Created in COMSOL Multiphysics 5.3a

Lid-Driven Cavity

Introduction

The lid-driven cavity is a benchmark case for validating computational methods for fluid dynamics problems. The problem consists of a 2D square cavity in which the upper wall has a tangential velocity. This movement induces a flow characterized by a large vortex in the center of the cavity and smaller vortices in the corners. The magnitude of the Reynolds number affects the size and number of vortices in the flow. This model demonstrates how to define the boundary conditions for this problem in COMSOL Multiphysics. Additionally, it compares results for the velocity profile as well as the size and location of the vortices to a paper published by Ghia et al.

Model Definition

The lid-driven cavity problem is most elegantly modeled using a nondimensional form of the Navier-Stokes equations. Laminar Flow physics in COMSOL Multiphysics solve the traditional Navier-Stokes equations. For an incompressible stationary flow with no body forces, they are defined as:

$$\rho(\mathbf{u}\cdot\nabla)\mathbf{u} = -\nabla p + \mu\nabla^2\mathbf{u}$$

By nondimensionalizing the velocity ($\mathbf{u}^* = \frac{u}{\upsilon}$), pressure ($p^* = p/(\rho\upsilon^2)$), and length scale ($\mathbf{r}^* = \frac{r}{L}, \nabla^* = L\nabla$), the nondimensional Navier-Stokes equations can be written as:

$$(\mathbf{u}^*\cdot\nabla^*)\mathbf{u}^* = -\nabla p^* + \frac{1}{Re}\nabla^{*2}\mathbf{u}^*$$

where $Re = (\rho\upsilon L)/\mu$ is the Reynolds number.

The advantage to solving with the nondimensional form of the Navier-Stokes equations is that the flow can be characterized as a function of the Reynolds number only. Comparing the two forms of the Navier-Stokes equations, the values for the density and viscosity can be chosen appropriately such that the nondimensional form is solved in COMSOL Multiphysics.

The geometry consists of a square cavity with a side length of 1, which is the characteristic length scale for the flow. The density of the fluid is set to 1 while the viscosity is defined as $1/Re$.

For boundary conditions, the upper wall is prescribed as a moving wall with a horizontal velocity of 1. The remaining boundaries are considered to be no slip walls (zero velocity).

A pressure point constraint is used to create a well-defined problem. This condition is necessary for steady state analyses in closed systems since none of the boundary conditions fix the value of pressure in the domain.

A mapped mesh is applied with distributions such that more elements are stacked near the walls. This helps better resolve the boundary layer and corner vortices that appear in the flow. Higher mesh resolution near the walls is especially important when solving for higher Reynolds number flows. This meshing technique is an efficient way to discretize four-sided geometries while resolving the boundary layer.

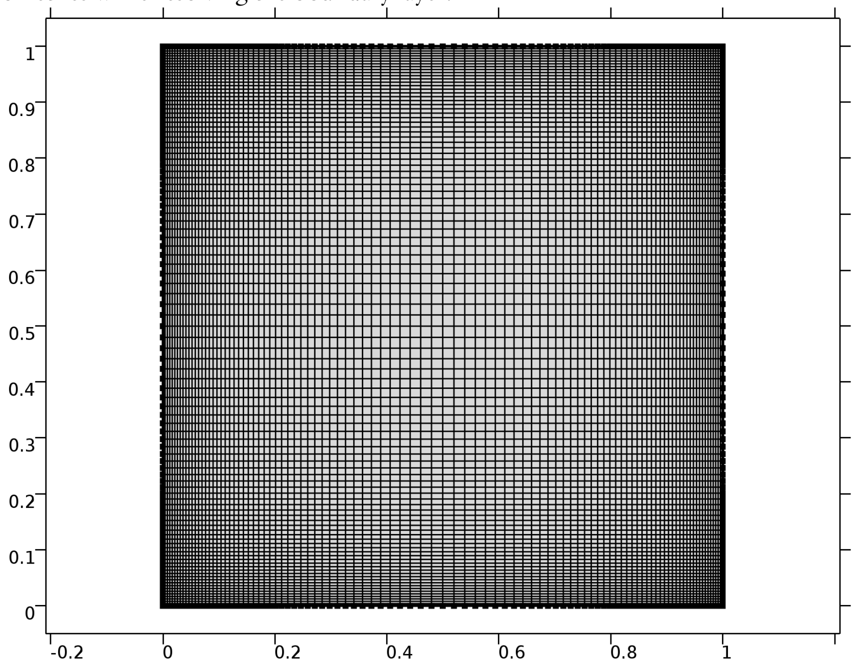


Figure 1: A mapped mesh with symmetric distributions efficiently resolves the high gradients in the boundary layer.

