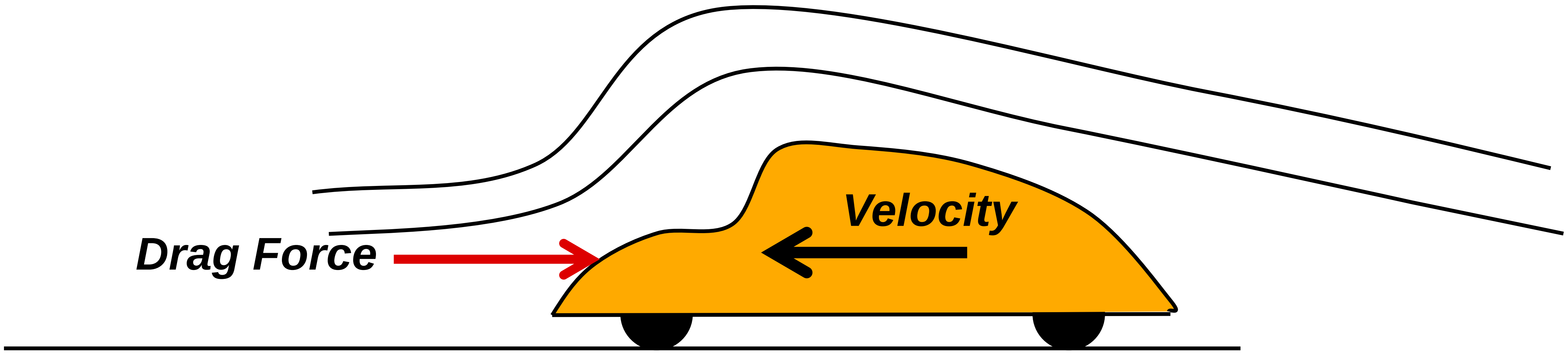
Basic Concepts: Drag

Objectives

- Introduce the drag force that acts on a body moving through a fluid
- Discuss the velocity and pressure distributions acting on the body
- Introduce the drag equation
- Show how the drag coefficient can be determined from a CFD program

Drag

- When an object moves through a fluid, there is a force acting on the body by the fluid.
- The force acts in the opposite direction of the relative velocity between the fluid and the body.
- The force depends on the shape and size of the body, as well as the density, viscosity, and velocity of the fluid.



Drag Equation

The Drag Force is given by the equation

$$F = \frac{1}{2} C_D \cdot \rho \cdot A \cdot V^2$$

F	➔	*Drag Force*
C_D	➔	*Drag Coefficient*
ρ	➔	*Fluid Mass Density*
A	➔	*Projected Area*
V	➔	*Relative Velocity*

Example: $C_D = 0.6$

$$\rho = 1.11x10^{-7} \frac{lbf\text{-}\sec^2}{in^3 - in}$$

$$A = 0.785\, in^2$$

$$V = 50 \frac{in}{\sec}$$

➔ $F = 6.54x10^{-5}\ lb$

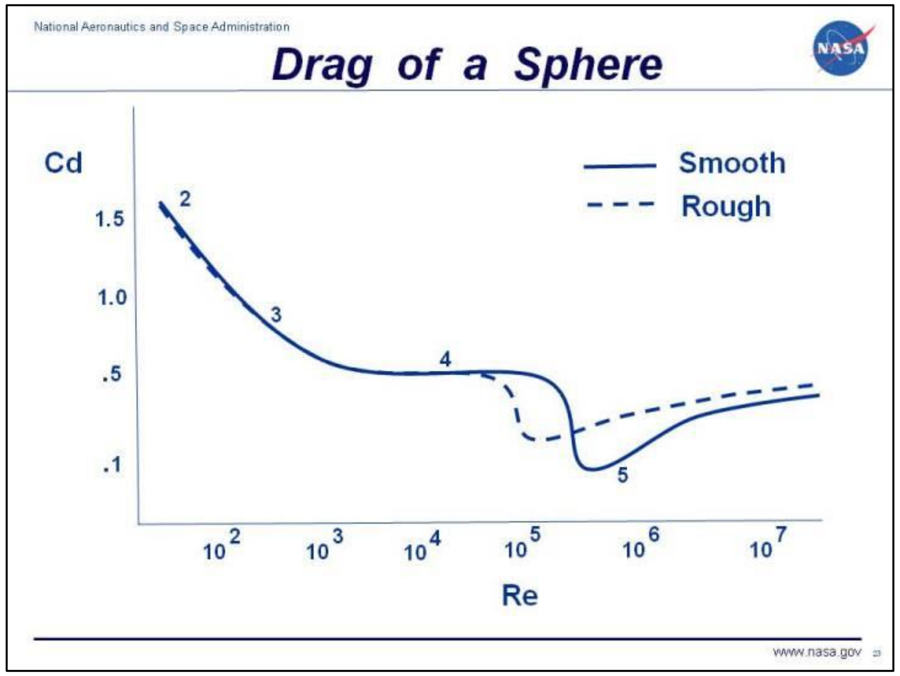
Reynolds Number

The Drag Coefficient is dimensionless, is determined experimentally, and is a function of the Reynolds number. The figure shows the Drag Coefficient for a sphere plotted as a function of the Reynolds Number.

Reynolds Number

$$Re_L = \frac{V \cdot L}{\nu}$$

Re_L	➔	Reynolds Number
V	➔	Relative Velocity
L	➔	Characteristic Length
ν	➔	Kinematic Viscosity



The Reynold's Number is a dimensionless quantity. The kinematic viscosity has units of length-squared divided by seconds.

