Case Study

Estimation of Airfoil Drag using Reynolds Transport Theorem and Ansys Fluent Software

Edward Kenyon and Shanti Bhushan, Mississippi State University

Edited by Gautham Varma, Ansys Academic Program

education@ansys.com

Ansys Software Used

This resource uses Ansys Fluent® fluid simulation software.

Summary

In this case study, Reynolds Transport Theorem (RTT) is used to calculate total, pressure and viscous forces acting on an airfoil at zero-degree angle of attack, and the results are compared with CFD predictions. To achieve this, CFD simulations are performed using Ansys Fluent software for Ma = 0.6 flow over NACA 0012 airfoil. The RTT total drag, pressure drag, and viscous drag estimates compare withing 1.66%, 1.42% and 4.2% of CFD predictions, reasonable.

Table of Contents

1. Problem Statement

A moving body encounters pressure and viscous drag. The pressure and viscous drag force calculations require knowledge of pressure and near-wall velocity gradient distribution over the entire surface, respectively. CFD can provide these two quantities with ease but is complicated in experiments. The alternative approach is to use the Reynolds Transport Theorem (RTT), wherein the pressure and momentum variation across the off-body control surfaces can help evaluate the total drag experienced by the body. This project uses CFD predictions of flow over an airfoil to validate the applicability of RTT in predicting drag. This project complements the project" Viscous Forces of Pipe Flow: CFD vs. Reynold's Transport Theorem, available on Ansys Innovation website and includes additional complexity. In particular, the drag force in the pipe flow project involved only frictional component, whereas in the test case considered herein the body drag involves both pressure and viscous components. In addition, the RTT is extended to include airfoil surface as a control surface to compute the viscous drag. The RTT predictions are then compared with CFD predictions obtained using near-wall velocity gradients.

2. Approach

The simulation was performed using Ansys Fluent software airfoil flow example[1]. The simulation involved 2D flow over a NACA0012 airfoil at 0 deg angle of attack. As shown in Fig. 1, the simulation domain included a C-type grid with an inflow boundary on the left, outflow boundary to the right, and no-slip conditions on the airfoil surface. The inflow boundary was specified to be pressure far-field with Ma = 0.6 and atmospheric pressure. The outflow boundary was specified to be pressure far-field at atmospheric conditions. Compressible flow simulations were performed using K-omega SST model for turbulence, ideal gas law for the equation of state, and Sutherland law for viscosity. Pressure-velocity coupling was performed using Rhie-Chow flux reconstruction. Second-order upwind scheme was used for spatial discretization. Steady state solution was obtained using automatic time step.

Figure 2 shows the converged solution. One can observe stagnation point at the leading edge, flow acceleration over the airfoil surface, and wake emerging from the trailing edge. The velocity and pressure contours show complimentary behavior as expected from Bernoulli's equation. CFD simulations provide total drag of 281.98 N and the pressure and viscous drag account for 131.89 N and 150.088 N, respectively.

Reynolds Transportation Theorem (RTT) states that the rate of change in a property for a system is equal to the time rate of change of the property within the control volume and the flux of the property through the control surface.

$$\frac{dB}{dt} = \frac{d}{dt}\int\int\int \rho\boldsymbol{\beta}\, d\forall + \int\int \rho\boldsymbol{\beta}(\boldsymbol{V}\cdot n)\, dA \qquad (1)$$

In the above equation B is the extensive property and is the intensive property. In the above equation, B = and =1 provides the mass conservation. Whereas B = V and =V, where V is velocity vector, gives the conservation of momentum equation. The rate of change of momentum can help in calculating the forces (F) acting on the control volume, such that:

$$\frac{d\rho\boldsymbol{V}}{dt} = \frac{d}{dt}\int\int\int \rho\boldsymbol{V}\, d\forall + \int\int \rho\boldsymbol{V}(\boldsymbol{V}\cdot\boldsymbol{n})dA = \boldsymbol{F} - \int\int p\,\boldsymbol{n}\, dA \qquad (2)$$

Note that a negative sign is used as pressure term, as it is compressive and acts opposite to the surface normal.

