Multi-Fidelity Computational Flow Assurance for Design and Development of Subsea Systems and Equipment

Simon Lo

CD-adapco, Trident House, Basil Hill Road, Didcot, OX11 7HJ, UK
Abstract

The commercial CFD software STAR-CCM+ was used to simulate transportation of viscous oil in a horizontal pipe. The CFD model was first validated against analytical solution for wall shear stress in laminar flows with constant physical properties. The model was then used to investigate the effects of heat loss since the pipe is subjected to cooling by the sea water on the sea bed.

The results show strong thermal stratification across the pipe with cooler fluid at the bottom of the pipe and warmer fluid near the gas-liquid interface. A layer of cooler fluid is also found next to the pipe wall. Higher values for density and viscosity are found in the cooler regions. The higher viscosity in the cooler layer around the wall was found to give a large increase in wall shear stress and therefore higher pressure drop.

Such strong variations in temperature and physical properties across the pipe are difficult to represent by averaged values as used in traditional one-dimensional analysis. The wall shear stress and pressure drop are likely to be under-estimated which could lead to serious problems in operating the pipeline. Three-dimensional CFD method presented in this paper can provide some very valuable information and therefore is an important tool in Computational Flow Assurance.

Keywords: Computational Flow Assurance, CFD, viscous oil transport, pipeline flow

1 Introduction

Flow Assurance is one of the most critical engineering and operational concerns in the oil and gas industry. Computational Flow Assurance (CFA) involves using software to simulate and study the flow behavior in pipeline and equipment so that potential problems could be identified and solved before such systems are installed in the field. This paper presents a CFA study conducted to examine the effects of heat loss in pipeline transportation of viscous oil.

Pipeline transportation of heavy crude oil is of great interest to the oil industry but it is a challenging problem because of the high viscosity of the oil. Since subsea pipelines are subjected to cooling by the ambient sea, the crude oil within will cool down and the viscosity
and density will increase along the pipeline. The cooling and viscosity increase start at the pipe wall significantly affecting the wall shear stress and therefore pressure drop in the pipe. These effects due to heat loss from the pipe are investigated in this paper using the commercial CFD software STAR-CCM+ [1].

2 Test problem

A horizontal pipe is considered in this paper. The pipe is 200m long (L=200m) and has a diameter of 0.2m (D=0.2m). The L/D ratio for the pipe is therefore 1000. The pipe is assumed to be occupied by gas flow at the top half and liquid flow in the lower half, see Figure 1. In this study, only the liquid flow is considered. It is assumed that the flow is symmetrical above the central vertical plane, so that only half of the pipe is modeled. A picture of the three-dimensional CFD grid used in the study is shown in Figure 2.

![Figure 1: Pipe cross section](image1.png)  ![Figure 2: CFD grid](image2.png)

3 Isothermal flow

A uniform velocity of \( U_{\text{inlet}} = 0.1 \) m/s is imposed at the inlet of the pipe. The oil temperature is set at \( T_{\text{inlet}} = 60^\circ\text{C} \). The physical properties of the fluid considered at 60°C are: density \( \rho = 887 \) kg/m³ and viscosity \( \mu = 0.02 \) Pa.s. The corresponding Reynolds number for the liquid flow is \( \text{Re} = 890 \), hence a laminar flow.

The top surface of the model is the gas-liquid interface. It is modeled as a flat rigid surface with an imposed shear stress of 0.02 N/m² which corresponds to an interface shear stress from a gas flow above the liquid at 5 m/s. The computed velocity profile at the end of the 1000D pipe length is shown in Figure 3.
Since isothermal flow is assumed, the physical properties of the fluid remain constant throughout the pipe. Fully developed flow condition is achieved within 40 pipe diameters. The wall shear stress calculated analytically using the method by Biberg [2] is 0.0757 N/m² and 0.0756 N/m² by the CFD model. This comparison study is therefore a validation of the CFD model.

4 Effects of heat loss

The CFD model described above is applied to study the effects of heat loss. The main effect of heat loss is the changes in physical properties. As the fluid is cooled its density and viscosity are increased. The temperature dependent density and viscosity of the model fluid used in this study are:

$$\rho = 927.05 - 0.6695T$$

where $\rho$ is density in kg/m³ and $T$ is temperature in °C.

$$\mu = A_0 + \sum_{i=1}^{n} A_i T^i$$

where $\mu$ is viscosity in Pa.s, $A_0=1.944\times10^{-1}$, $A_1 = -9.39\times10^{-3}$, $A_2 = 2.232\times10^{-4}$, $A_3 = -2.944\times10^{-6}$, $A_4 = 2.044\times10^{-8}$ and $A_5 = -5.788\times10^{-11}$.

Specific heat $C_p=1.9$ kJ/kg and thermal conductivity $k=0.16$ W/(mK) are assumed to be constant.

The flow boundary conditions described for isothermal flow above are retained unchanged. The inlet fluid temperature is set at 60°C, the wall temperature at 10°C due to cooling from the ambient sea and the top surface (gas-liquid interface) temperature at 60°C assuming the fast moving gas remains close to the inlet temperature.
The computed results at the end of the pipe (L=1000D) are shown in Figures 4 to 7. Figure 4 shows the axial velocity profile is rather similar to the isothermal results shown in Figure 3. However, the higher velocity region is confined to the upper centre region of the flow. Figure 5 shows very clearly the thermal stratification of the flow. The lower half of the flow has lower temperature. The higher temperature in the top layer of the flow is maintained by the high temperature imposed at the top surface.

Figure 4  Axial velocity profile  Figure 5  Temperature profile

Figure 6  Density profile  Figure 7  Viscosity profile

The density and viscosity profiles are shown in Figures 6 and 7. As expected the density and viscosity are higher in the cooler area, i.e. bottom of the pipe. Significantly, higher values are also found in a thin layer next to the wall. This higher viscosity in the near wall region is responsible for the large increase in wall shear stress shown in Figure 8 and in the pressure drop shown in Figure 9.
The results described above show that variation in temperature and physical properties across the pipe cross section can be very large. In a one-dimensional analysis such variations are represented by averaged values. In such analysis, the details of the profiles are missing but more importantly the sharp change in temperature and viscosity near the wall cannot be captured. Therefore the wall shear stress and pressure drop are likely to be underestimated. The consequence of such error could be serious if the pressure drop value is used in sizing of the compressor, for example.
5 Conclusions

The commercial CFD software STAR-CCM+ was used to simulate transportation of viscous oil in a horizontal pipe. The CFD model was validated against analytical solution of wall shear stress for laminar flows with constant physical properties. The model was then used to investigate the effects of heat loss and the subsequent change in physical properties.

The results show strong thermal stratification across the pipe with cooler fluid at the bottom of the pipe and warmer fluid near the gas-liquid interface. A layer of cooler fluid is also found next to the pipe wall. The higher viscosity in the cooler layer around the wall was found to give a large increase in wall shear stress and therefore higher pressure drop in the pipe.

Strong variations in temperature and physical properties across the pipe are difficult to represent by averaged values as used in traditional one-dimensional (1D) analysis. In such 1D analysis, the wall shear stress and pressure drop are likely to be under-estimated which could lead to serious problems. Three-dimensional CFD method as presented in this paper can provide some very valuable information and therefore is an important tool in Computational Flow Assurance.

References

