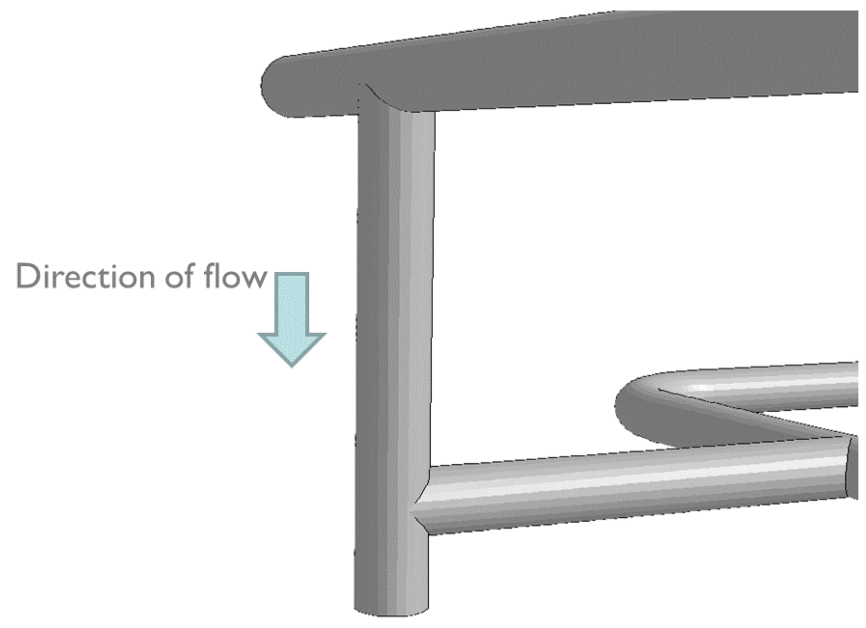
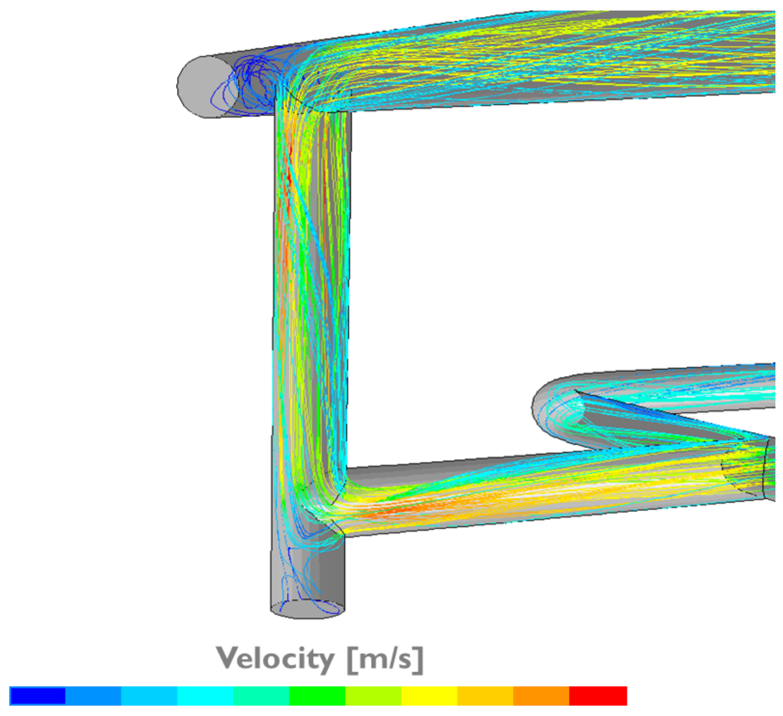
On the use of computational fluid dynamics for predicting issues relating to produced sand

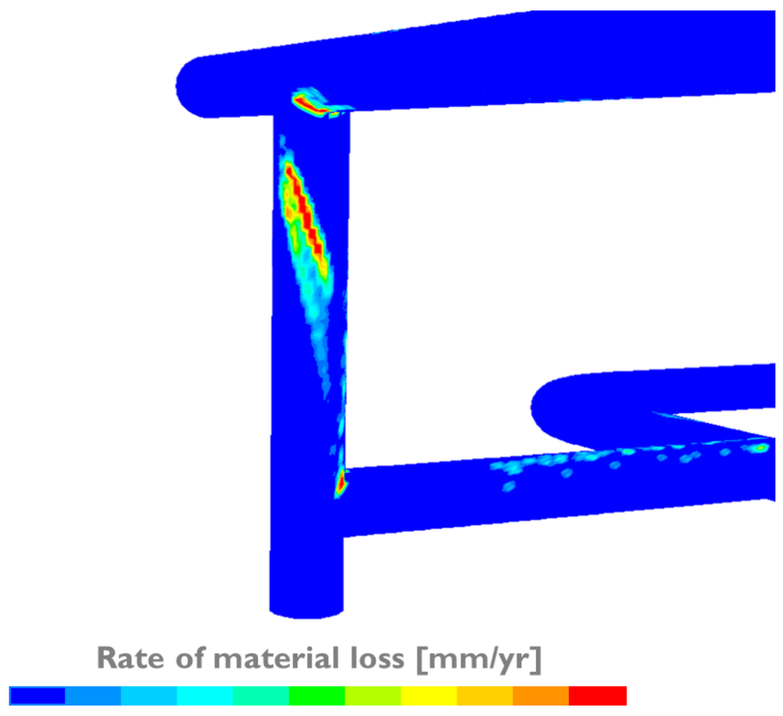
Sand production is unwanted. Produced sand can cause equipment to erode, can cause blockages within the production system and can cause production equipment to function less efficiently. These effects and their impact can be considered using computational fluid dynamics (CFD), an advanced engineering simulation approach which can provide a better understanding of the underlying physics and, therefore, ways of mitigating problems associated with produced sand.

