Case Study

Viscous Forces of Pipe Flow using Ansys Fluent: CFD vs. Reynold’s Transport Theorem

Mitchell Pegoda and Shanti Bhushan, Mississippi State University

Edited by Gautham Varma, Ansys Academic Development Team

education@ansys.com

Ansys Software Used

This resource uses Ansys Fluent® fluid simulation software.

Summary

Calculation of friction loss or viscous drag acting on a body (wall) requires knowledge of the near wall velocity gradient over the entire surface. For flows with variable flow regimes, such as developing flow in a pipe, evaluation of flow gradients over the entire surface is a challenge both analytically and experimentally. On the other hand, Reynolds transport theorem (RTT) can provide an estimate of the body forces from the pressure forces and momentum transfer across the inlet and outlet boundaries. In this project, the applicability of RTT in estimating body forces is validated for a pipe flow case, wherein the body force is primarily because of viscous drag. The results show that RTT estimates of viscous forces compared within 0.1% of the CFD predictions.

Table of Contents

1. Problem Statement

Pipe flow presents the fluid-structure interaction between the fluid and the wall of the pipe. This interaction produces a frictional (viscous) force acting opposite of the flow direction. This viscous force can be calculated using the Reynold's Transport Theorem, RTT, considering the total force acting on the control volume and subtracting the pressure forces and momentum imbalance. Herein, we will be using computational fluid dynamics, CFD, particularly fluid flow within a pipe which provides numerical estimates of the viscous forces on the system to validate the accuracy of RTT calculations.

2. Simulation Set-up

A turbulent pipe flow simulation was performed using Ansys Fluent software. The simulation will provide a numerical estimation of the viscous force acting on the pipe. It will also provide the axial velocity and pressure profiles at the inlet and the outlet of the pipe to be used in RTT to compute the viscous forces. The RTT estimates of viscous forces will be then compared with numerical predictions. Fig. 1 shows developing fluid flow through a pipe and what is expected to be reproduced by the CFD. The CFD set up is shown in Fig. 2, which follows the Ansys Innovation Simulation example – Turbulent Pipe Flow – Entrance Length – Homework[1].

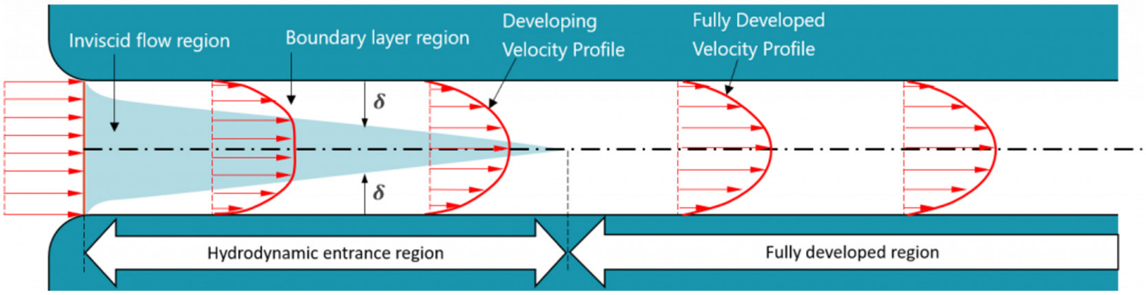

Figure 1. Developing flow within a pipe. Figure taken from Ansys Innovation website.

The simulation was set-up assuming airflow with density () of 1.225 kg/m3 and dynamic viscosity () of 1.789410-5 kg/m-s. The pipe has diameter of 1 m, and length of 5 m. 2D axisymmetric simulations are performed using only half domain using axis boundary at the centerline and no-slip boundary at the wall. Inflow boundary condition with uniform velocity Uin = 1.16 m/s and pressure Pin = 1.62Pa was specified at the inlet. The outlet boundary was specified to be at gauge pressure of 1 Pa.

The numerical scheme used for the simulation included distance-based Rhie Chow flux scheme for pressure-velocity coupling, least square cell base gradient scheme, second-order spatial scheme for pressure Poisson equation, and second-order upwind scheme for momentum equation. Pseudo time step with adaptive time step size was used for temporal integration. Simulation was performed over 500 iterations and residuals were monitored and was deemed converged as the residuals dropped to 10-6.