In the study, an auxiliary sweep is used to solve for a range of Reynolds numbers (100 to 10000). By using an auxiliary sweep, the solution for each parameter is solved then passed as an initial condition to the next parameter in the sequence to be solved, which speeds up the computation. This technique is also referred to as nonlinearity ramping, and it can be used to improve the convergence of highly nonlinear models.

The results are compared for each Reynolds number to the paper published by Ghia et al.

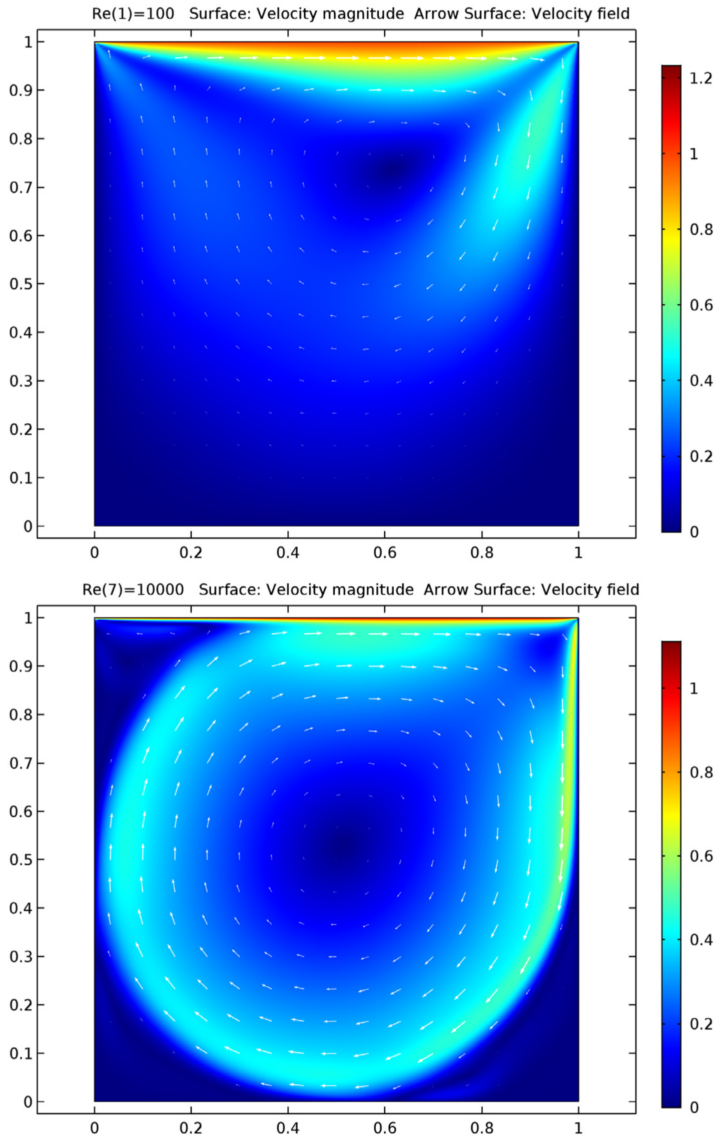


Figure 2: A large vortex forms in the center of the cavity. For the higher Reynolds number flow (Re =10,000), the vortex extends more prominently into the cavity due to the increased inertia.

[Figure 1](#) and [Figure 2](#) show the velocity profiles for a Reynolds number of 100 and 10000, respectively. In both cases, the fluid velocity approaches 1 near the top moving wall and zero near the no slip side and bottom walls. The central vortex rotates faster for Re = 10000 than for Re = 100 due to the increased inertia in the flow for the higher Reynolds number. Lower velocity regions appear in the bottom and left corners of the cavity where the secondary vortices are located.