Pipework erosion

In this example CFD is used to simulate the flow within the pipe network. The incoming flow is seeded with sand particles and their associated trajectory is calculated. The local erosion effects are then calculated using a correlation (for example, by Salama, Oka, University of Tulsa or DNV) which relates the local velocity and angle of impact of the sand particles on the internal surfaces of the pipework to the rate of material loss.
It is important to recognise that the rate of material loss is likely to vary, both in magnitude and the location of any erosion hotspots, during the operational life. This may be due to changes in the flow behaviour (and, therefore, the impact velocity and angle of sand particles carried by the flow) due to variations in the prevalent flow conditions and/or sand loading. The flow behaviour may also change due to cumulative erosion which can significantly affect the geometry of the material surfaces bounding the flow — see for example, the erosion hotspots predicted at the inside of each bend in the contour plot for the rate of material loss opposite.
Material loss may also depend upon the flow regime. Material loss may, for example, be lower in some scenarios where sand is carried by wet gas because, in the event of an annular flow regime, the liquids can form a protective layer across the material surfaces bounding the flow which cannot form in the corresponding dry gas case.
CFD offers an approach which can capture these effects — variations in flow conditions and/or sand loading, cumulative erosion and associated changes in geometry/flow behaviour, and multiphase flow regimes — to provide a better understanding of the important parameters impacting upon erosion.