1 https://innovationspace.ansys.com/courses/courses/real-internal-flows/lessons/simulation-examples-homework-and-quizzes-4/topic/turbulent-pipe-flow-entrance-length-homework/

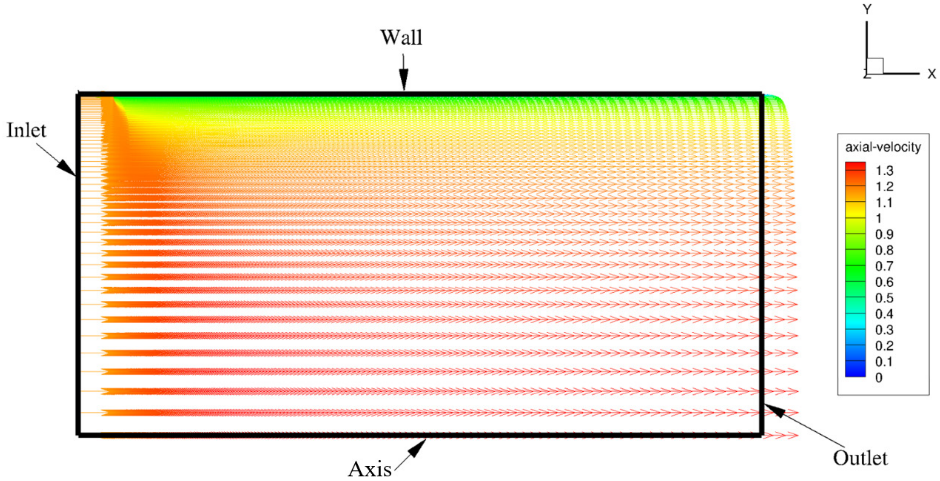

Figure 2. CFD Setup/Axial Velocity Vector Plot. The plot is scaled in the Y direction to show the flow trends.

3. CFD Results

Once the solution converged, the pressure profile over the length of the pipe and velocity profile at the inlet and the outlet were written to a file and plotted in excel (Fig. 3 and 4). As expected, the flow streamline developed from a uniform profile at the inlet to a parabolic profile also shown in Fig. 2 and 3(a). The pressure on the other hand was uniform both at the inlet and the outlet and varied linearly throughout the length of the pipe (Fig. 3b). The CFD predictions of the viscous forces acting on the pipe was -1.2409 N also notes as shown in Table 1. Note that the viscous force is negative as it acts in the direction opposite to the flow.

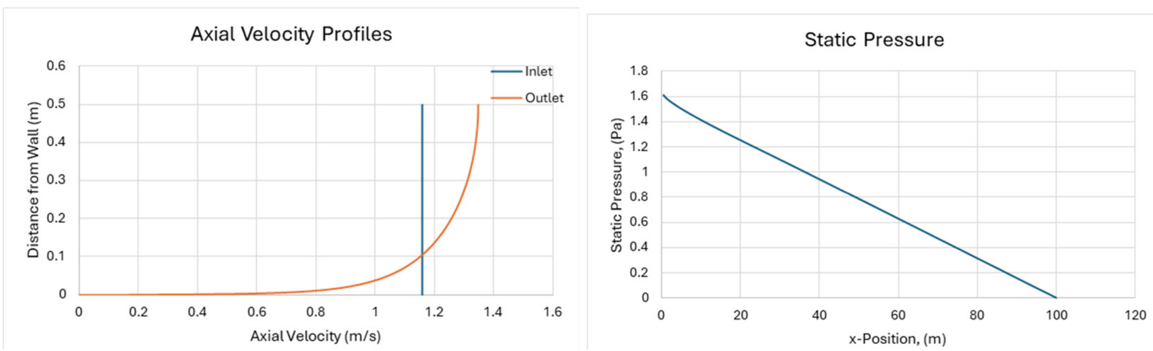

Figure 3. Axial velocity profile at the inlet and outlet (left-a),
and pressure profile over the length of the pipe (right-b).