http://www.grc.nasa.gov/WWW/k-12/airplane/dragsphere.html

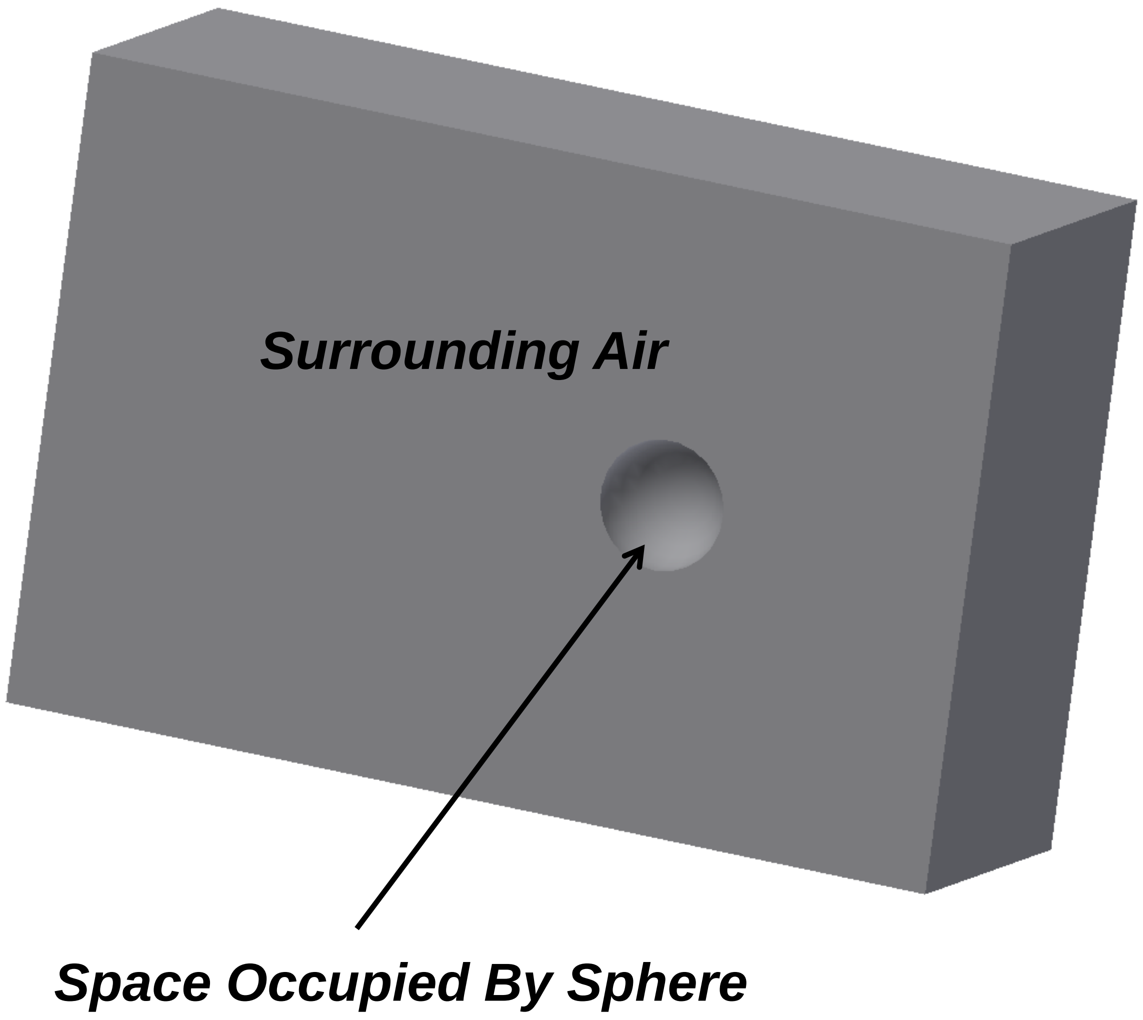
Example - Sphere

Autodesk Simulation CFD will be used in this example to basic concepts associated with flow over bodies.

The specific problem will be flow around a sphere.

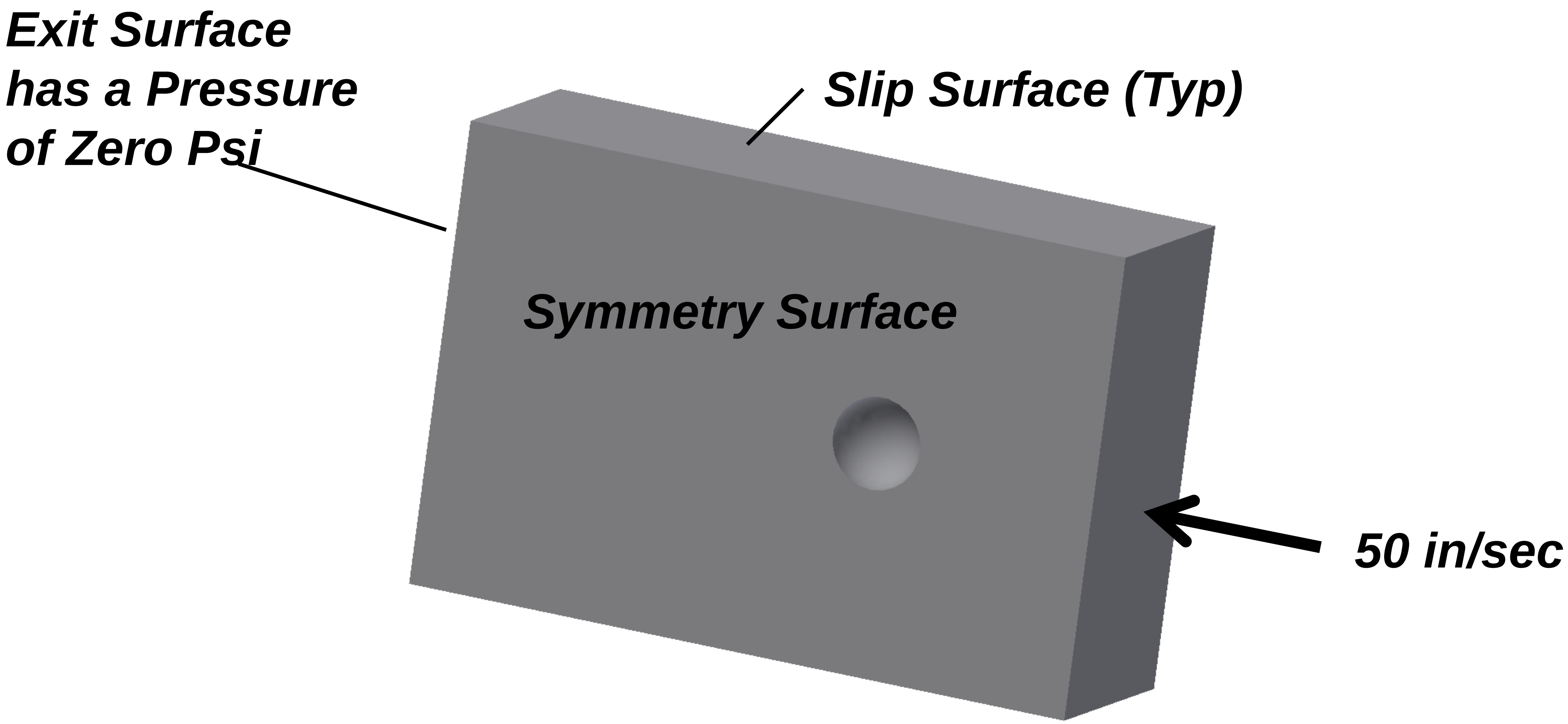
Autodesk Inventor was used to model the air surrounding the sphere.

The sphere has a diameter of 1 inch and the surrounding air is five inches by seven inches by 2.5 inches thick.



