Fluid Dynamics

CAx Tutorial: Flow Around a Cylinder

Guided Tutorial #1

Deryl O. Snyder
C. Greg Jensen

Brigham Young University
Provo, UT 84602

Special thanks to:

PACE, Fluent, UGS Solutions, Altair Engineering;

and to the following students who assisted in the creation of the Fluid Dynamics tutorials:

Leslie Tanner, Cole Yarrington, Curtis Rands, Curtis Memory, and Stephen McQuay.

Flow Around a Cylinder

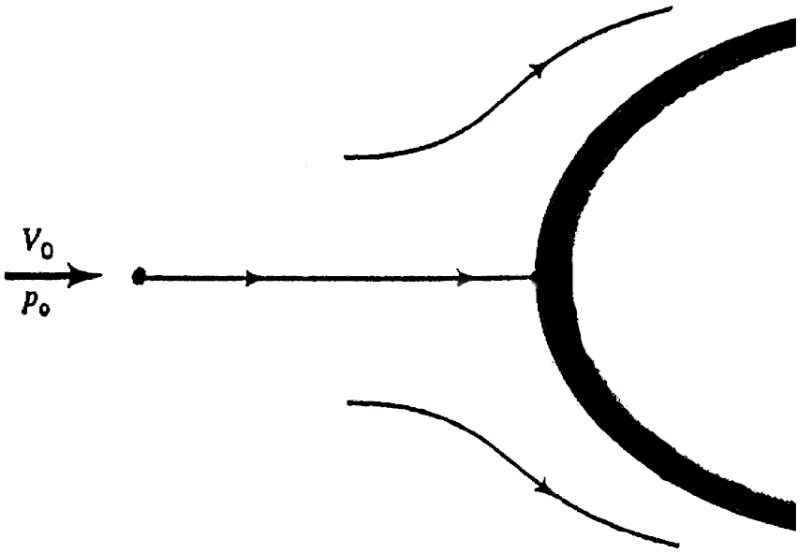
2D Flow

In this tutorial, Gambit® is used to create and mesh the geometry. Once preprocessing is complete, Fluent will be used to solve the flow problem. Fluent will then be used to analyze the solution.

This basic 2D tutorial should prepare you to model, mesh, define and analyze more complex fluid flow problems.

The methods expressed in these tutorials represent just one approach to modeling, defining and solving 2D problems. Our goal is the education of students in the use of CAx tools for modeling, constraining and solving fluids application problems. Other techniques and methods will be used and introduced in subsequent tutorials.

Water flows around a cylinder of radius 0.1 m at a Reynolds number of 20. Solve for pressure and viscous drag coefficients and display the vortcies in the wake of the cylinder.



Flow Around a Cylinder

Creating Geometry

Begin the problem by creating geometry in **Gambit**.

The computational domain will be a rectangle encompasing the 2-D cylinder in the center. At a Reynolds number of 20, vortex shedding will be present. It will be necessary to create a mesh with higher resolution in the wake of the cylinder to better resolve these vorticies.

First create the points that will be used to create the lines and then faces of the domain.

Operation > Geometry > Vertex > Create Vertex

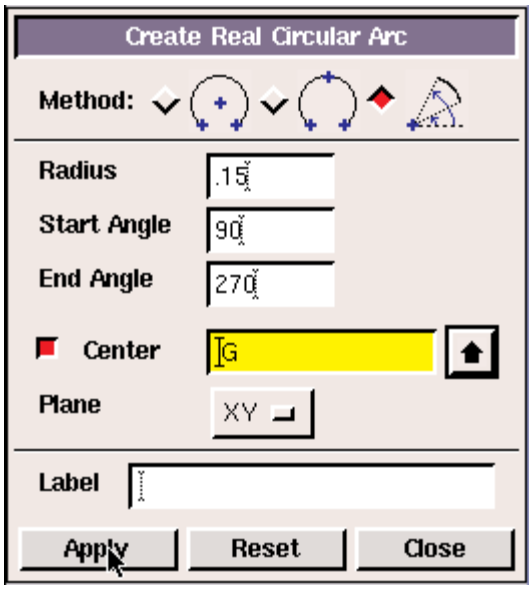
Name	X Position	Y Position
A	-2	-1.5
B	2	-1.5
C	2	1.5
D	-2	1.5
E	2	-0.15
F	2	0.15
G	0	0