4. Drag Predictions using Reynolds Transport Theorem

Mass and Momentum Conservation Using Reynolds Transport Theorem (RTT) provides conservation principle of quantity B_{sys} of a system such that:

$$\frac{dB_{sys}}{dt} = \frac{d}{dt}\int_{CV} \rho\beta d\forall + \int_{CS} \rho\beta(\boldsymbol{V}\cdot\boldsymbol{n})dA$$

where, $\beta=B_{sys}/m_{sys}$ is the per unit mass (m_{sys}) of the quantity, V is the velocity vector and n is the outward surface normal.

For mass conservation, B_{sys}=m and β=1, resulting in:

$$\underbrace{\frac{dm_{sys}}{dt}}_{=0} = \underbrace{\frac{d}{dt}\int_{CV} \rho d\forall}_{=0} + \int_{CS} \rho(\boldsymbol{V}\cdot\boldsymbol{n})dA$$

Since the flow achieves a steady state, the forts term on the right is zero. The left had side is governed by the physics, mass of the system remains same. Thus, the mass conservation simplifies to:

$$\int_{CS} \rho(V\cdot n)dA = 0$$

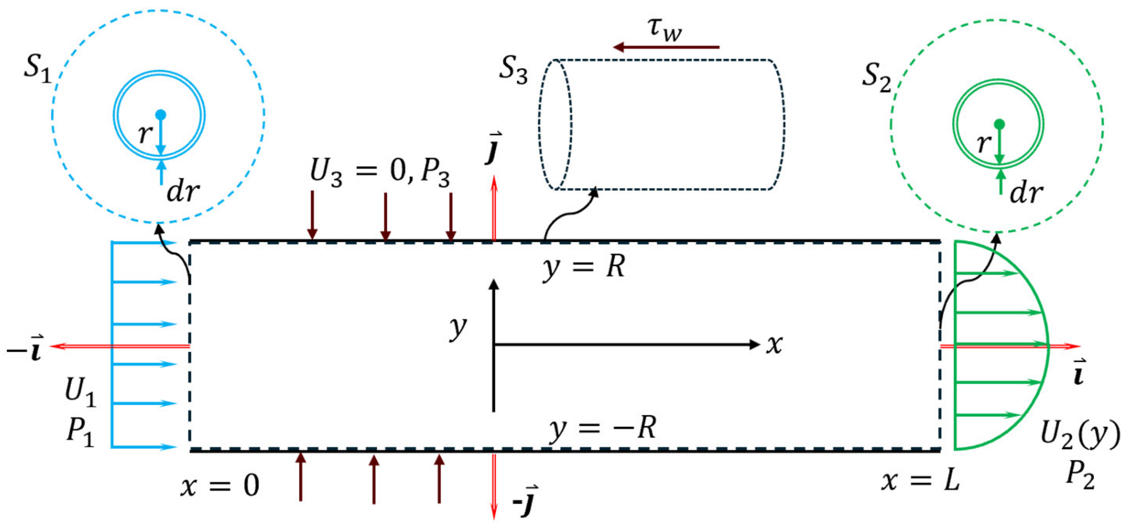

Figure 4. Control volume for the pipe flow.

As shown in Fig. 4, the control volume for the pipe flow case has three surfaces circular inlet surface S_1, cylindrical outlet surface S_2 and cylindrical wall surface S_3. Using the velocity components V=Ui+Vj the integral for each surface becomes:

$$-\int_{S_1} \rho U_1 dA + \int_{S_2} \rho U dA + \underbrace{\int_{S_3} \rho(V\cdot n)dA}_{=0} = 0$$

The last term on LHS (integral on surface S_3) is zero because of no-slip boundary condition. The flow is uniform on surface S_1 resulting in mass flow of:

$$m_{in} = -\int_{S_1} \rho U_1 dA = \rho U_1 \pi D^2/4 \qquad (1)$$

The velocity profile varies along the radial direction on surface S_2 as shown in Fig. 5, thus, the integral is:

$$m_{out} = \int_{S_2} \rho U dA = \int_0^{D/2} \rho U 2\pi\, r\, dr \qquad (2)$$

The integral is computed using trapezoidal rule in excel using the velocity profile as:

$$\int_0^{D/2} \rho U 2\pi\, r\, dr = \sum_{k=1}^{N} \frac{2\rho\pi[(Ur)_{k-1} + (Ur)_k]}{2}\Delta r$$

where N=100 uniformally spaced stencil points used to discretize the velocity profile in the radial direction, k and k-1 are the local stencil points. Δr=D/2N. The results of the numerical integration are shown in Table 1 below.