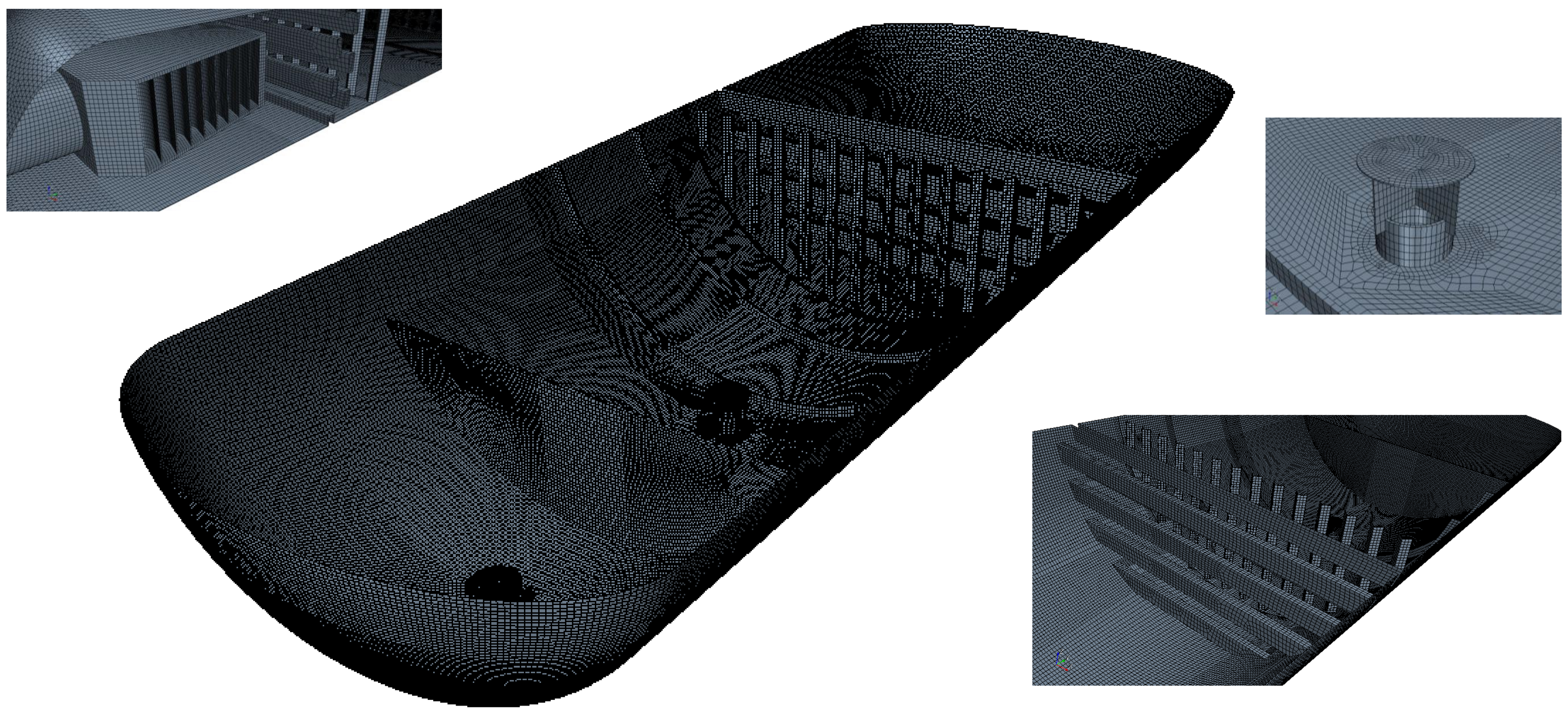


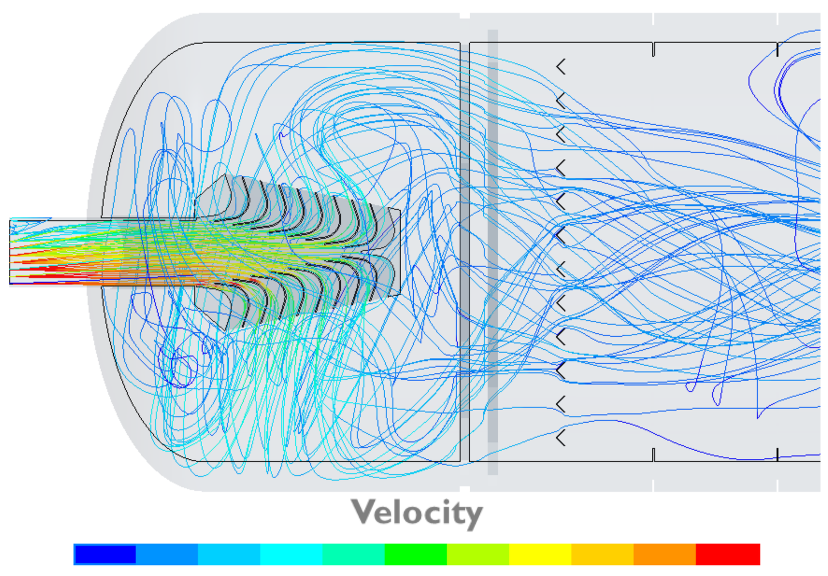


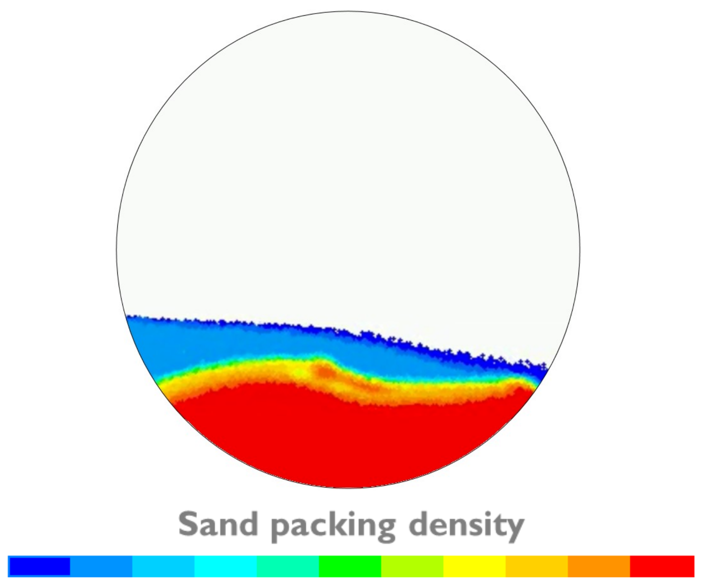


Sand accumulation within separator vessels and the associated impact upon efficiency

In this example CFD is used to predict the multiphase flow within the separator. A mixture of gas, oil and water enters the vessel at the inlet vane device to assess the efficiency of the vessel in terms of liquid carry-over and gas carry-under for the case without sand.
The incoming flow is subsequently seeded with sand particles which are allowed to accumulate within the vessel. This reduces the cross-sectional area for flow within the vessel, causing the flow to accelerate, which leads to a reduced residence time within the vessel and an associated reduction in the efficiency of the vessel.
The impact of sand accumulation upon the efficiency of the vessel can be considered across a range of different flow scenarios that might occur during the operational lifetime of the vessel.
The CFD approach can subsequently be used to simulate the fluidisation of the accumulated sand within the vessel due to the action of de-sanding jets as part of a sand-washing system, for example, to investigate the effectiveness of the system and provide understanding that could be used to improve the design of the system.







abercus