1 https://innovationspace.ansys.com/courses/courses/basics-of-compressible-flows/lessons/simulation-examples-homework-and-quizzes-10/topic/simulation-example-an-airfoil-in-different-flow-regimes/

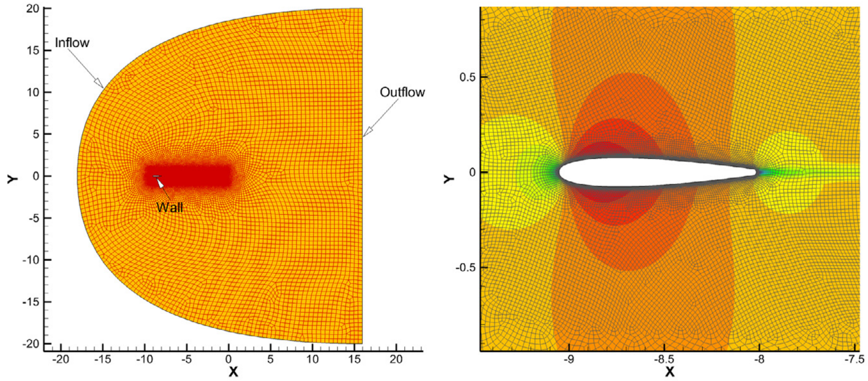

Figure 1. Simulation domain, grid and boundary condition.

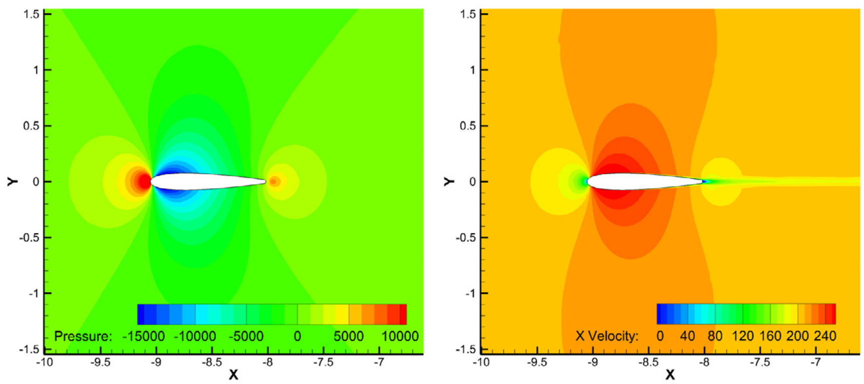

Figure 2: Converged solution of Ma = 0.6 flow over an airfoil.

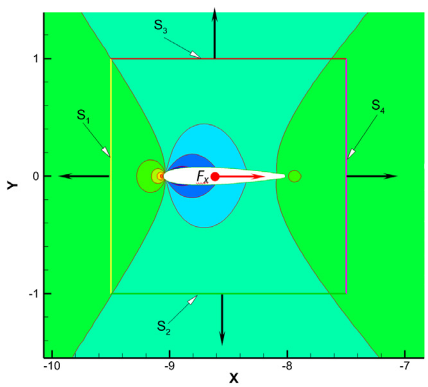

Figure 3. Control surfaces used for RTT integration. The black arrows show the surface normal.

The control surface for the RTT application is shown in Fig. 3, which involves 4 surfaces S_1, S_2, S_3 and S_4. Notice that the surface normal is always outward, i.e., the surface normal for S_1 is along -X direction ($-i_x$), for S_4 along X direction (i_x), for S_2 along -Y direction ($-i_y$) and for S_3 along Y direction (i_y). The RTT equations for mass and momentum can be then discretized on to the above surfaces. In addition, as the flow is steady, the time derivative terms are zero.

The RTT for mass conservation (Eq. 1) dictates that:

$$\frac{d\boldsymbol{B}}{dt} = \iint\limits_{S_1} \rho(\boldsymbol{V}\cdot\boldsymbol{n})dA + \iint\limits_{S_2} \rho(\boldsymbol{V}\cdot\boldsymbol{n})dA + \iint\limits_{S_3} \rho(\boldsymbol{V}\cdot\boldsymbol{n})dA + \iint\limits_{S_4} \rho(\boldsymbol{V}\cdot\boldsymbol{n})dA = 0$$

Using the surface normal direction, the above equation simplifies to:

$$-\iint_{S_1} \rho U\, dy - \iint_{S_2} \rho V\, dx + \iint_{S_3} \rho V\, dx + \iint_{S_4} \rho U\, dy = 0 \tag{3}$$

where, U and V are the velocity components along X and Y direction, respectively.