From RTT momentum conservation B_{sys}= mU and β= U,

$$\frac{dMsys}{dt} = \underbrace{\frac{d}{dt}\int_{CV} \rho U d\forall}_{=0} + \int_{CS} \rho U (V \cdot n) dA$$

$$F_p + F_v = \int_{CS} \rho U (V \cdot n) dA = -\int_{S_1} \rho U_1^2 dA + \int_{S_2} \rho U^2 dA + \underbrace{\int_{S_3} \rho U (V \cdot n) dA}_{=0}$$

the local change is neglected due to steady flow as in mass conservation. The surface flux is calculated by integrating the momentum across the cross-sectional area for each of the surfaces S_1, S_2, and S_3. Again, there is no flux through the outer surface, S_3 due to the no slip condition. The RTT integral at S_2 is again approximated by a trapezoidal rule in excel:

$$M_{out} = \int_0^{D/2} \rho U^2 2\pi\, r\, dr = \sum_{k=1}^{N} \frac{2\rho\pi\,[(U^2\, r)_{k-1} + (U^2\, r)_k]}{2} \Delta r \qquad (3)$$

For the inlet, S_1, the flux can be simplified to

$$M_{in} = -\int_{S_1} \rho U_1^2\, dA = \rho U_1^2 \pi D^2/4 \qquad (4)$$

as the velocity profile is uniform. To find the viscous force, the pressure force is calculated from the Ansys Fluent software pressure data at the outlet and inlet and is subtracted from the total force given by the RTT. This value is compared to the viscous force output provided by Ansys Fluent software.

$$F_v = -F_p + M_{in} + M_{out} \qquad (5)$$

$$F_p = (P_{in} - P_{out})\pi R^2 \qquad (6)$$

The results shown in Table 1 demonstrate mass conservation with very little error. Also, the Ansys Fluent software viscous force report matches the RTT calculation with an error of 0.09%. The small error between CFD and RTT validates its accuracy.

Table 1: Mass conservation and force predictions obtained using RTT is compared with CFD predictions

m_{in} (kg) (Eq. 1)	m_{out} (kg) (Eq. 2)	F_p (N) (Eq.6)	M_{in} (N) (Eq. 3)	M_{out} (N) (Eq. 4)	RTT F_v (N) (Eq. 5)	CFD F_v (N)
1.11605	1.115533	1.27E+00	-1.29	1.32572817	-1.24	-1.2409
Error 0.046357%					Error 0.0904%	

5. Conclusions

In this case study, Ansys Fluent software was employed to validate the Reynold's Transport Theorem. The analysis was based on developing pipe flow, where an axisymmetric simulation was conducted for air to generate the velocity and pressure profiles at the inlet and outlet of the pipe. These profiles were then integrated according to Reynold's Transport Theorem. The simulation provides a numerical estimate of frictional force. The RTT calculation matched the CFD results closely, within 0.1%, validating the Reynold's Transport Theorem. Ansys Fluent software proved to be an invaluable tool for this project, offering accurate flow modeling capabilities for developing pipe flow.

Use and Reproduction
The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited. The full Academic Terms & Conditions can be found using this link.

Document Information
This case study is part of a set of teaching resources to help introduce students to topics related to fluids.

Ansys Education Resources
To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit www.ansys.com/education-resources.

Feedback
Here at Ansys, we rely on your feedback to ensure the educational content we create is up-to-date and fits your teaching needs.

Please click the link here out a short survey (~7 minutes) to help us continue to support academics around the world utilizing Ansys tools in the classroom.

ANSYS, Inc.
Southepointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com