CFD Modeling of Corrugated Flexible Pipe

The flexible metal pipe has been used in smaller diameters for more than 30 years for all kind of cryogenic Liquid Natural Gas (LNG) transfer applications (Refs. [1,3]). Today these LNG loading systems have evolved into a complex system, which have to respect increasingly stringent rules and standards while continuing to maintain high levels of safety and availability. One of the main problems in these systems is to predict internal turbulent flow behavior, hence the associated pressure drop in the corrugated configuration of flexible pipes. Metallic corrugated pipes are well known structures, which can withstand tensile and internal pressure loads, as well as perform better from a fatigue and heat transfer standpoints. However, series of corrugations can induce complex and undesirable flow behavior in the pipes. The wavy configuration of the corrugations promotes turbulence and therefore improves heat transfer. For both design and operational standpoint, the LNG transfer from ship to ship is a relatively new application of this well known technology (Figure 1(a)). The basic design of LNG transfer pipe is illustrated in Figure 1(b).

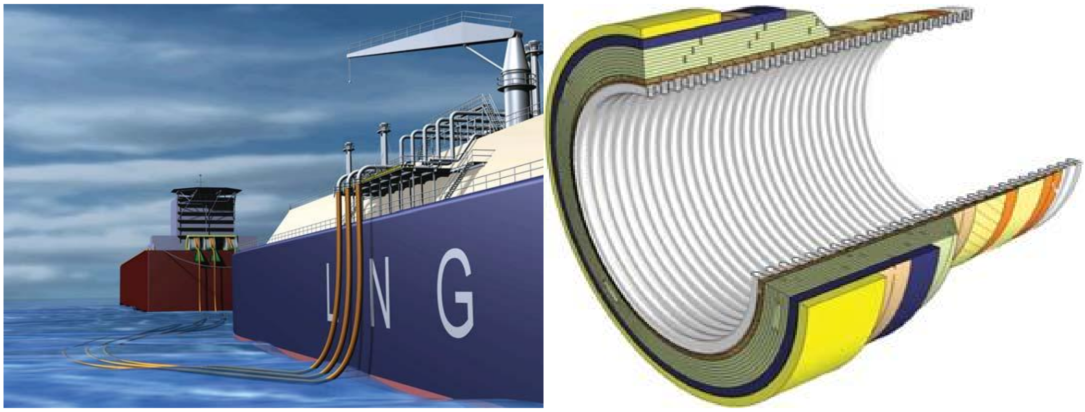

Figure 1: LNG transfer applications (a) Offshore LNG transfer system
(b) Common design for LNG flexible pipe (Ref. [2])

The objective of this case study is to present CFD modeling of fully developed turbulent flow through a flexible corrugated pipe and to investigate the pressure drop reduction by introducing liner materials. The reduction in cost and complexity of developing a robust cryogenic liner or corrugation filler, plus eventual certifications, would be significant and needs to be worth the improvement (decrease) in pressure drop. To estimate the variation of the pressure in the corrugations, we do not model the phase change and the bubbles cavitation but accurately evaluate the pressure drop along the pipe. The pressure drop estimation can be useful to deduce the upstream pressure which can be imposed to stay everywhere downstream above the phase change pressure. This work also aims to establish a framework to be used in large scale numerical simulations of the offshore transfer of cryogenic fluids. A 3-D CFD approach is considered more appropriate than 2-D axisymmetric one, since the wavy corrugation profiles lead to a great deal of internal turbulent structures for a high Reynolds number over Re > 10 million.

Three geometries of the bellows' (corrugation) depth are considered to determine the potential value of a cryogenic liner, corrugation filler or geometric variations for the 16" pipe. We consider the length of 3D flow domain with L = 6D matching earlier work on the direct numerical simulations of fully developed pipe flow. For the parametric design study, we select three configurations with varying depths A* (A/ID): A*=0.06047 (base), A*=0.01583 (liner1), and A*=0.00798 (liner2); where A denotes the depth and ID is inner diameter of the pipe.

