FLUENT - Flow over an Airfoil- Step 7

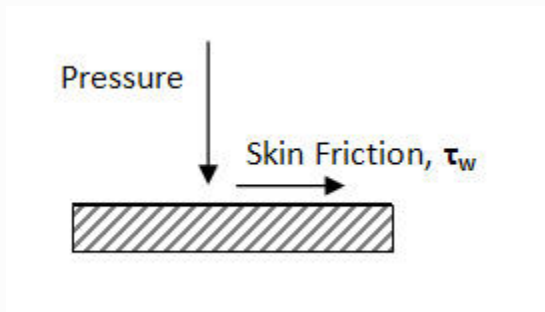## Step 7: Validate the Results

FLUENT report forces in term of pressure force and viscous force. For instance, we are interested in the drag on the airfoil,

$(Drag)_{total} = (Drag)_{pressure} + (Drag)_{viscous}$

Drag due to pressure:

$$(Drag)_{pressure} = \oint P\hat{n}.\hat{e}_d dS$$

Drag due to viscous effect:

$$(Drag)_{viscous} = \oint \tau_w \hat{t}.\hat{e}_d dS$$

where

e_d is the unit vector parallel to the flow direction.

n is unit vector perpendicular to the surface of airfoil.

t is unit vector parallel to the surface of airfoil.

Similarly, if we are interested in the lift on the airfoil,

$(Lift) = (Lift)_{pressure} + (Lift)_{viscous}$

Lift due to pressure:

$$(Lift)_{pressure} = \oint P\hat{n}.\hat{e}_l dS$$

Lift due to viscous effect:

$$(Lift)_{viscous} = \oint \tau_w \hat{t}.\hat{e}_l dS$$

where

e_l is the unit vector perpendicular to the flow direction.

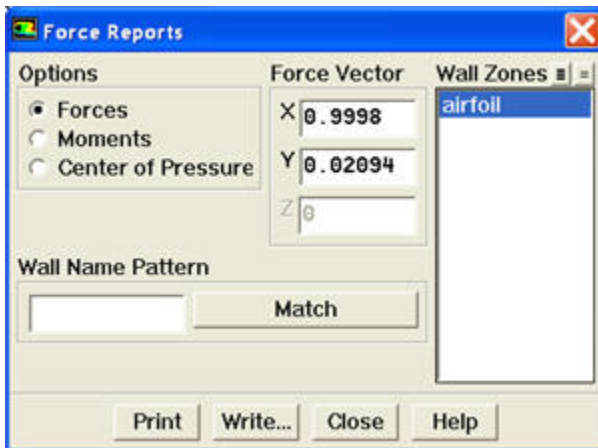n is unit vector perpendicular to the surface of airfoil.

t is unit vector parallel to the surface of airfoil.

Report Force

We will first investigate the Drag on the airfoil.

Main Menu > Report > Forces...

Select *Forces*. Under *Force Vector*, enter 0.9998 next to *X*. Enter 0.02094 next to *Y*. Select *airfoil* under *Wall Zones*. Click *Print*.

Here's is what we see in the main menu:

```
Force vector: (0.99980003 0.02094 0)
                         pressure        viscous          total       pressure       viscous            to
tal
zone
name                        force          force          force    coefficient    coefficient    coefficient
                                n              n              n

------------------------ -------------- -------------- -------------- -------------- --------------
--------------
airfoil                     3.8125084              0      3.8125084   0.0024897052              0   0.0024897
052
------------------------ -------------- -------------- -------------- -------------- --------------
--------------
net                         3.8125084              0      3.8125084   0.0024897052              0   0.0024897
052
```

$C_d = (C_d)_{pressure} + (C_d)_{skin\ friction}$

where

$(C_d)_{pressure}$ is due to pressure force.

$(C_d)_{skin\ friction}$ is due to viscous force.

Indeed, we see that the $(C_d)_{skin\ friction}$ is zero because of the inviscid model.

> ⓘ   In reality, $(C_d)_{skin\ friction}$ has biggest contribution to drag but ignored because of the inviscid model that we specify. $(C_d)_{pressure}$ should be zero, but it is not zero because of inaccuracies and numerical dissipation during the computation.

Now, let's look at the lift coefficient.

Main Menu > Report > Forces...

Select *Forces*. Under *Force Vector*, enter -0.02094 next to *X*. Enter 0.9998 next to *Y*. Select *airfoil* under *Wall Zones*. Click *Print*.

Here's is what we see in the main menu:

```
Force vector: (-0.02094 0.99980003 0)
                              pressure        viscous          total       pressure        viscous
total
zone name                        force          force          force    coefficient    coefficient
coefficient
                                     n              n              n
------------------------ -------------- -------------- -------------- -------------- --------------
--------------
airfoil                       1008.3759              0      1008.3759      0.6585058              0
0.6585058
------------------------ -------------- -------------- -------------- -------------- --------------
--------------
net                           1008.3759              0      1008.3759      0.6585058              0
0.6585058
```

Similarly, lift force is due to the contribution of pressure force and viscous force.

$C_l = (C_l)_{pressure} + (C_l)_{skin\ friction}$

where

$(C_l)_{pressure}$ is due to pressure force.

$(C_l)_{skin\ friction}$ is due to viscous force.

Since our model is inviscid, $(C_l)_{skin\ friction}$ is zero. We see that the lift coefficient compare well with the experimental value of 0.6.

> ⓘ Do note that the lift coefficient for inviscid model is higher than the experimental value. In reality, if we take into account the effect of viscosity, we will have $(C_l)_{skin\ friction}$ of negative value. The viscous effect will lower the overall lift coefficient. Since our inviscid model neglect the effect of viscosity, we have a slightly higher lift coefficient compared to the experimental data.

Grid Convergence

A finer mesh with four times the original mesh density was created. The lift coefficient was found to be 0.649.

	Original Mesh	Fine Mesh	% Dif
C_l	0.647	0.649	0.3%
C_d	0.00249	0.00137	45%

We see that the difference in drag coefficient is very large. We used inviscid case for our model, so we are expecting a C_d of zero. However, since the parameter of interest is the lift coefficient, and the value lift coefficient does not deviate much from original mesh to fine mesh, we concluded that the fine mesh is good enough.

> ⓘ The modeling result obtained is still off from the literature result. Further validation steps are needed before we can conclude about the accuracy of our model. Other parameter that will affect the validity of our result is the choice of viscous model. We used inviscid model which basically assumed that the flow inviscid and totally ignore the effect of boundary layer near the airfoil surface. We might want to try out turbulence model for this high Reynolds number flow.

Summary

Following table shows comparison of modeling result with experimental data.

	C_l	C_d
FLUENT Fine Mesh	0.649	0.00137
Experiment	0.6	0.007
Theory	-	0

Though further validation steps are still needed before we can come up with a model that will accurately represent the physical flow, this simple tutorial demonstrates the use of reasonable assumption and approximation in obtaining understanding of physical flow properties around an airfoil.

Reference

The experimental data is taken from **Theory of Wing Sections** By Ira Herbert Abbott, Albert Edward Von Doenhoff pg. 488

Google scholar link

Go to Problem 1

See and rate the complete Learning Module

Go to all FLUENT Learning Modules