A dynamic approach to flow assurance modelling

Chris Smith, Production Assurance Manager at Xodus Group

Aug 2013

Modelling the hydraulic and thermal relationships between fluids and gasses in a production environment can be an inexact science. As systems become more complicated, determining the impact of these interactions within oil and gas pipelines and infrastructure becomes more challenging.

Traditional pipeline modelling systems have been around for almost 20 years, and international energy consultancy Xodus Group has recently undertaken a number of projects which have presented flow assurance engineering challenges that could not adequately be resolved using standard multiphase transient simulator modelling tools (such as OLGA).

In these instances, Computational Fluid Dynamics (CFD) modelling has been used in addition to standard techniques to achieve advanced flow assurance solutions. Some of these projects would not have been viable without the use of CFD, and it has also enabled pioneering subsea projects to take place.

The traditional approach

Current tools most commonly used allow large, complex, full scale network modelling, with the ability to include wellbore, reservoir and process equipment operations. An entire subsea pipeline network can be modelled, even with numerous wells and multiple lines.

This can be completed within reasonable timeframes. For example, a large model could run in a day, giving significant computational output. Due to the fast run time, software can play out the full 20 years of production of an oil well and calculate the temperature and hydraulic effects over time to aid optimum system design.

These programmes also enable full multiphase pressure, volume, temperature (PVT) capability, including component tracking through the network to calculate changing compositions. Transient operations are possible with some software which provide data on the change in hydraulic relationship with time as well as flow conditions. Therefore, many parameters can be analysed from a single run.

However, there are limitations with this type of software. Detailed analysis of specific points in the system is not possible due to section resolution required to allow modelling of large scale networks. It can be focused on two or five meter length sections, but to examine the first ten centimeters downstream of a mixing point, for example, fidelity is lost.

It is not possible to model complex geometries for specific equipment such as valve internals, spools, manifolds, and process equipment. Traditional software will only model a round pipe so localised areas of concern and detail may be missed such as flow distributions, cold spots caused by flow paths and erosion risks. Certain flow paths cause this, for example where there is a tight elbow in a valve there can be an erosion risk at the edge of the elbow because of the significant flow energy on that area.

Additional to these hydraulic challenges, thermal models are also limited and not always sufficient for complex pipe and equipment arrangements, or certain transient events. Thermal models also work on a standard cylindrical shape, so calculating heat loss in a pipe that is part submerged is impossible as it is not universally cylindrical. Also, natural convection (both internal and external) and radiation are not accounted for.

Using CFD in flow assurance

Analysis of low material temperatures caused by rapid gas expansion (JT cooling) is possible using CFD. This includes cold spots and thermal gradients, which can go beyond the design limits of the material, and traditional pipeline tools don't give the distinction required. Situations can be evaluated including choke valves and downstream pipework at start-up, and process equipment and pipework during blowdown.

CFD also offers detailed thermal modelling of part buried, backfilled, rock dump and mattressed pipelines; complex pipelines; and subsea heat exchangers or cooling spools. Analysis of solids erosion is also possible at flow path deviations to identify erosion hot spots.

Flow distribution at flow path splits such as manifolds, metering skids and finger slug-catchers can be modelled. The latter is the most important because without equal distribution there is not benefit in having finger slug catchers in place. Other systems would just equally split the flow path without deriving the actual likely flow split.

With rapid valve closures, CFD enables analysis of peak pressure pulses close to valves. This water hammer effect causes very localised high pressure waves which can't be analysed in other systems as they smooth out all the hydraulics.

Combining CFD with full network modelling tools

Although CFD modelling is powerful and suitable for precise evaluation of specific areas, it is not viable for modelling large scale networks or life of field analysis due to its computational intensity.

In some instances, it is useful to combine the analysis carried out in CFD with pipeline modelling tools to allow full network analysis using CFD derived results. It allows for the accuracy of CFD with the speed of pipeline modelling ▸

The combined effort can result in accurate overall heat transfer coefficient (U value) for complex pipeline systems or partial burial to determine the heat loss coefficient. Accurate cool down rate can also be calculated by tuning heat loss parameters, which is especially vital during shutdowns.

Application of maximum erosion velocities for solids laden fluids can be done to determine what gas velocities will cause an erosion risk. CFD also allows for detailed analysis of phase distribution in through pipe to demonstrate dispersal in gas water mixes. This combined approach bears many benefits and has been utilised in a number of Xodus projects.

Case study – thermal modelling

In a subsea oil development in the Norwegian North Sea, the engineering consultancy worked on a tie-back to a FPSO project with multiple well pads. The specific challenges for thermal modelling in OLGA included the subsea cooling coils or spools, and the pipeline was partially embedded in soil.

It also involved bundles of pipelines from wells to risers, each consisting of two production lines and one water injection line. The multi-dimensional heat transfer mechanisms in the bundled pipeline system were complex and not adequately captured using standard OLGA modelling. Another challenge was heat loss during shut-down, and survival time due to wax and hydrate concerns which had to be factored in.

CFD software Star-CCM+ was used to model a short section of the pipeline bundle and derive the U value. This considered conduction; the K values for aged and un-aged insulations and effect of spacers, as well as convection; the interactions between bundle pipelines and the effect of film coefficient. The U value changed through field life and along the length of the bundle, and the cool-down response was analysed with CFD. OLGA parameters; U value, heat capacity, and effective annulus current speed were tuned to match the response of CFD analysis. Those combined parameters were analysed in CFD and then put into the pipeline software to allow modelling of the whole system. Heat loss in the cooling coil was analysed in STAR-CCM+ and the effective U value was calculated and applied in the OLGA model. The effect of seawater current speed was minimal as natural convection caused a chimney effect and drew cool seawater up through the coil.