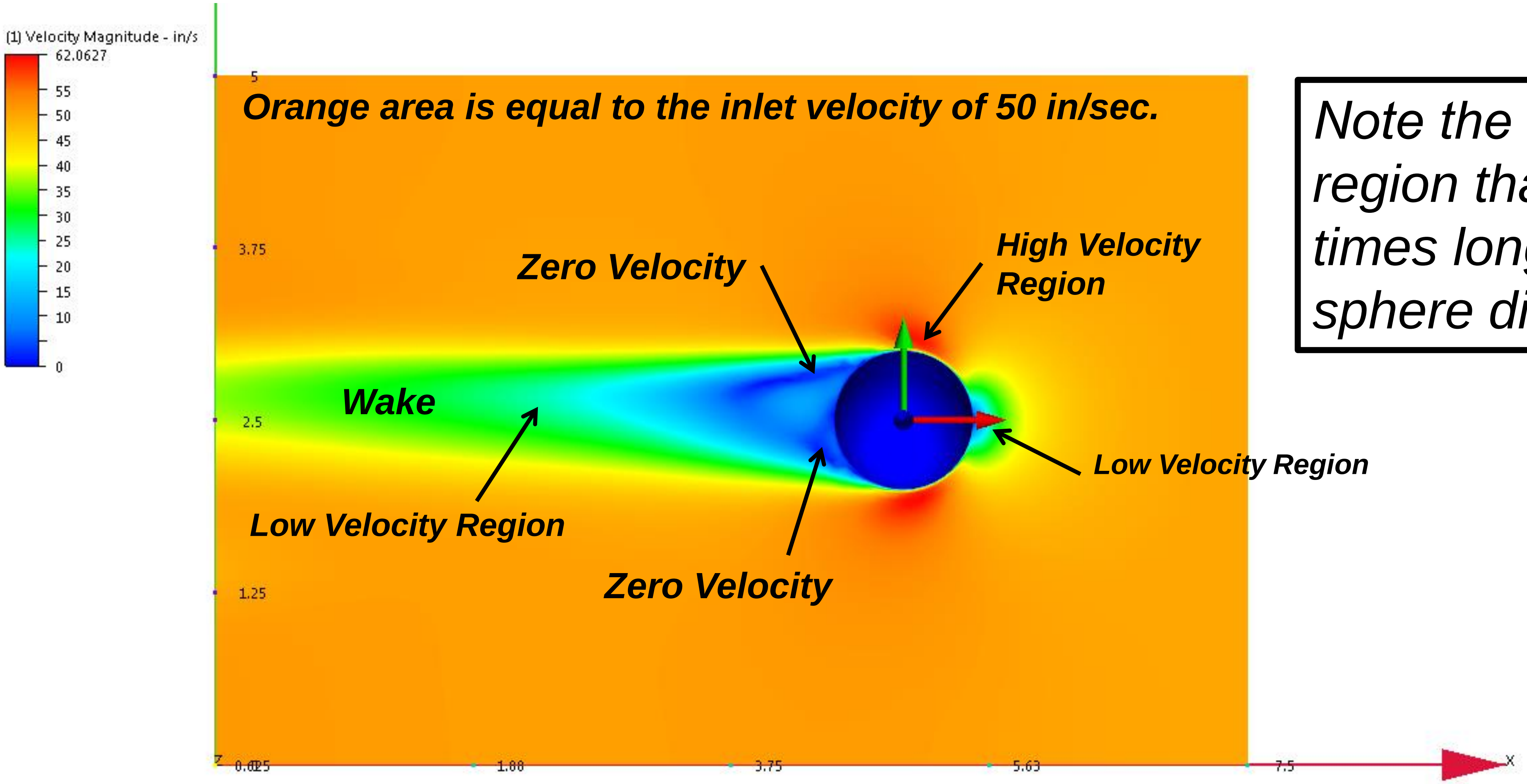
Input Data

The data necessary to run the problem in Simulation CFD is shown in the figure.



Velocity Distribution

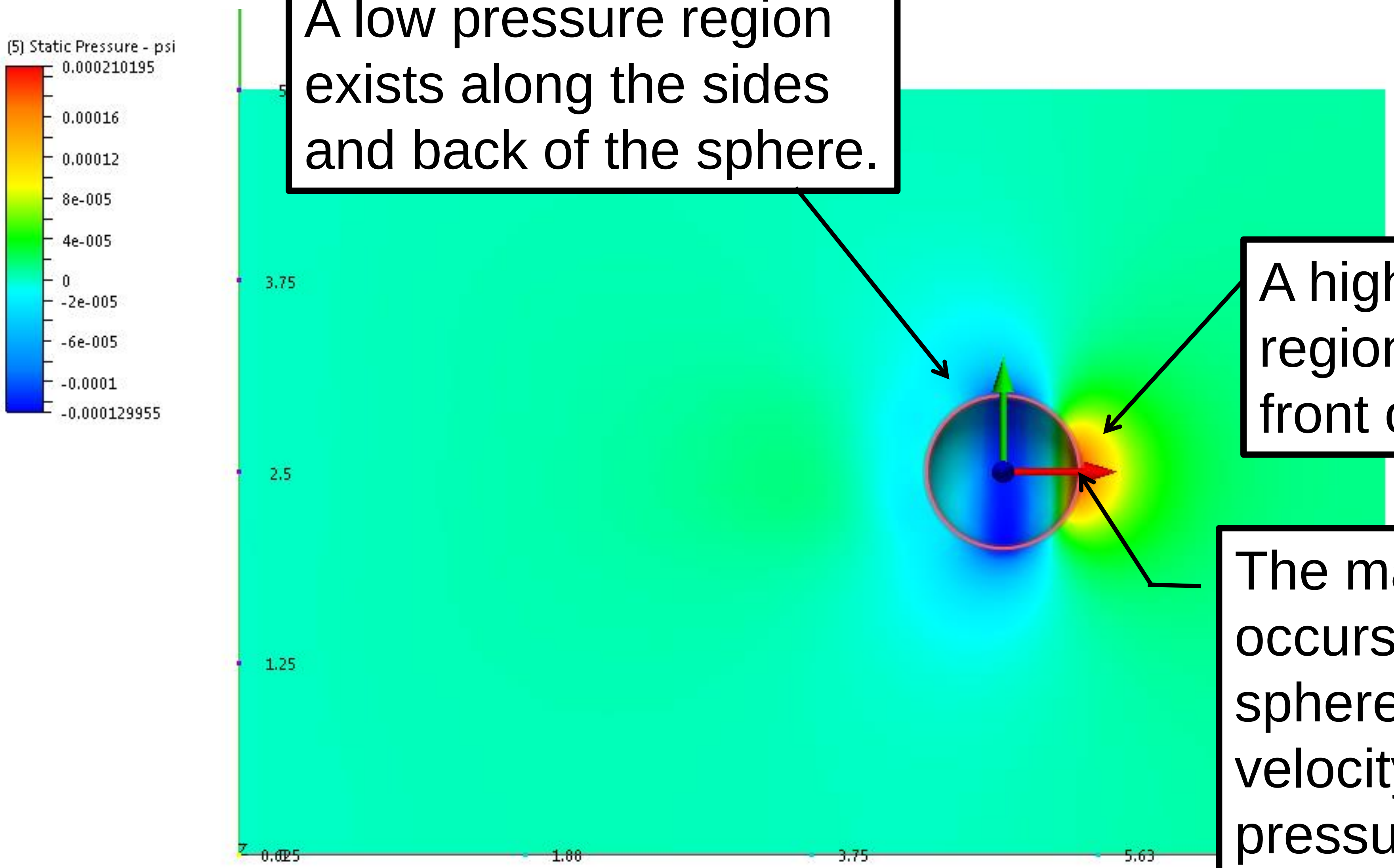
The velocity distribution shows a low velocity region at the front of the sphere. The blue region in back of the sphere has areas of zero velocity. The flow speeds up along the sides of the sphere.



Note the long "wake" region that is many times longer than the sphere diameter.

Pressure Distribution

The unequal pressure distribution causes a net force (drag) on the sphere.