Create an arc centered at point **G**. Use the third option under **Method** to specify a center point and a sweep angle. Create the arc with the following dimensions:

Radius=0.15

Start Angle=90

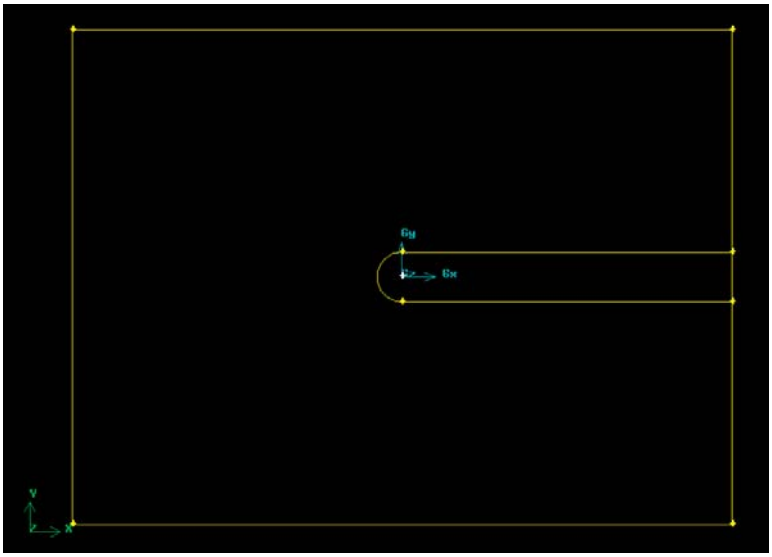
End Angle=270



Flow Around a Cylinder

Creating Geometry

Now draw the straight lines that will complete the domain. Connect the points to create the following line segments: **AB, BE, EF, FC, CD, DA,** and the horizontal lines connecting the new verticies created at the ends of the arc and the points **F and E**.



Create two faces: a small slender face stretching from the center of the domain to the far right edge of the rectangle containing a fine mesh to resolve the wake and a larger face surrounding it that will contain a rougher mesh.

Operation > Geometry > Face > Form Face

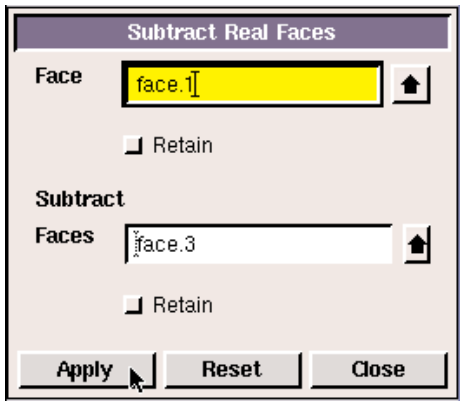
Select the four edges (including the arc) that make the long slender face and click **Apply**. Then create the larger face that consists of all the edges except the smallest vertical edge on the right vertical wall

Now make the cylinder by creating a circle face.

Operation > Geometry > Face > Create Face

Create a circle of **Radius=.05** and click **Apply**.
Now subtract the circle face from the slender face. Do not retain the faces themselves.

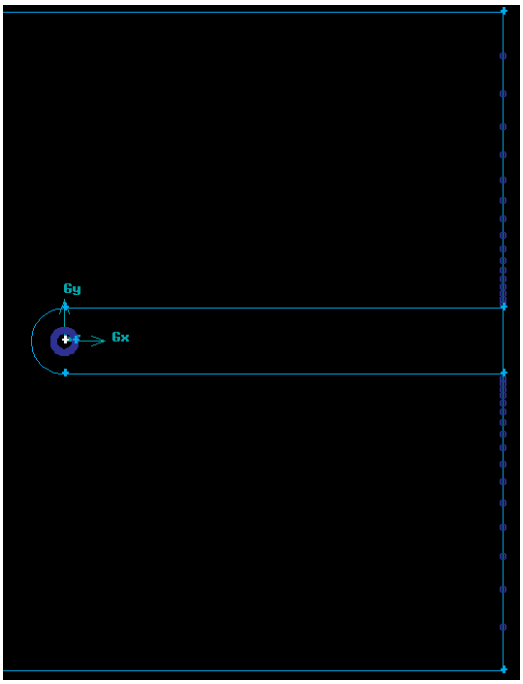
Operation > Geometry > Face > Boolean Operations