Pipeline modelling tools don't accommodate natural convection which would mean inaccurate results in that sort of design. CFD allowed the thermal losses from this complex pipeline system to be modelled with high accuracy.

The OLGA models were tuned to match the heat transfer coefficient supported by detailed CFD analysis, rather than from standard published figures. This accounted for all thermal interaction mechanisms and accurately simulated the cool down response of the overall bundle system. Life of field and transient analysis was then performed on this basis. In this case, due to the complex thermal system and the fluid temperature being critical to design, without use of CFD it may have been unviable to produce this well. The lack of accuracy would have meant engineering with large safety margins. In doing so, it would have proved impossible to solve the problems at hand. Using this high fidelity analysis removed some of the conservatisms and enabled a tightly engineered design.

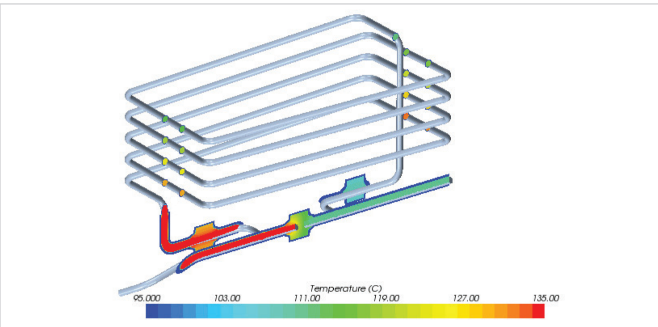
Case study – mixer design

This case study focuses on the design of a simultaneous water and gas injection (SWAG) system in a subsea oil field where production wells were tied back to an FPSO. This was a particularly unusual system and a first for subsea.

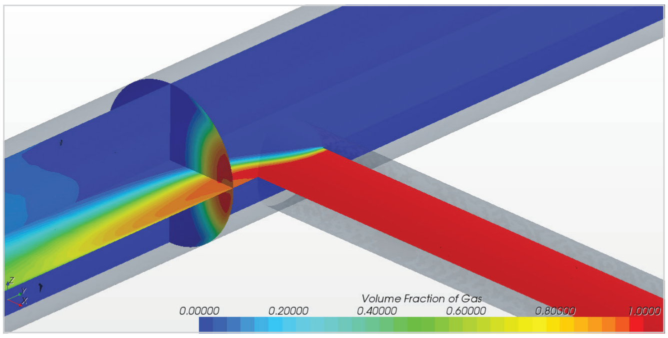
The water and gas injection fluids are delivered to a subsea injection well in separate flow lines and combined immediately upstream of the well prior to injection. The complexity of this design demanded that the gas is fully dispersed within the water stream such that no separation occurred in the wellbore, which may have lead to elevated hydrate risk. The injection system, including the well, was modelled using OLGA, however detailed analysis was required at the commingling of the fluids, where STAR-CCM+ was used to analyse gas dispersion achieved from a range of available mixing technologies.

OLGA analysis of the system showed combined fluid would operate in the slug regime at expected process conditions which would result in very poor injection and could damage the well. The Group was tasked with designing a mixing device to overcome this problem.

Initially a simple T-piece mixing solution was explored, but with poor results as the gas and water separated quickly. Peak mixing occurred at approx 6m downstream of the mixer with stratification appearing 40m downstream of the mixing point ▸

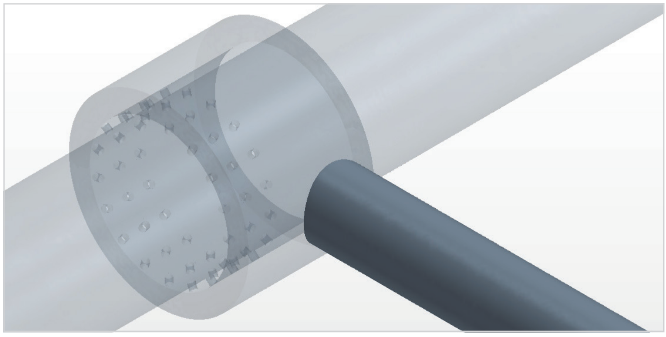


Subsea cooling coil pipework.



Gas flow into a continuous water stream through a simple tee piece.

A collar system was then considered, where gas flows around the metal collar and is then injected in multiple points around the circumference of the water pipe. This proved more effective with improved mixing which was enhanced further when the injection points were situated only on the bottom of the pipe.



Water and gas stream mixing collar.

The outcomes of this case were that some level of stratification occurs 40m downstream of the mixer irrespective of what design was in place. The sleeve mixer design was significantly more effective than the simple T-piece join which the client had initially pursued.

Locating the gas injection holes only at the bottom of sleeve increased the mixing significantly. Bubble diameter was the most significant factor in successful mixing and reducing the velocity has little impact on mixing uniformity.

These modelling outcomes allowed the functional design to be specified ensuring that mixing occurred as close as possible to the injection point. Small holes were then employed to decrease bubble diameter where the mixing device had holes on bottom of sleeve only.

Conclusion

Both case studies proved that CFD can be combined with traditional pipeline modelling software to provide the best of both applications in terms of detail and speed for accurate flow assurance modelling.

Without employing this technique, some projects may not be as successful, or may not be able to commence at all. While computational powers may limit the ability to use CFD across an entire system, there is no doubt this technology will play an increasingly important role in flow assurance.

Used in addition to standard techniques, CFD has established its role in achieving advanced flow assurance solutions. Some of the projects where this has been utilised would not have been viable otherwise, and it has also enabled leading engineering solutions to be applied subsea for the first time ■