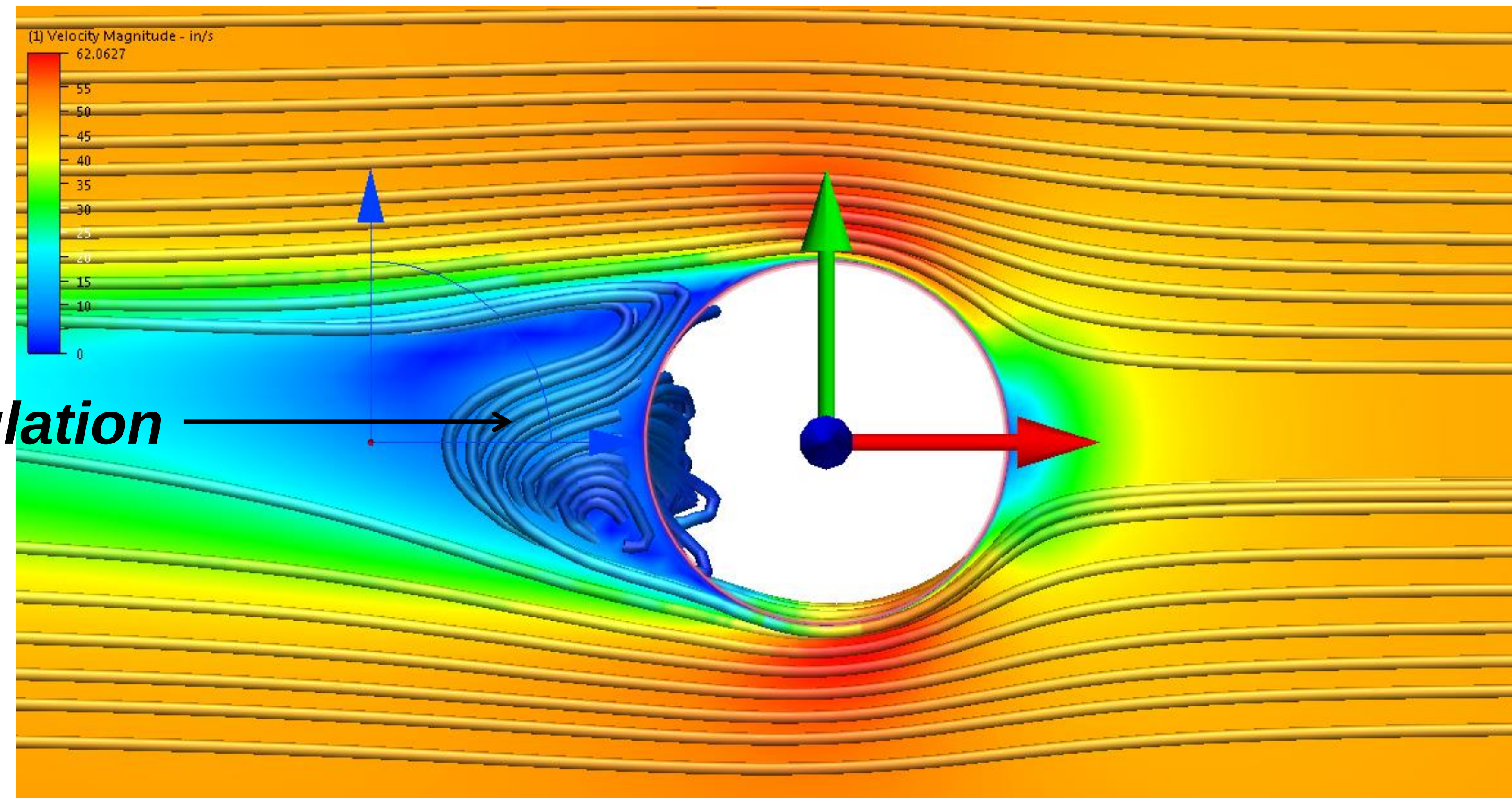
A low pressure region exists along the sides and back of the sphere.

A high pressure region exists at the front of the sphere.

The max pressure occurs at the tip of the sphere. At this point, velocity is zero and the pressure is called the stagnation pressure.

Recirculation

Particle traces show a recirculation region on the back-side of the sphere. The recirculation is 3-dimensional and wraps around the side of the sphere.