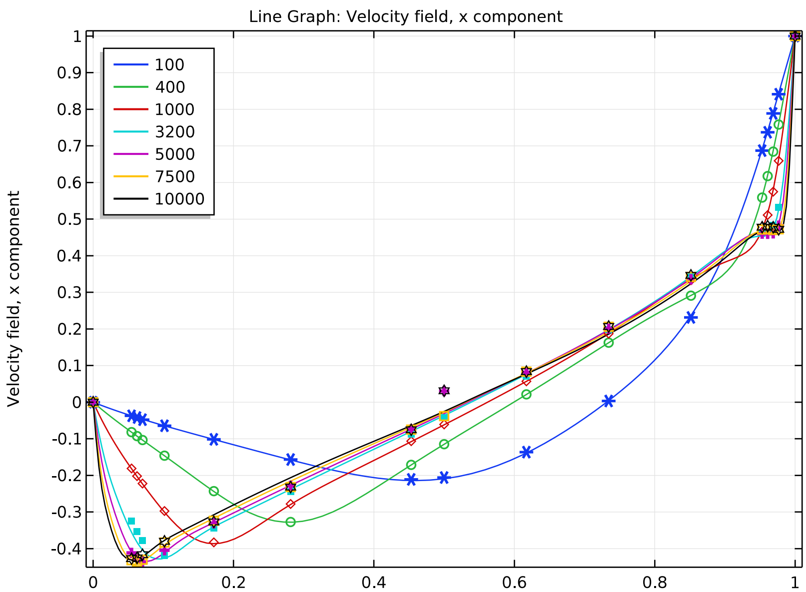


Figure 3: The velocity profile at the vertical centerline from Ghia et al matches closely with the COMSOL Multiphysics simulation results.

[Figure 3](#) plots the x component of velocity "u" versus the y location along a vertical line in the center of the cavity. At the bottom of the cavity (y=0), the no slip condition is satisfied (u=0). At the top of the cavity (y=1), the moving wall velocity is reached (u=1).

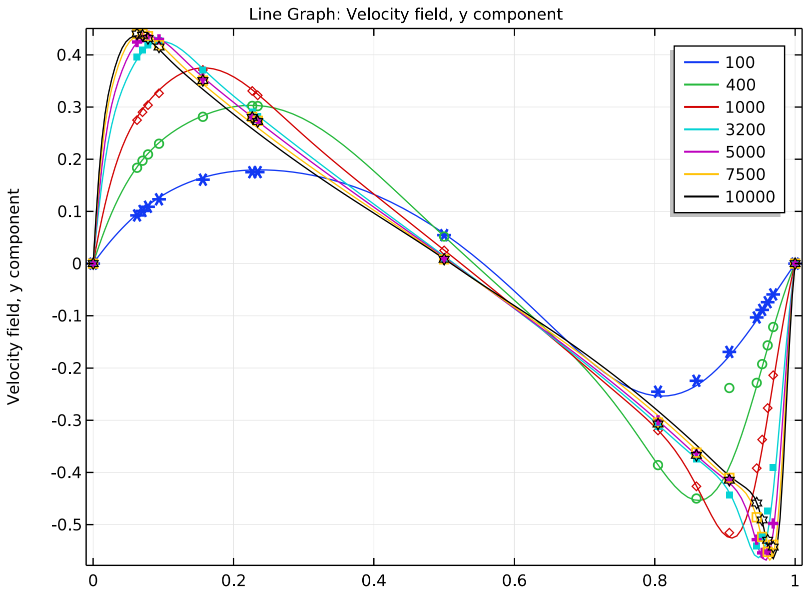


Figure 4: The velocity profile at the horizontal centerline from Ghia et al matches closely with the COMSOL Multiphysics simulation results.

Figure 4 plots the y component of velocity "v" versus the x location along a horizontal line in the center of the cavity. The no slip condition (v=0) is satisfied on the left and right walls (x=0, x=1). As the Reynolds number increases, the magnitude of the maximum velocities increases and the locations of the peak velocities shift closer to the walls.

In Figure 3 & Figure 4, the simulation results (solid line) match closely with the results generated by Ghia et al (data points) for the entire range of Reynolds numbers solved.