Flow Around a Cylinder

Meshing Geometry

Create edge meshes around the circle and right side vertical edges as shown to the right.



The circle edge should be meshed with a **Sucessive Ratio = 1** and an **Interval Count = 40**.

The two edges should be meshed with a **Last Length = 0.2** and an **Interval Count = 17**. The mesh should be finer moving closer to the X-axis.

Now make a boundary layer around the cylinder.

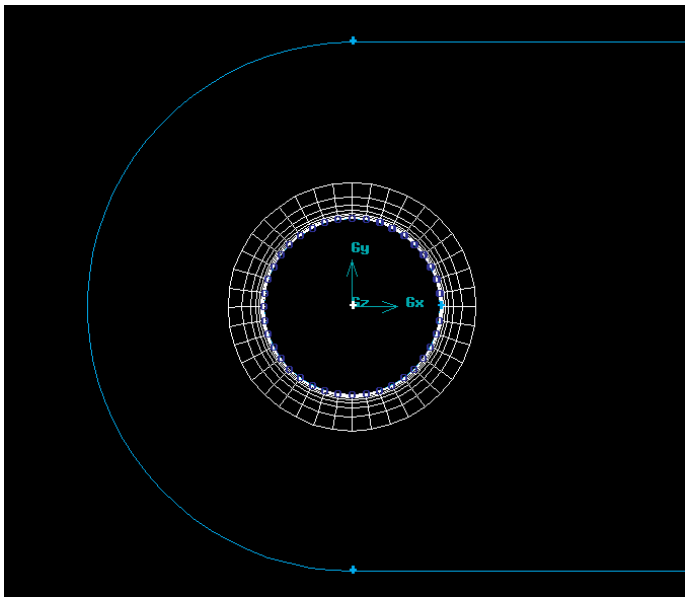
Operation > Mesh > Boundary Layer > Create Boundary Layer

Create the boundary layer with the following dimensions:

First Row = 0.0001

Growth Factor = 1.625

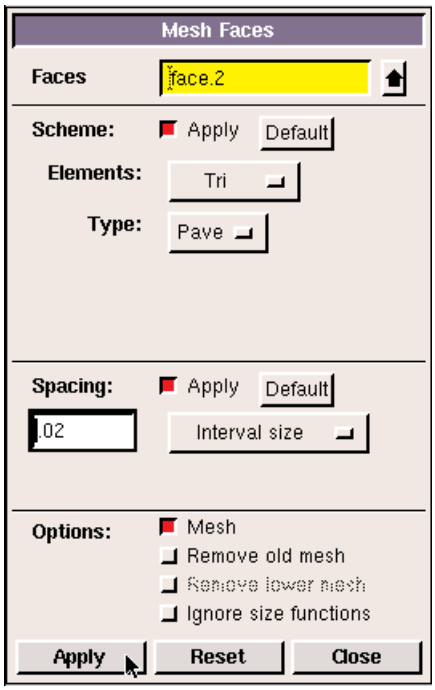
Rows = 10



Flow Around a Cylinder

Meshing Geometry

Mesh the two faces with **tri** elements.



Operation > Mesh > Mesh Face

The slender face should be meshed with an **Interval Size = 0.02**. The larger face with an **Interval Size = 0.2**. The mesh should resemble the image shown below.

Finally, specify the solver type and boundary conditions.