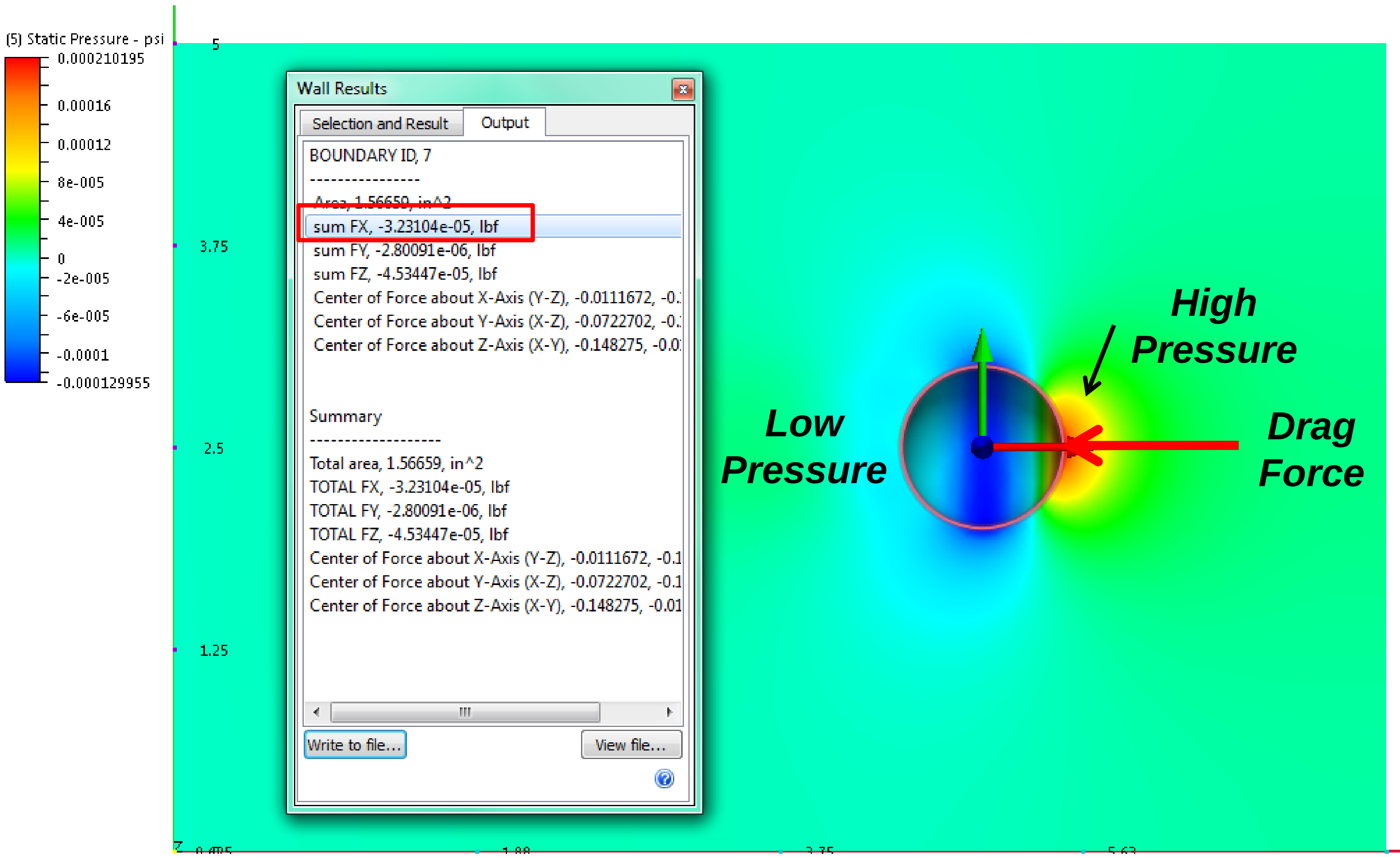


Recirculation or a vortex on the back side of a bluff (not streamlined) body is common at higher Reynolds numbers.

The vortex is associated with boundary layer separation.

Drag Force

The computed force in the direction of the flow is 3.23x10-5 lb_f. This is the force acting on half of the sphere. The force for both halves of the sphere is twice this value or 6.46x10-5 lb_f.



The force acting on the sphere is obtained by integrating the pressure over the sphere using the Wall Calculator found in Autodesk Simulation CFD.

Drag Coefficient

The drag coefficient based on the computed force agrees reasonably well with the experimental results shown in the figure. Better agreement could be obtained by adjusting the turbulence and mesh parameters used in the computations.

Drag Coefficient Calculations

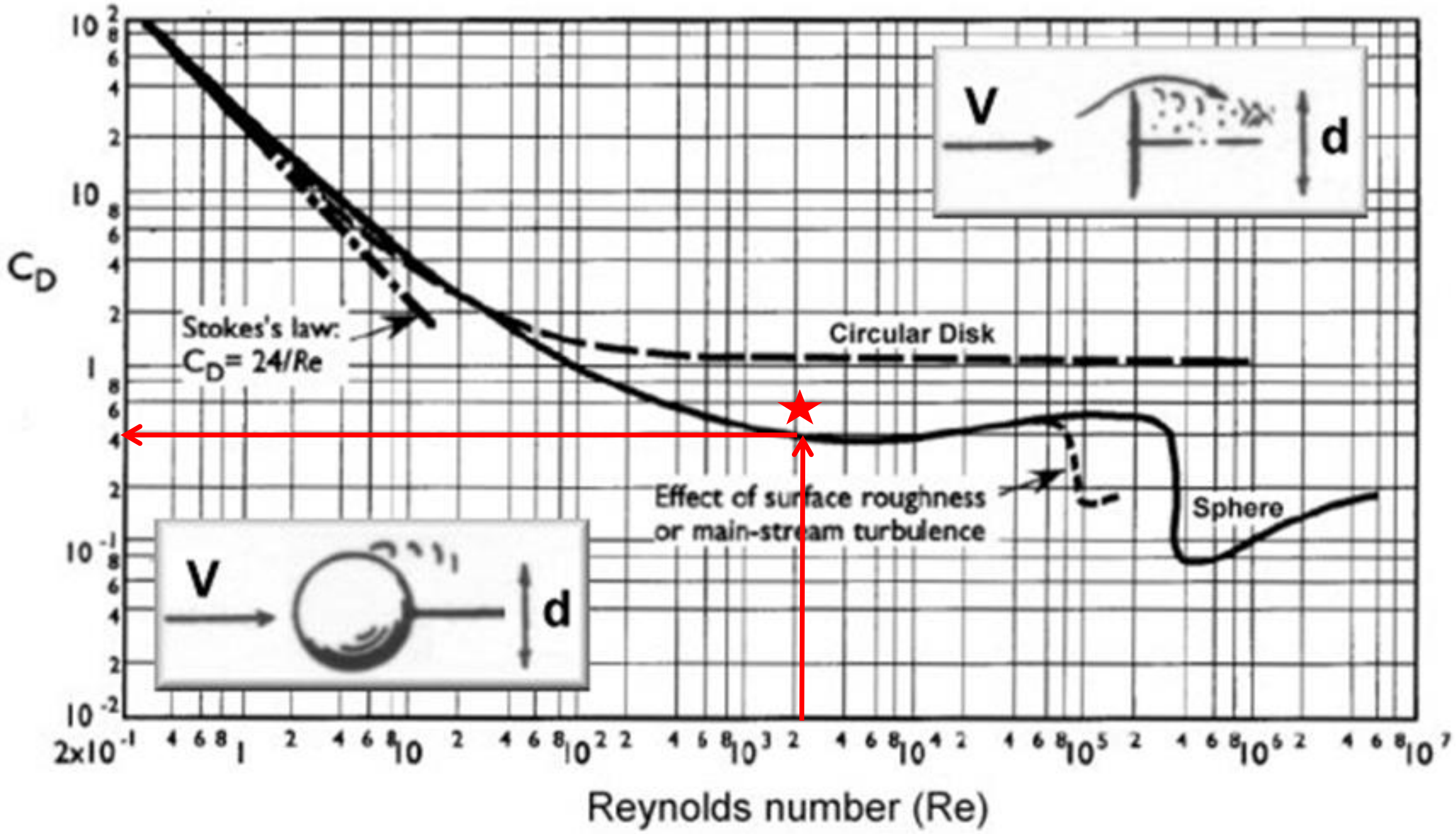
$$Re_D = \frac{V \cdot D}{\nu} = \frac{\left(50\frac{in}{\sec}\right)(1in)}{0.0236\frac{in^2}{\sec}} = 2{,}120$$

$$A = \frac{\pi \cdot D^2}{4} = \frac{\pi \cdot (1in)^2}{4} = 0.785in^2$$

$$C_d = \frac{2 \cdot F}{\rho \cdot V^2 \cdot A}$$

$$= \frac{2 \cdot 6.46x10^{-5}lbf}{\left(1.11x10^{-7}\frac{lbf-\sec^2}{in^3 \cdot in}\right)\left(50\frac{in}{\sec}\right)^2\left(0.785in^2\right)}$$