The turbulence level is typically high due to the corrugations and turbulence modeling is critical to get the accurate predictions. To model the steady effects of the turbulence on the mean flow field, we employ the Spalart-Allmaras Reynolds Averaged Navier Stokes (RANS) model. For unsteady simulations, we employ Delayed Detached Eddy Simulation (DDES), a hybrid RANS model with Large Eddy Simulation (LES). In the LES based on dynamic subgrid scale estimation, an attempt is made to capture the large scale unsteady motions which carry the bulk of the mass and momentum in a flow, but the near wall turbulence behavior is treated with a wall function. In the DDES model, we resolve the large eddies that have the biggest effect on the wall shear stress and use the RANS equations to describe the flow near the wall. This was done not only to economize on mesh size, but also because most pipes have relatively rough walls. Wall functions reduce mesh size by providing an integrated relationship between the wall and the logarithmic region of the boundary layer. To simulate the large length of corrugated pipe with fully developed flow, periodic conditions are applied between the outlet (exit) and inlet (entrance) of the domain.

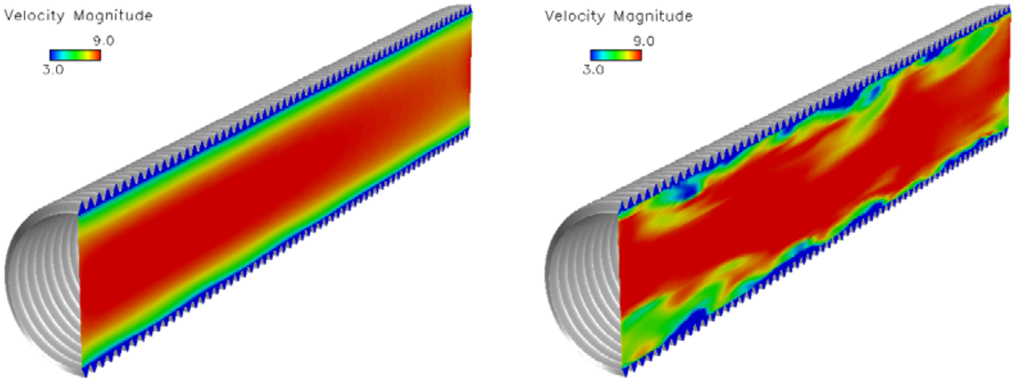

Figure 2: Streamwise variation of velocity magnitude contours in the corrugated pipe at flow rate Q=3333 m3/h: (a) RANS model (b) Delayed-DES model

Figure 2(a) shows the contours of velocity magnitude using the RANS model at the Reynolds number of Re=9.38E6 for the base model of corrugated pipe. The fully developed and time averaged steady flow behavior can be observed from the figure. As expected from the RANS model, there are no physical unsteady motions in the velocity field. Figures 2(b) shows the contours of streamwise velocity at the cross section of the corrugated pipe with the DDES model. The 3D turbulence structures and unsteadiness in the flow can clearly be inferred in the image.

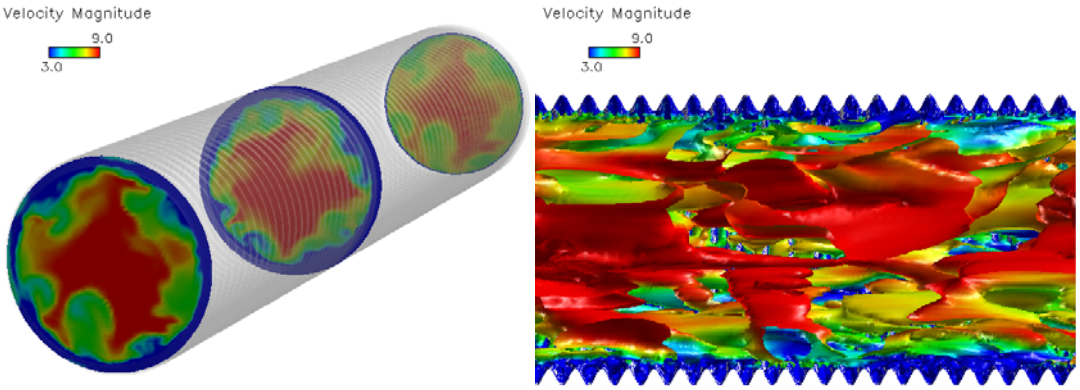

Figure 3: (a) Instantaneous velocity magnitude contours at the cross-sectional planes for flow rate Q=3333 m3/h (Re = 9.38E6) (b) Iso-surface of vorticity variable (Q-criterion) colored by velocity magnitude

Figure 3(a) shows the contours of cross-stream velocity magnitude at the three cross section planes of the corrugated pipe. Significant circumferential variations in the velocity magnitude can be seen in the figure. These local variations are coupled with vorticity, which is defined as the rotation of the velocity field. Figure 3(b) shows complex 3D turbulent structures of low-speed streaks and in-plane streamwise vortices.