Solve > Fluent 5/6

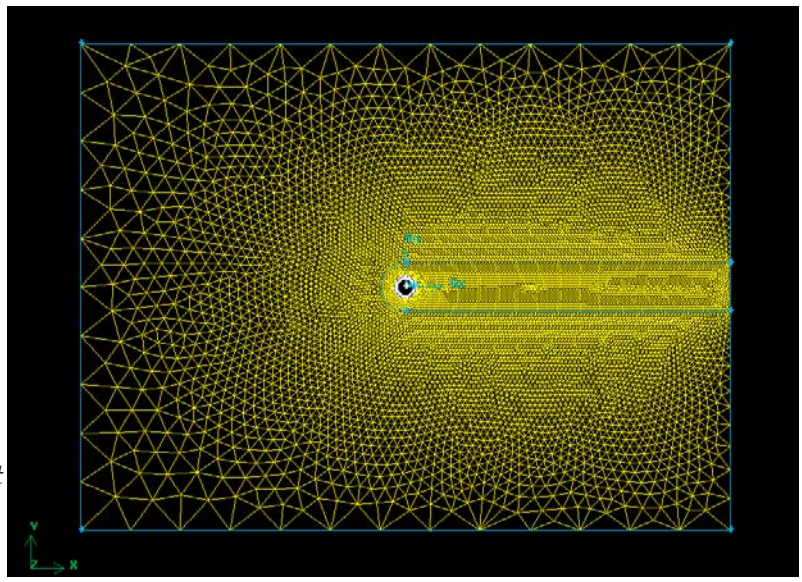
Operation > Zones > Specify Boundary Types

Create boundary conditions as follows:

Top, bottom, left vertical wall = **Velocity Inlet**

Cylinder = **Wall**

Right vertical wall (all three edges) = **Outflow**



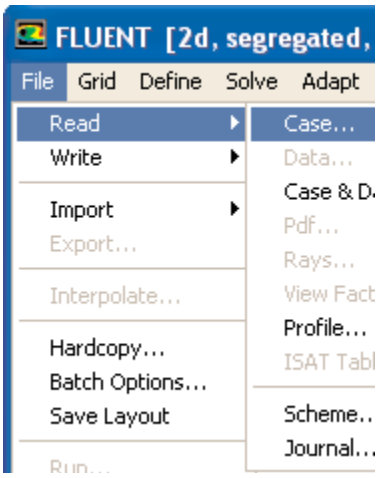
Save the Gambit file and export the Fluent mesh.

*If problems are encountered in creating the mesh, the completed mesh with boundary conditions specified can be loaded from the file "**Cylinder1_meshed.dbs**" into Gambit and "**Cylinder1_meshed.msh**" into Fluent*

Flow Around a Cylinder

Starting in Fluent

Load the mesh into Fluent.



File > Read > Case...

Check the mesh for errors.

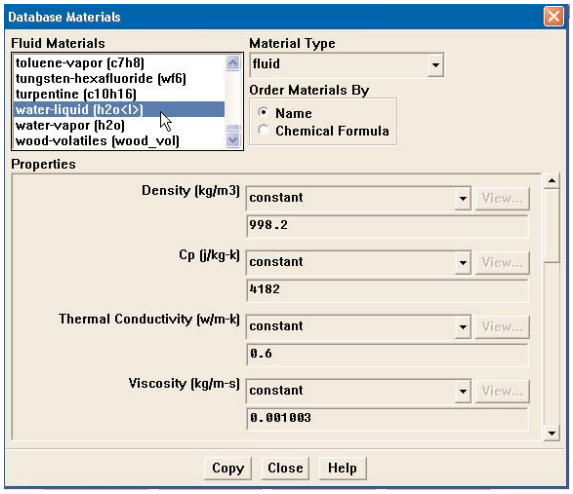
Grid > Check

For this problem, the default **Solver** settings will be sufficient. Ensure that the proper viscous model is selected.

Define > Models > Viscous...

Ensure that **Laminar** is selected.

Now recall liquid water from the materials database so that it can be specified in the boundary conditions.



Define > Materials...

Enter the database by clicking on **Database**. Select **water liquid (h2o<l>)** in the **Fluid Materials** list. Click **Copy** and then **Close**.

Now move the reference pressure into the flow domain.

Define > Operating Conditions...

Set the **Y** location to **1 m**.