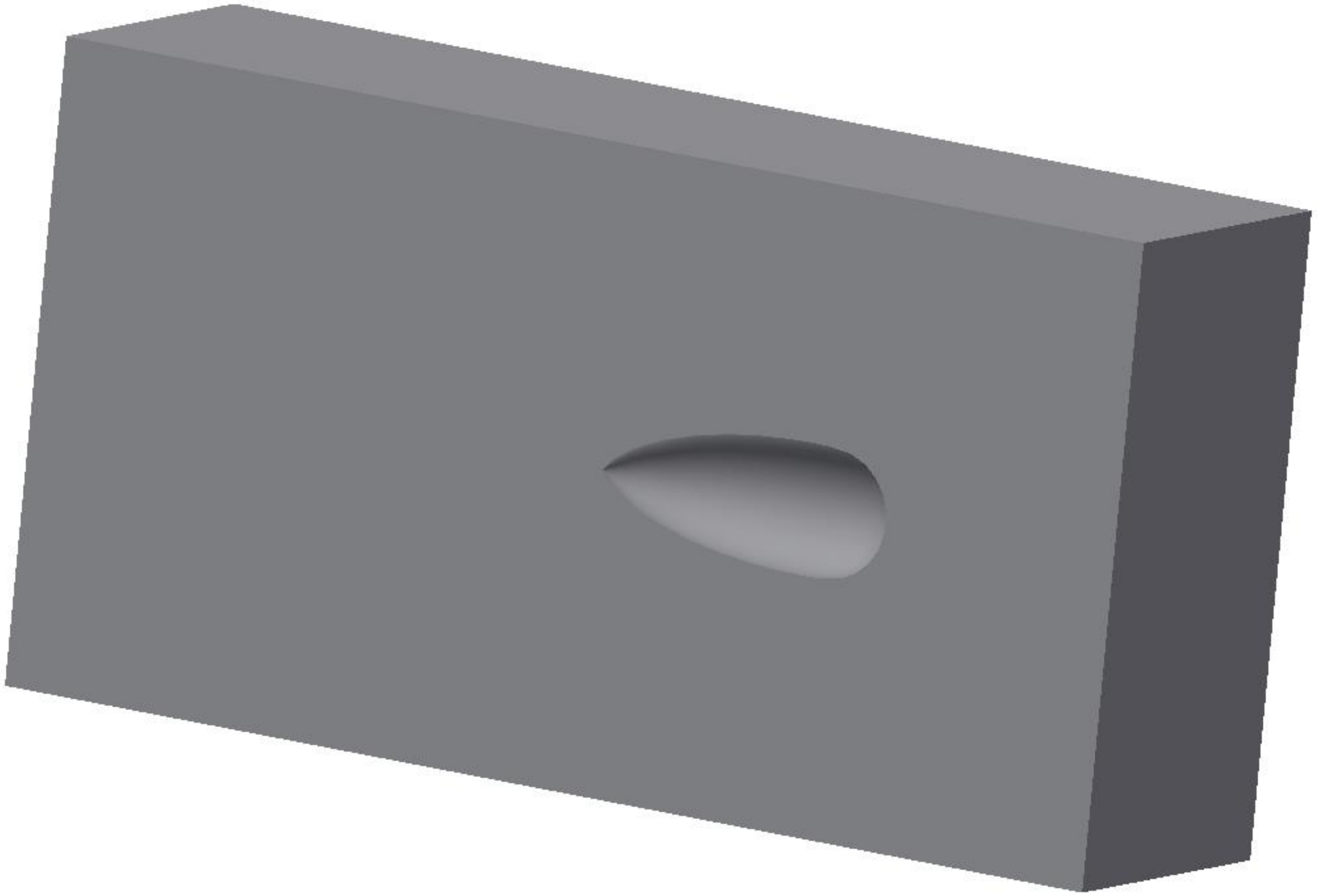
$$= 0.593$$



http://www.aerospaceweb.org/question/aerodynamics/q0231.shtml

Example - Streamlined Geometry

A similar problem was analyzed using Autodesk Simulation CFD, except a streamlined tail was added to the back of the sphere.

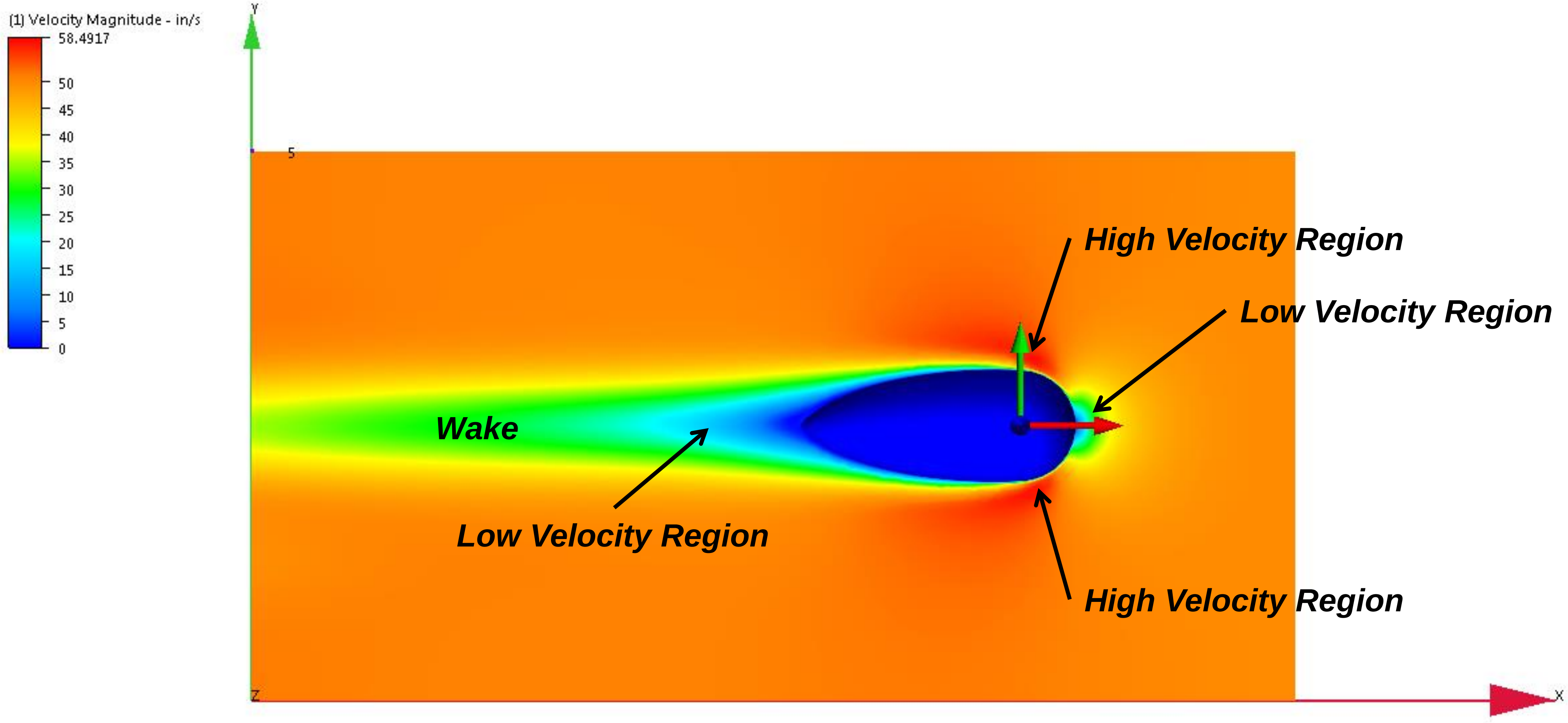


The inlet velocity, projected area, and air properties are the same as those used in the sphere example.

The Reynolds number is also the same.