The RTT for momentum (Eq. 2) gives:

$$\boldsymbol{F} = \int\int \rho \boldsymbol{V}(\boldsymbol{V}\cdot\boldsymbol{n})dA + \int\int p\,\boldsymbol{n}\, dA$$

Using the control surfaces, the above equation simplifies to:

$$\boldsymbol{F} = -\iint_{S_1} \rho \boldsymbol{V} U\, dy - \iint_{S_2} \rho \boldsymbol{V} V\, dx + \iint_{S_3} \rho \boldsymbol{V} V\, dx + \iint_{S_4} \rho \boldsymbol{V} U\, dy - \boldsymbol{i_x}\iint_{S_1} p\, dy - \boldsymbol{i_y}\iint_{S_2} p\, dx + \boldsymbol{i_y}\iint_{S_3} p\, dx + \boldsymbol{i_x}\iint_{S_4} p\, dy \tag{4}$$

The drag force is along the X direction, which can be computed from above equation by dot product of the force with i_x, i.e.,

$$F_x = \boldsymbol{F}\cdot\boldsymbol{i_x} = -\iint_{S_1} \rho UU\, dy - \iint_{S_2} \rho UV\, dx + \iint_{S_3} \rho UV\, dx + \iint_{S_4} \rho UU\, dy - \iint_{S_1} p\, dy + \iint_{S_4} p\, dy \tag{5}$$

Note that the above force is the force required to keep the airfoil at place (as if the airfoil is held by a mounting system), which is opposite to the drag force, $F_T=-F_x$.

Further, the drag force acting on the control volume must equal the viscous (F_V) and pressure (F_P) forces acting on the body surfaces as shown in Fig. 4:

$$F_T = F_P + F_V \tag{6}$$

The pressure drag can be obtained from the surface pressure distribution:

$$F_P = -\int\int p\,\boldsymbol{n}\, dA\cdot\boldsymbol{i_x} = -\iint_{S_5} p\, dy \tag{7}$$

Once F_T is estimated from Eq. (5) and F_P from Eq. (7), then Eq. (6) can provide an estimate of F_V.

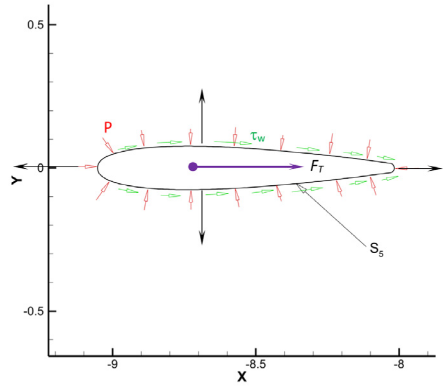

Figure 4. The pressure (P) and viscous force (w) acting on the airfoil surface. The red arrows show the compressive pressure acting surface normal. The green arrow shows wall shear acting along the surface. The black arrow shows the surface normal.

3. Results and Analysis

The CFD predictions were interpolated on the control surfaces S_1, S_2, S_3 and S_4 to obtain the velocity, pressure and density distribution along the surfaces, as shown in Fig. 5. The CFD results provided the pressure distribution on the airfoil surface S_5 as shown in Fig. 6. As expected, the pressure variation on the top and bottom surface of the airfoil are same, as the airfoil is expected to have no lift.