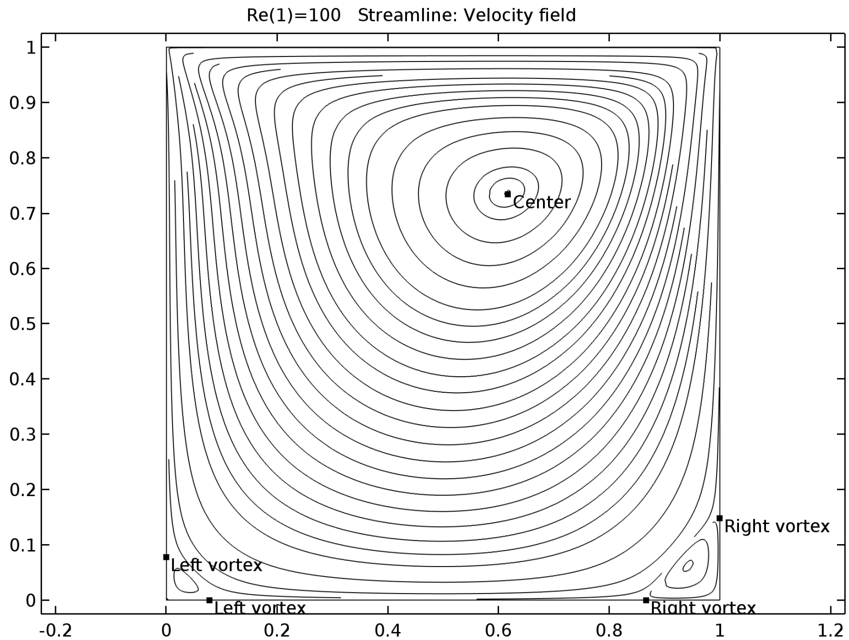


Figure 5: Two corner vortices appear in the flow for a Reynolds number of 100. Their placement, as well as the position of the center vortex, are in close agreement with the results published by Ghia et al.

Figure 5 plots streamlines for a Reynolds number of 100, which show the formation of a large central vortex and two smaller corner vortices. The central vortex spins clockwise, and due to separation near the corners, the two smaller corner vortices that spin counterclockwise are formed. The size of the corner vortices and the placement of the central vortex, indicated with annotations, are in close agreement with values generated by Ghia et al.

Reference

1. U. Ghia et al, "High-Re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method," Journal of Computational Physics, 48, 387-411, 1982.

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GEOMETRY 1

Set the global unit system to "No Units". This is the first step in defining the problem in non-dimensional form.

ROOT

1 In the **Model Builder** window, click the root node.

2 In the root node's **Settings** window, locate the **Unit System** section.

3 From the **Unit system** list, choose **None**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `lid_driven_cavity_parameters.txt`.

GEOMETRY 1

Square 1 (sq1)

On the **Geometry** toolbar, click **Primitives** and choose **Square**.

By setting the density to 1 and the viscosity to 1/Re, the non-dimensionalized Navier-Stokes equations will be solved. The Reynolds number "Re" will be changed dynamically in **Study 1** using an auxiliary sweep.

MATERIALS

Material 1 (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, locate the **Material Contents** section.
3. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	1		Basic
Dynamic viscosity	mu	1/Re		Basic

LAMINAR FLOW (SPF)

Wall 2

1. On the **Physics** toolbar, click **Boundaries** and choose **Wall**.
2. Select Boundary 3 only.
3. In the **Settings** window for **Wall**, click to expand the **Wall Movement** section.
4. Select the **Sliding wall** check box.
5. In the U_{w} text field, type 1.

A pressure point constraint is necessary in the closed system when performing stationary analysis in order to create a well-defined problem. It is applied in the corner far away from high gradients. By assigning a relative pressure of p=0, the absolute pressures in the cavity will be similar to the reference pressure level of 1 atm defined under **Laminar Flow** .

Pressure Point Constraint 1

1. On the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.
2. Select Point 1 only.

MESH 1

Mapped meshing can be used in some cases to efficiently discretize four sided geometries. It also allows the user to control the distribution of elements in order to better resolve the high gradients near the no slip walls. In this case, a symmetric distribution with geometric spacing is used to place more elements near the sides of the cavity.

Distribution 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.

2 Right-click **Mapped 1** and choose **Distribution**.

3 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.

4 From the **Selection** list, choose **All boundaries**.

5 Locate the **Distribution** section. From the **Distribution properties** list, choose **Predefined distribution type**.