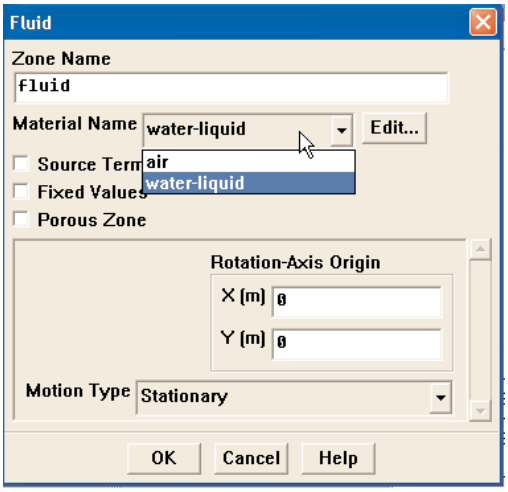
Flow Around a Cylinder

Defining the Problem

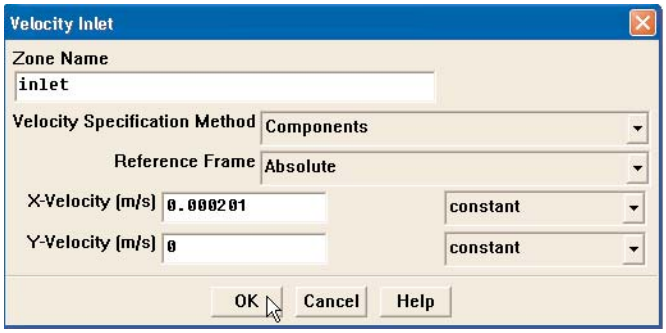
Boundary conditions can now be set.

Define > Boundary Conditions...

Select **fluid** in the selection menu on the left and then click on **Set...** Change **Material Name** to **water-liquid**.



Now click on **inlet** in the **Zones** menu and enter the **velocity-inlet** window. Change **Velocity Specification Method** to **Components** and enter a velocity of **2.01e-4** m/s (liquid water at Re = 20) next to **X-Velocity**.



Change the discretization method to a higher order scheme.

Solve > Controls > Solution...

Change the **Discretization** for **Momentum** to **2nd Order Upwind**.

The flow domain can now be initialized.

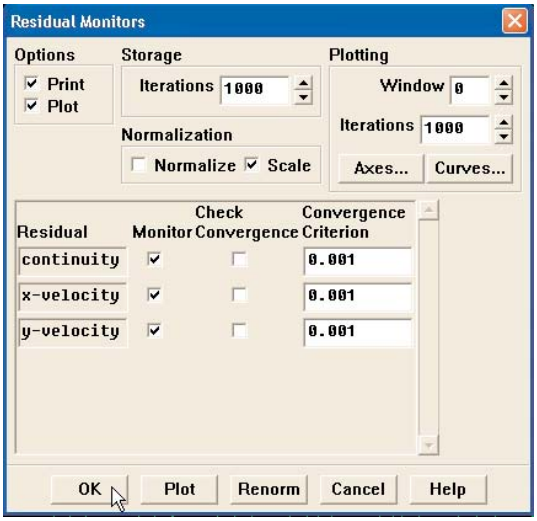
Solve > Initialize > Initialize...

Initialize the flow with the **inlet** conditions.

Flow Around a Cylinder

Solving the Problem

Enable the plotting option for residuals and turn off automatic convergence checking.



Solve > Monitors > Residual...

Now change the reference values Fluent uses to calculate coefficients and forces.

Report > Reference Values...