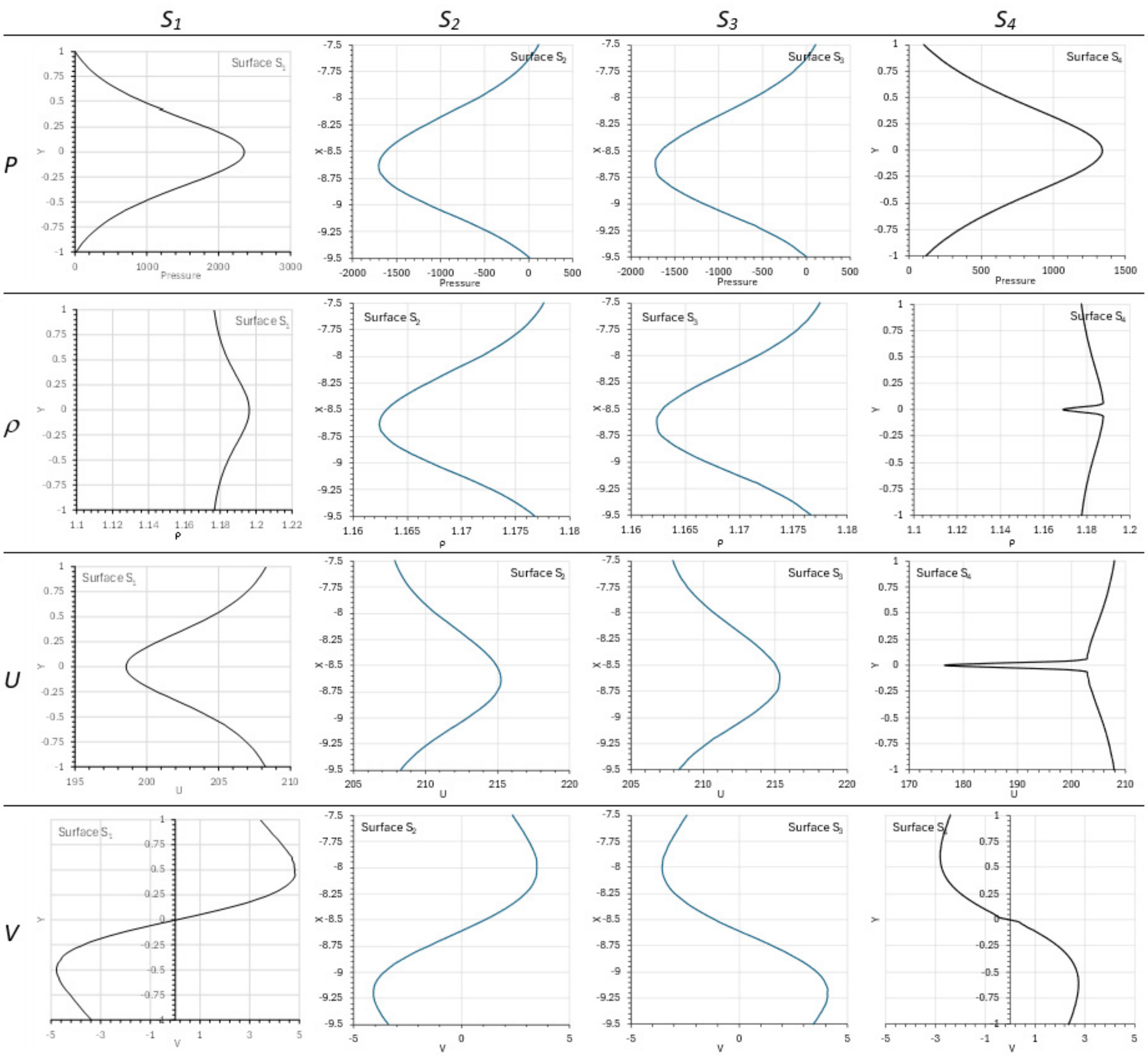

Figure 5: Velocity, pressure and density variation over the control surfaces.

The above profiles were used in Eqs. (3), (5) and (7) to evaluate the integrals using trapezoidal rule assuming a span-wise width of 1m, as summarized in Table 1. For Eq. (3), the magnitude of the integral over surfaces S_1 and S_4 were around 483.32 kg/s and 483.88 kg/s values, respectively, and over the surfaces S_2 and S_3 were 0.297 kg/s and 0.269 kg/s, respectively. This is expected, as the mass flow is primarily across the surface normal to the flow. The summation over all the surfaces shows a residual value of 1.6 e-3 kg/s, which is 3.33e-4% of the inflow mass flux (through S_1). The small error could be because of numerical integration; thus, the RTT analysis satisfies the mass conservation well, i.e., mass entering the control volume balances the mass exiting the control volume.

The integral terms for Eq. (5) shows momentum flux of 9845.3 N and 9907.8 N across S_1 and S_4, respectively. Notice that the downstream surface has more momentum flux than those at the upstream.