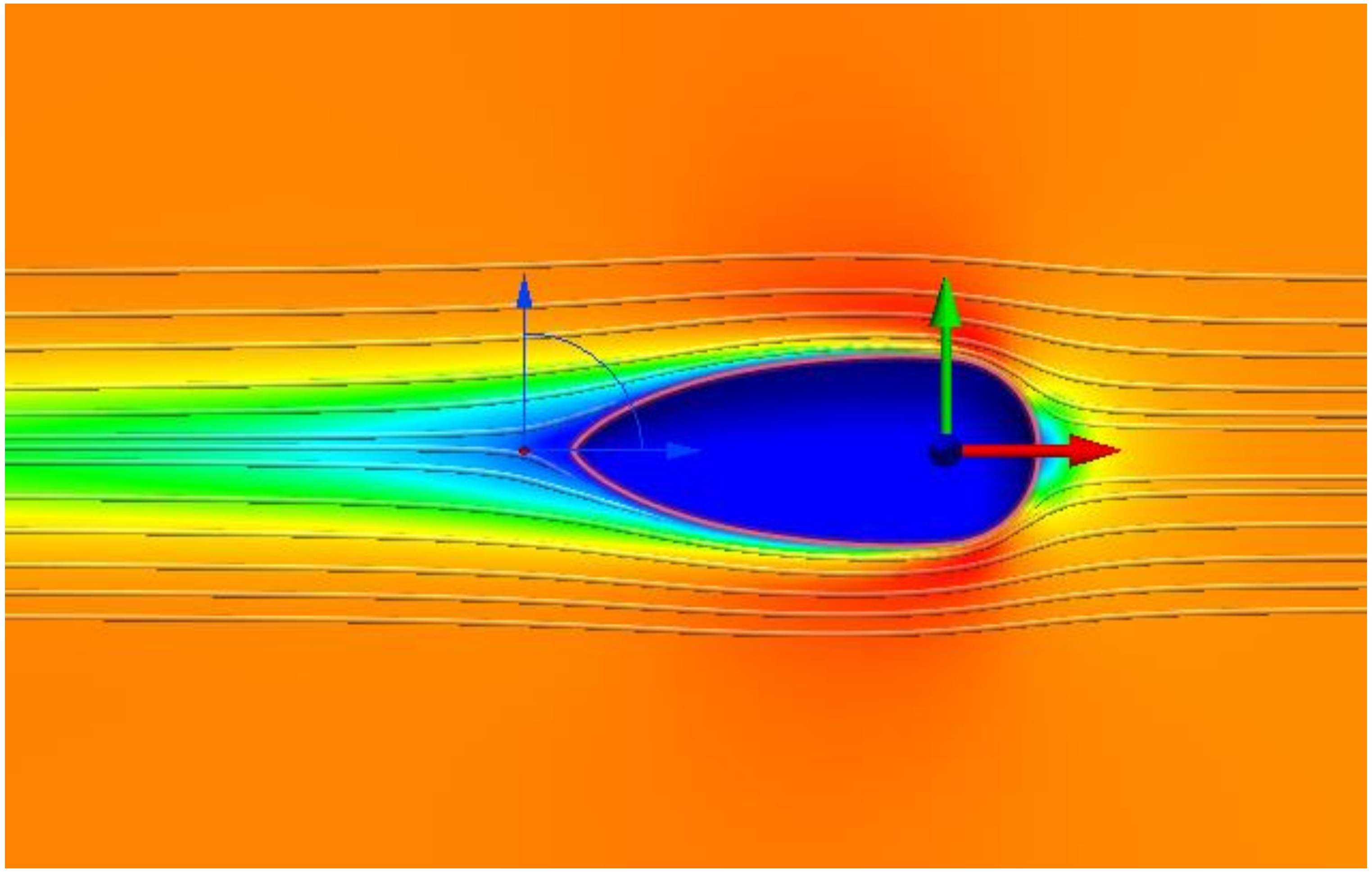
Velocity Distribution

The velocity distribution shows a low velocity region at the front of the sphere. The flow speeds up along the sides of the sphere. In many respects, this looks similar to the velocity distribution computed for the sphere.



Recirculation

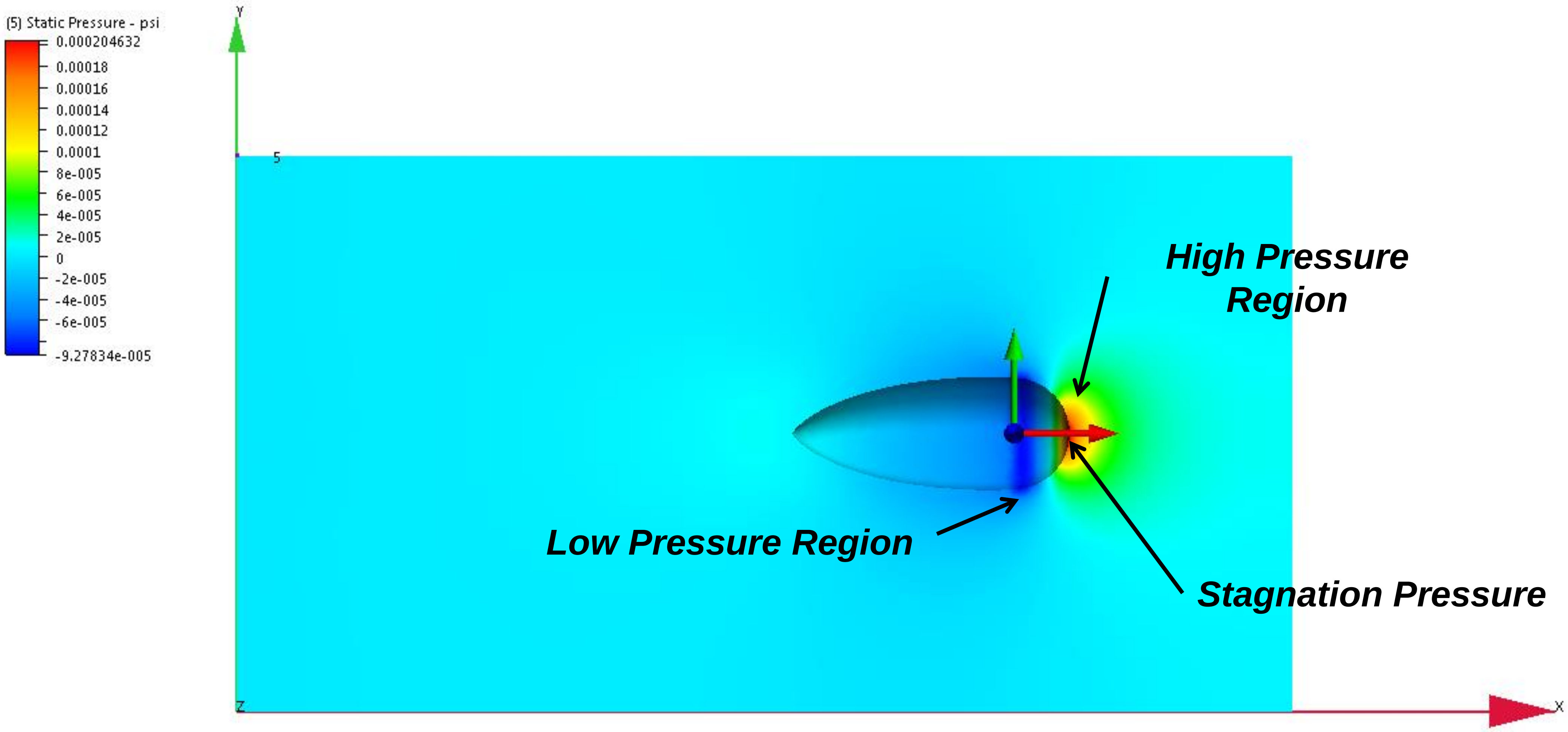
The appearance of the particle traces are much different looking than those for the sphere. Note the absence of the recirculation region at the end of the streamlined object. The particle traces follow the contours of the body much better than those associated with the sphere.