Modify the following fields with the corresponding values:

Area = 0.1 m^2

Density = 998.2 kg/m^3

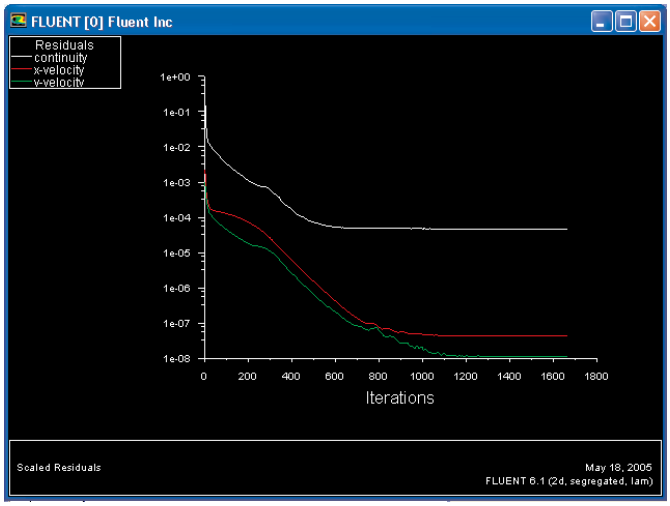
Velocity = 2.01e-4 m/s

Viscosity = 0.001003 kg/m-s

The problem is ready to be iterated.

Solve > Iterate...

Start with 2000 iterations.

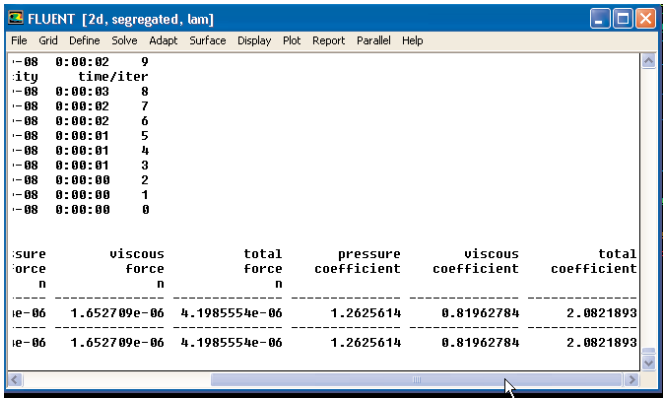


*If problems are encountered in setting up this problem in Fluent, the solved problem can be read in as a "**Case & Data...**" from the file "**Cylinder1_solved.cas**".*

Flow Around a Cylinder

Analyzing the Solution

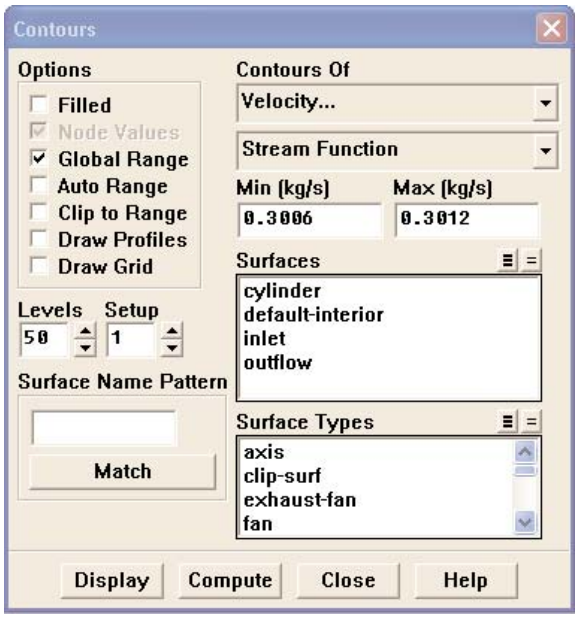
Once Fluent has stopped iterating, the forces and force coefficients on the cylinder can be calculated.



Report > Forces...

Select the **cylinder** under **Wall Zones** and then click **Print**. In the main Fluent window a readout of the forces on the cylinder should appear. Use the horizontal scroll bar to pan to the right. Fluent reports the **Pressure Coefficient** is **1.262** and the **Viscous Coefficient** is **0.818**. These values are in very good agreement with coefficeints obtained experimentally which were found to be **1.25** and **0.8** respectively.

Now display contours of stream functions.



Display > Contours...

Change **Contours Of** to **Velocity...** and select **Stream Function** in the pull-down menu just below.

Uncheck **Auto Range** at the top left of the **Contours** window.

Change **Levels** to **50**.