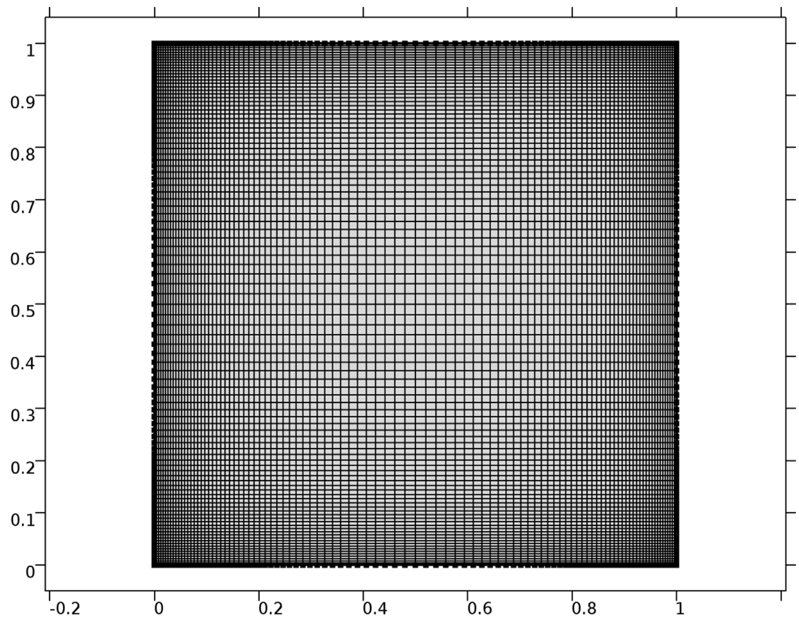
6 In the **Number of elements** text field, type 100.

7 In the **Element ratio** text field, type 5.

8 From the **Distribution method** list, choose **Geometric sequence**.

9 Select the **Symmetric distribution** check box.

10 Click **Build All**.



STUDY 1

Step 1: Stationary

By adding an auxiliary sweep, multiple values of the Reynolds number can be computed. An auxiliary sweep uses the solution to the previously computed parameter as an initial condition to solving for the current parameter, which speeds up the computation when solving for multiple parameters. This technique can also be used to improve the convergence of a simulation in a process known as nonlinearity ramping.

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 Click to select row number 1 in the table.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Re	100 400 1000 3200 5000 7500 10000	

7 On the **Home** toolbar, click **Compute**.

RESULTS

Velocity (spf)

1 In the **Model Builder** window, under **Results** click **Velocity (spf)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Parameter value (Re)** list, choose **100**.

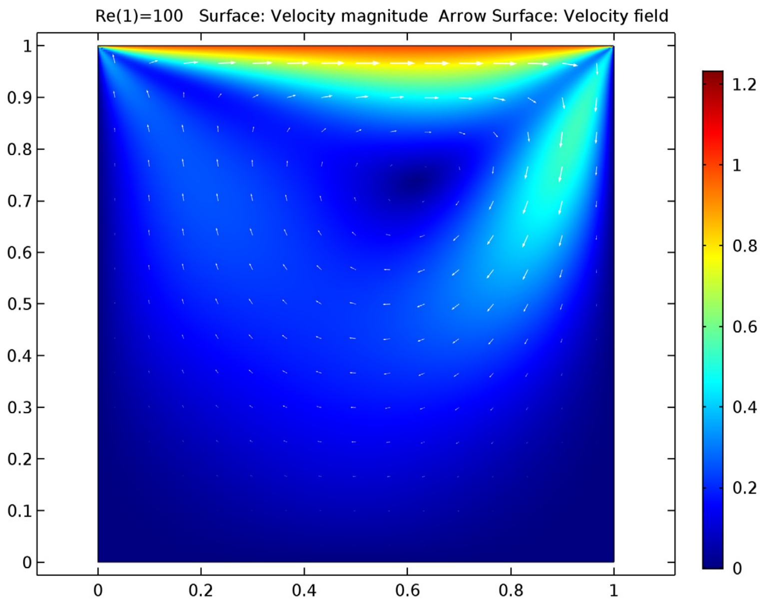
Arrow Surface 1

1 Right-click **Results>Velocity (spf)** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, locate the **Coloring and Style** section.