Figure 4(a) shows the variation of coefficient of friction for the range of Reynolds number for the three configurations of varying depths and the smooth pipe. The friction factor was determined by evaluating the pressure gradient along the pipe from the integrated pressure values. For the baseline case, the friction coefficient is consistently larger than the liner1 (1/4 depth of base) & liner2 (1/8 depth of base) geometries. Notably, the wall shear stress of the liner2 model is converging towards the stress values corresponding to the smooth pipe. This implies that, by introducing liner materials, the coefficient of friction can be reduced by 80% with respect to the deeper metallic hose configuration. Due to complex flow behavior and recirculation in the base & liner1 models, the friction factor changes significantly with the Reynolds numbers. Figure 4(a) also presents the roughness theory predictions given by the lines. For the smooth pipe, the CFD results and the theory have an excellent match. However, for the corrugated shapes the roughness theory seems to differ up to 24%.

Figure 4(b) shows a summary of the friction factor computed based on the pressure drop for the steady RANS with the DDES on the same meshes. A reasonable consistency in the predictions of integrated pressure drop can be seen in the figure. By tuning the grid distributions, an improved match between the RANS and DDES may be obtained. For the base and liner 1 geometry at Re~10M, an inflectional behavior in the pressure drop and wall shear stress have been observed in the RANS and DDES results. This dip in the frictional drag may be attributed to the sudden shift in the point of separation for the base and liner 1 geometries. In this range, the laminar viscous sub-layer portion of boundary layer may become unstable and undergoes transition to turbulence. For values of Re >10M, the separation point slowly moves upstream as the Reynolds number is increased, resulting in an increase of the friction factor.

For the liner 2 and smooth pipe, the geometry is streamlined and the point of separation and the transition of boundary layer remain somewhat unchanged.