This seems counterintuitive; however, this behavior is observed because the upstream surface is close to the airfoil and the flow is affected by stagnation flow. It is expected that the reduction in momentum will be balanced by the increase in pressure force. The momentum flux across S_2 and S_3 are 54.917 N and -60.78 N. Considering the direction of the normal there is momentum loss from the control volume from these surfaces. The magnitude of pressure force on S_1 and S_4 are 2151.2 N and 1445.3 N, respectively. Summation of the momentum and pressure force terms over all the surfaces as in Eq. (5), provides a net force of F_x=-286.67 N. Note that the force direction is negative, this is because RTT provides reaction to the drag force. The RTT prediction of the drag is 1.67% higher than the CFD predictions, which is deemed reasonable.

The pressure drag can be obtained from the surface integral in Eq. (7), which gives F_P= 130.01 N. The CFD solution provides F_P=131.89 N. Thus, the surface integral compares within 1.42% with the CFD. The viscous drag can be obtained using Eq. (6), which gives F_V=156.66 N. RTT predictions of the viscous drag is 4.2% higher than the CFD predictions.

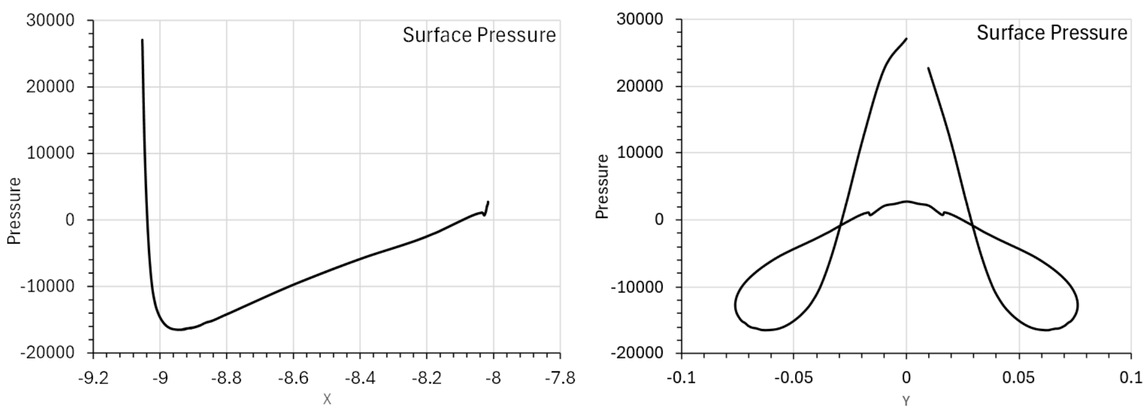

Figure 6. Surface pressure distribution over the airfoil surface.

Table 1: Summary of the control surface integrals.

CFD Predictions	
F_P	131.89 N
F_V	150.088 N
F_T	281.978 N

Surface/Integral	S_1	S_2	S_3	S_4	S_5
∫ρU dy (kg/s)	4.8332E+02			4.8388E+02	0
∫ρV dx (kg/s)		2.6929E-01	-2.9710E-01		
Mass Conservation (Eq. 3)	-1.611E-03 kg/s (= 3.33E-04%S_1)				
∫ρUU dy (N)	9.8543E+04			9.9078E+04	
∫ρUV dx (N)		5.4917E+01	-6.0784E+01		
∫p dy (N)	2.1512E+03			1.4453E+03	130.01 N (ε=1.42%)
Total drag, $-F_x$ (Eq. 5)	286.67 N (% = 1.66)				
F_V (Eq. 6) (N)	156.66 N (= 4.2%)				

4. Conclusions

In this study, calculation of total, pressure and viscous forces acting on an airfoil at zero-degree angle of attack is computed using Reynolds Transport Theorem, and the results are compared with CFD predictions. To achieve this, steady state compressible CFD simulations are performed for NACA 0012 airfoil at Ma = 0.6, and the flow variables (density, velocity and pressure) are interpolated on user specified control surfaces around the airfoil. The trapezoidal rule is then used to compute mass flux, momentum flux and pressure forces acting on the control surfaces. RTT estimate of mass conservation shows a nominal error of 3e-4%. The total drag estimate using RTT compares withing 1.66% of CFD. The pressure drag computed using surface pressure integral compares within 1.42% of the CFD values, and the viscous drag calculation compares within 4.2% of the CFD predictions. Overall, the RTT predictions are deemed reasonable.

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