3 From the **Color** list, choose **White**.

4 On the **Velocity (spf)** toolbar, click **Plot**.



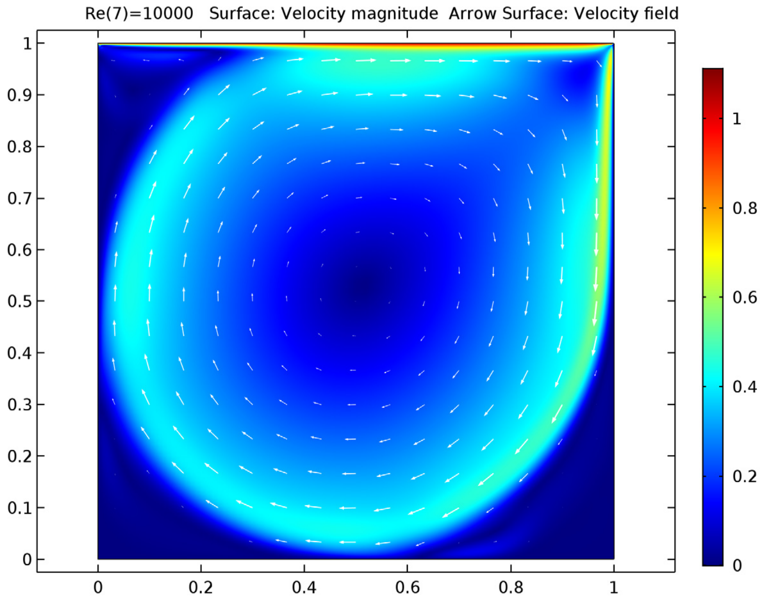
Velocity (spf)

1 In the **Model Builder** window, under **Results** click **Velocity (spf)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Parameter value (Re)** list, choose **10000**.

4 On the **Velocity (spf)** toolbar, click **Plot**.



Cut Line 2D 1

1. On the **Results** toolbar, click **Cut Line 2D**.
2. In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.
3. In row **Point 1**, set **x** to `.5`.
4. In row **Point 2**, set **x** to `.5`.
5. In row **Point 2**, set **y** to `1`.

Cut Line 2D 2

1. On the **Results** toolbar, click **Cut Line 2D**.
2. In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.
3. In row **Point 1**, set **y** to `.5`.
4. In row **Point 2**, set **y** to `.5`.

The data from Ghia et al is imported in order to compare it to the results from COMSOL.

Table 1

1. On the **Results** toolbar, click **Table**.
2. In the **Settings** window for **Table**, locate the **Data** section.

3 Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `lid_driven_cavity_literature1.txt`.

Table 2

1 On the **Results** toolbar, click **Table**.

2 In the **Settings** window for **Table**, locate the **Data** section.

3 Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `lid_driven_cavity_literature2.txt`.

The x-component of the velocity "u" is compared along the vertical centerline between the data from Ghia et al and the COMSOL results.

1D Plot Group 3

1 On the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `u vs y` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Cut Line 2D 1**.

4 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Line Graph 1

1 Right-click **u vs y** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `u`.

4 Click to expand the **Legends** section. Select the **Show legends** check box.

Table Graph 1

1 In the **Model Builder** window, under **Results** right-click **u vs y** and choose **Table Graph**.

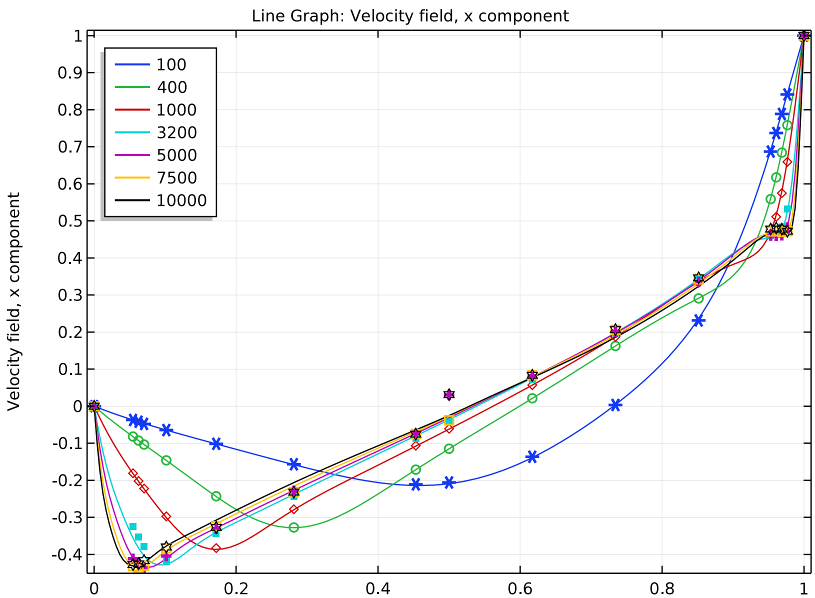
2 In the **Settings** window for **Table Graph**, locate the **Coloring and Style** section.