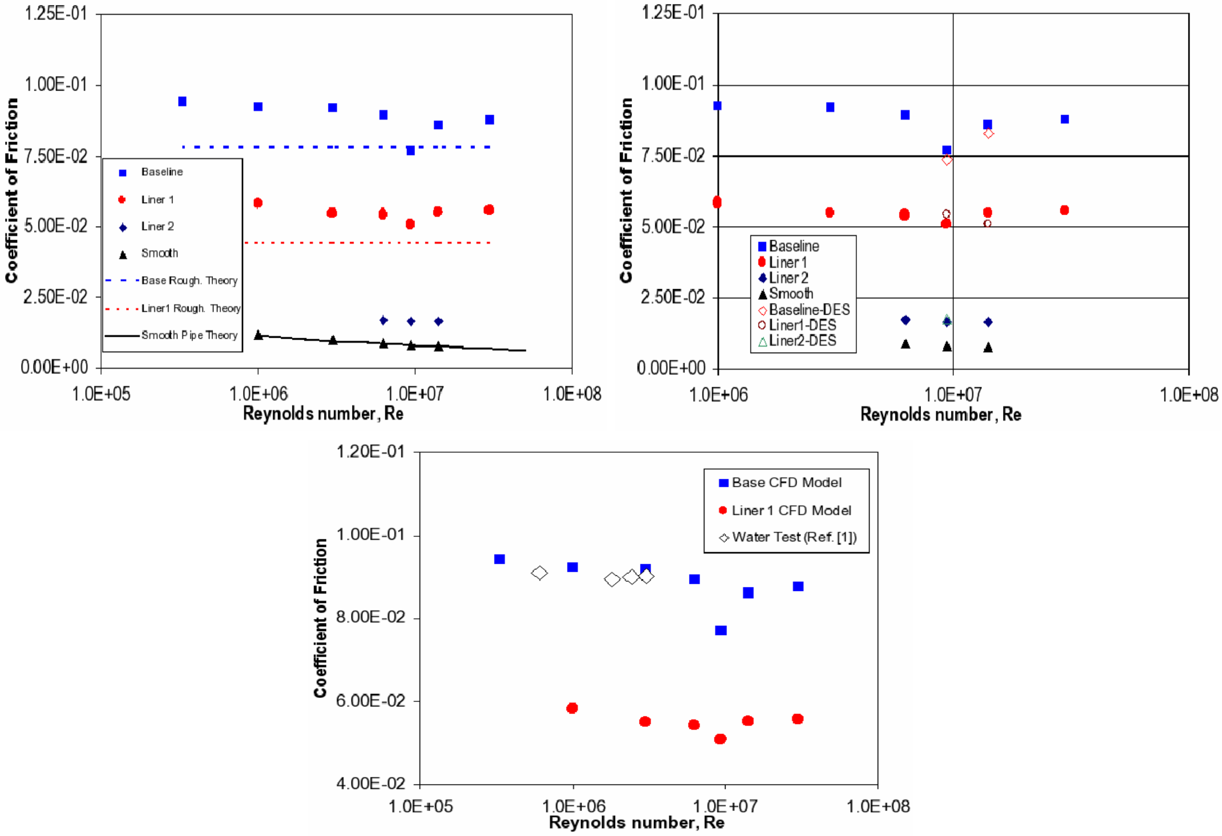

Figure 4: (a) Variation of friction coefficients with Reynolds number and comparison with the theory (b) Variation of friction coefficients for the range of Reynolds number for the RANS and DDES models (c) Comparison of the CFD results for A*=0.0604 of 16” ID pipe with the water test (Ref. [1])

Figure 4(c) shows the comparison of CFD values with the experimental test done with water in 10.5” ID pipe (Ref. [3]). The friction factors are compared with respect the non-dimensional dynamic similarity parameter, Reynolds number. The depth and shape of the corrugation profiles are marginally different between the 16” ID pipe and 10.5” pipe. A reasonable agreement between the CFD and experimental values can be seen.

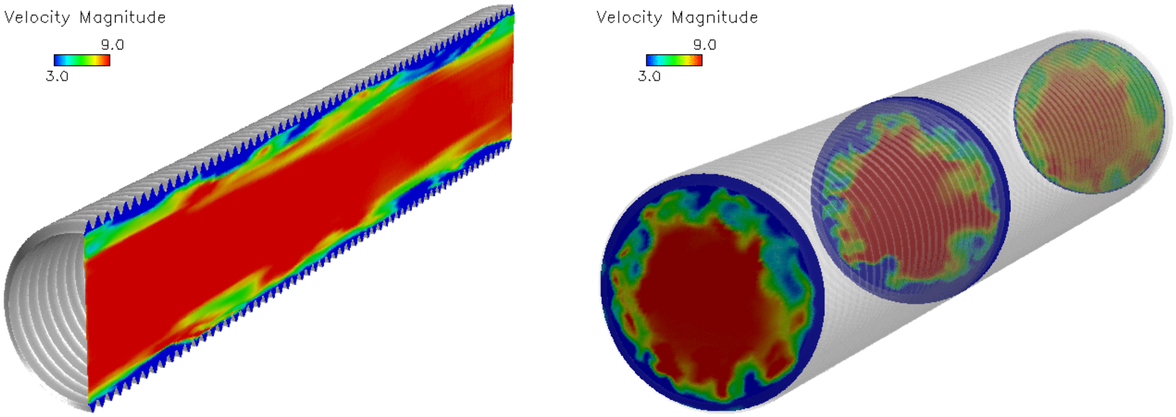

Figure 5: Delayed-DES results showing streamwise and cross-sectional variations.

In corrugated pipe applications, flow physics (e.g., recirculation, separation, mean flow three-dimensionality, streamline curvature, flow acceleration) and geometry play an important role. In this study, we showed that the CFD modeling using AcuSolve can offer an accurate and powerful predictive tool for estimating the macroscopic pressure drop and complex flow phenomenon in the corrugations. The 3D steady RANS and DDES models available in AcuSolve provided a consistent estimate of the pressure drop and friction factor for varying flow rates. Significant 3D turbulence effects are found for the pipe geometry with circular corrugations suggested by both qualitative features and quantitative information. Cryogenic flexible pipe based LNG transfer system seems to be a good candidate for CFD modeling, and to qualify the pipe system for the LNG industry requirements. The reader may wish to consult Ref. [4] for further details.

References

[1] Framo Engineering AS Report, “CFD Calculations of Corrugated Flexible Pipe,” 4577-0313-D, 2006.
[2] http://www.technip.com/pdf/OffshoreLNG.pdf
[3] Frohne, C., Harten, F., Schippl, K., Steen, K.E., Haakonsen, R., Jorgen, E. and Høvik, J. “Innovative Pipe System for Offshore LNG Transfer,” OTC 19239, 2008.
[4] Jaiman, R., Oakley, O. Jr., and Adkins, D., "CFD Modeling of Corrugated Flexible Pipe," OMAE2010-20509 (submitted)