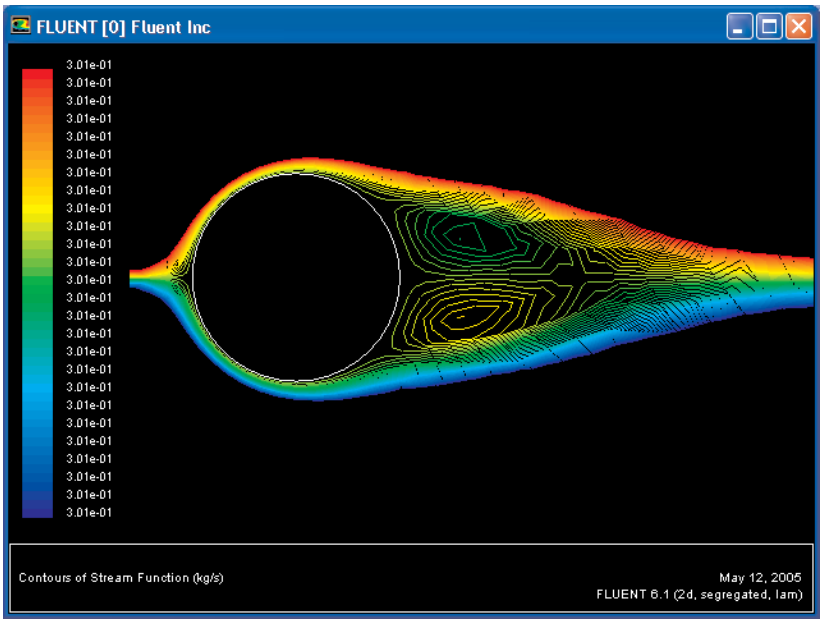
Enter **0.3006** in the **Min** field and **0.3012** in the **Max** field.

Click **Display**.

Flow Around a Cylinder

Analyzing the Results

Stream functions directly around the cylinder should be displayed. Zoom in on the cylinder. Two vorticies can be seen in the wake. If desired, re-display more contours by increasing the **Levels** in the **Contours** window. It can be seen that these stream lines are very similar to the experimental image of a cylinder at a similar Reynolds number (Re = 24.3) shown below.



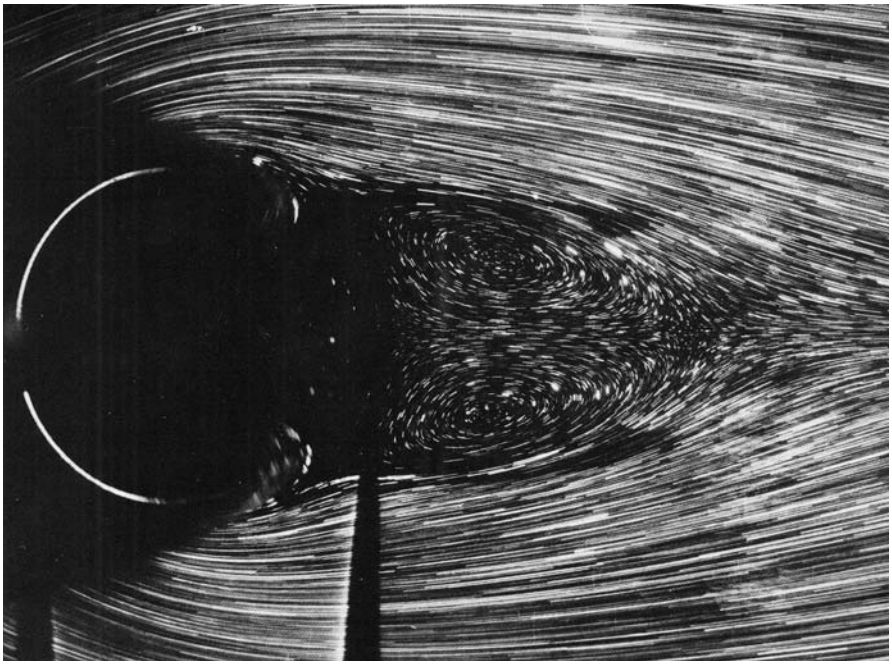


Photo by Coutanceau & Bouard 1977, from Van Dyke, Milton, *An Album of Fluid Motion*.