3 Find the **Line style** subsection. From the **Line** list, choose **None**.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

5 From the **Positioning** list, choose **In data points**.

u vs y



The y-component of the velocity "v" is compared along the horizontal centerline between the data from Ghia et al and the COMSOL results.

1D Plot Group 4

1. On the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `v vs x` in the **Label** text field.
3. Locate the **Data** section. From the **Data set** list, choose **Cut Line 2D 2**.

Line Graph 1

1. Right-click **v vs x** and choose **Line Graph**.
2. In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
3. In the **Expression** text field, type `v`.
4. Locate the **Legends** section. Select the **Show legends** check box.

Table Graph 1

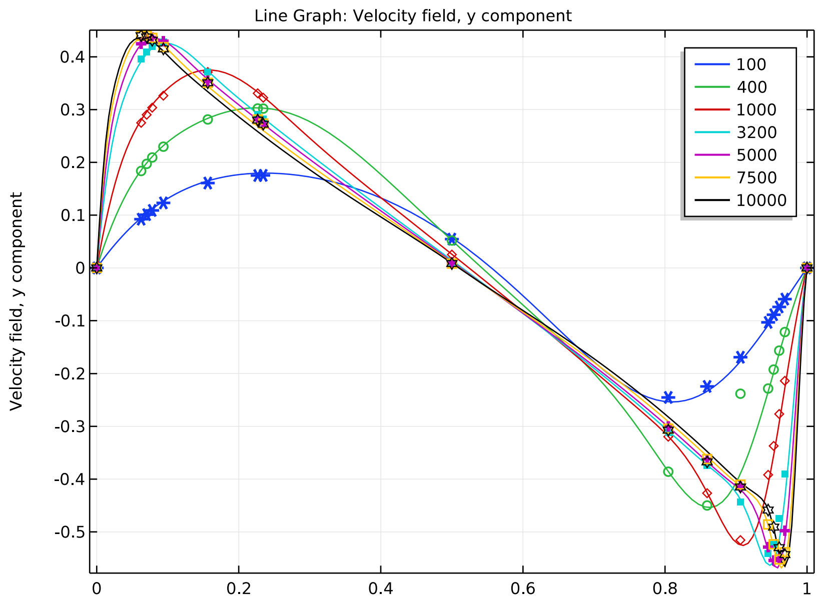
1. In the **Model Builder** window, under **Results** right-click **v vs x** and choose **Table Graph**.
2. In the **Settings** window for **Table Graph**, locate the **Data** section.
3. From the **Table** list, choose **Table 2**.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

6 From the **Positioning** list, choose **In data points**.

v vs x



2D Plot Group 5

1 On the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Streamline Plot` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (Re)** list, choose **100**.

Streamlines are plotted for a Reynolds number of 100, and the locations of the center and corner vortices are compared to those from Ghia et al using annotations.

Streamline 1

1 Right-click **Streamline Plot** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

3 From the **Positioning** list, choose **Uniform density**.

4 In the **Separating distance** text field, type `0.02`.

Annotation 1

1 In the **Model Builder** window, under **Results** right-click **Streamline Plot** and choose **Annotation**.

2 In the **Settings** window for **Annotation**, locate the **Annotation** section.

3 In the **Text** text field, type `Center`.

4 Locate the **Position** section. In the **x** text field, type `xC`.

5 In the **y** text field, type `yC`.

Annotation 2

1 Right-click **Streamline Plot** and choose **Annotation**.

2 In the **Settings** window for **Annotation**, locate the **Annotation** section.

3 In the **Text** text field, type `Right vortex`.

4 Locate the **Position** section. In the **x** text field, type `xR`.

Annotation 3

1 Right-click **Streamline Plot** and choose **Annotation**.

2 In the **Settings** window for **Annotation**, locate the **Annotation** section.

3 In the **Text** text field, type `Right vortex`.

4 Locate the **Position** section. In the **x** text field, type `1`.

5 In the **y** text field, type `yR`.

Annotation 4

1 Right-click **Streamline Plot** and choose **Annotation**.

2 In the **Settings** window for **Annotation**, locate the **Annotation** section.

3 In the **Text** text field, type `Left vortex`.

4 Locate the **Position** section. In the **y** text field, type `yL`.

Annotation 5

1 Right-click **Streamline Plot** and choose **Annotation**.

2 In the **Settings** window for **Annotation**, locate the **Annotation** section.

3 In the **Text** text field, type `Left vortex`.

4 Locate the **Position** section. In the **x** text field, type `xL`.

Streamline Plot