In this example, the flow is not having to expend energy to recirculate the flow at the back of the body, which makes it much more efficient.

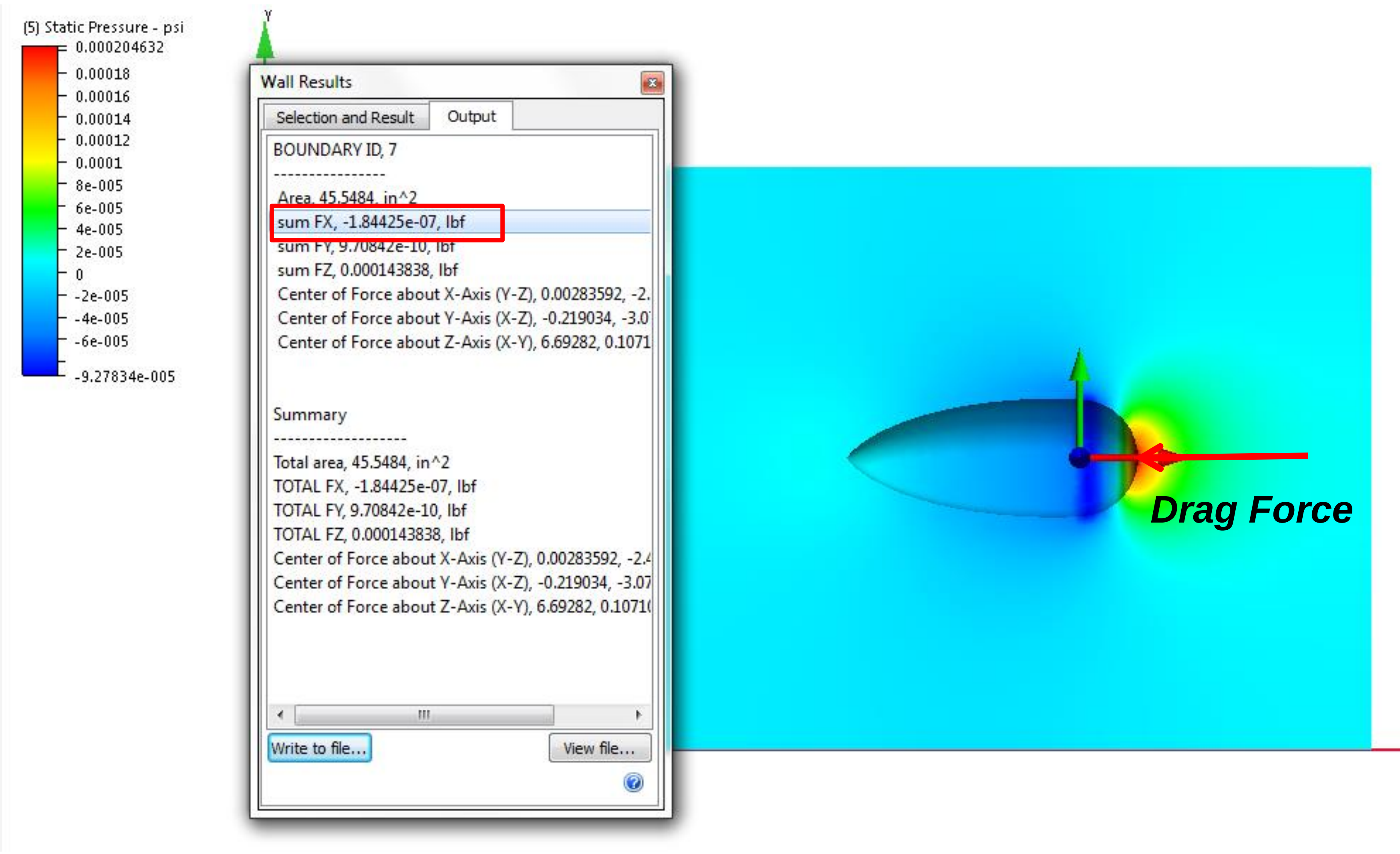
Pressure Distribution

The pressure distribution has many of the same characteristics as that for the sphere.



Drag Force

The computed force in the direction of the flow is 1.84×10^{-7} lb$_f$. This is the force acting on half of the object. The force for both halves of the sphere is twice this value or 3.68×10^{-7} lb$_f$.



The drag force of 3.68×10^{-7} lb$_f$ is much less than the drag force of 6.46×10^{-5} lb$_f$ computed for the sphere.

Drag Coefficient

The drag coefficient for the streamlined object is mush less than that for the sphere.

Sphere

$$\mathrm{Re}_D = \frac{V \cdot D}{\nu} = \frac{\left(50\,\frac{in}{\sec}\right)(1in)}{0.0236\,\frac{in^2}{\sec}} = 2{,}120$$

$$A = \frac{\pi \cdot D^2}{4} = \frac{\pi \cdot (1in)^2}{4} = 0.785in^2$$

$$C_d = \frac{2 \cdot F}{\rho \cdot V^2 \cdot A}$$

$$= \frac{2 \cdot 6.46x10^{-5}\,lbf}{\left(1.11x10^{-7}\,\frac{lbf - \sec^2}{in^3 \cdot in}\right)\left(50\,\frac{in}{\sec}\right)^2 \left(0.785in^2\right)}$$

$$= 0.593$$

Streamlined

$$\mathrm{Re}_D = \frac{V \cdot D}{\nu} = \frac{\left(50\,\frac{in}{\sec}\right)(1in)}{0.0236\,\frac{in^2}{\sec}} = 2{,}120$$

$$A = \frac{\pi \cdot D^2}{4} = \frac{\pi \cdot (1in)^2}{4} = 0.785in^2$$

$$C_d = \frac{2 \cdot F}{\rho \cdot V^2 \cdot A}$$

$$= \frac{2 \cdot 3.68x10^{-7}\,lbf}{\left(1.11x10^{-7}\,\frac{lbf - \sec^2}{in^3 \cdot in}\right)\left(50\,\frac{in}{\sec}\right)^2 \left(0.785in^2\right)}$$

$$= 0.033$$

Summary

- This module has introduced concepts associated with the drag force that acts on a body that moves relative to a fluid.
- Flow field phenomena associated with drag have been demonstrated using Autodesk Simulation CFD.
- The recirculation region on the back side of bluff bodies was shown to have a big influence on the drag force.
- The reduction in the Drag Coefficient achieved by streamlining the body was demonstrated.