
Tutorial 1.

Flow in a Lid-Driven Cavity

Introduction

The purpose of this tutorial is to illustrate the setup and solution of the two-dimensional laminar fluid flow for a lid-driven cavity.

This tutorial demonstrates how to do the following:

- Read an existing mesh file in **FLUENT**.
- Verify the grid for dimensions and quality.
- Change the material properties.
- Carry out solver settings and perform iterations.
- Examine the results and compare them with experimental data.
- Display and create animation for pathlines.

Prerequisites

This tutorial assumes that you have little experience with **FLUENT** but are familiar with the interface.

Problem Description

The lid-driven cavity flow is probably one of the most studied fluid problem in the field of computational fluid dynamics. Lid-driven cavity flow retains a rich fluid flow physics manifested by multiple counter rotating recirculating regions on the corners of the cavity depending on the Reynolds number.

In this tutorial, we consider a square cavity with a height $H = 1$ m (Figure 1.1). The top wall is moving with a velocity of 1 m/s in X direction, while bottom and side walls are stationary. The flow is considered to be flow at $Re = 1000$.

Preparation

1. Copy the following files `cavity.msh`, `data-uv1.xy` and `data-vv1.xy` to your working folder.
2. Start the 2D double precision (2ddp) version of **FLUENT**.

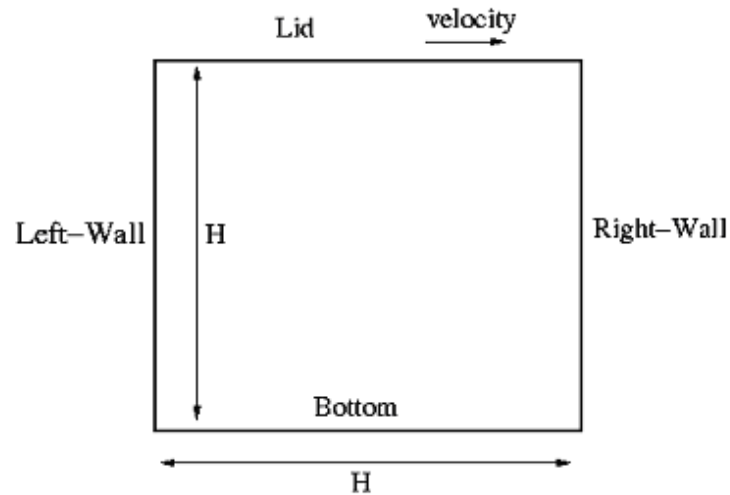


Figure 1.1: Problem Schematic

Setup and Solution

Step 1: Grid

1. Read the mesh file `cavity.msh`.

File → **Read** → Case...

FLUENT will read the mesh file and report the progress in the console window.

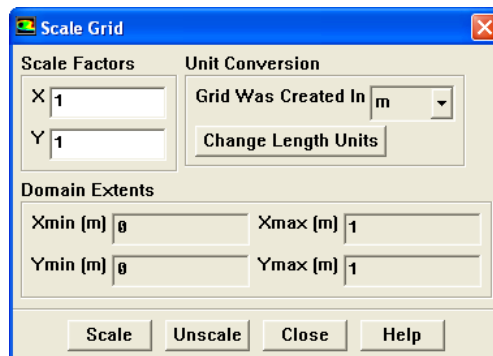
2. Check the grid.

Grid → Check

This procedure checks the integrity of the mesh. Make sure the reported minimum volume is a positive number.

3. Scale the grid.

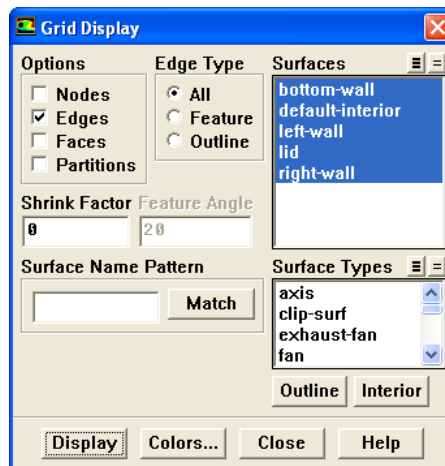
Grid → Scale...



Check the domain extents to see if they correspond to the actual physical dimensions. Otherwise the grid has to be scaled with proper units.

4. Display the grid (Figure 1.2).

Display → Grid...



- (a) Click Colors... in the Grid Display.



- i. Enable Color by ID in the Options list.
 - ii. Close the Grid Colors panel.
- (b) Click Display and close the Grid Display panel.

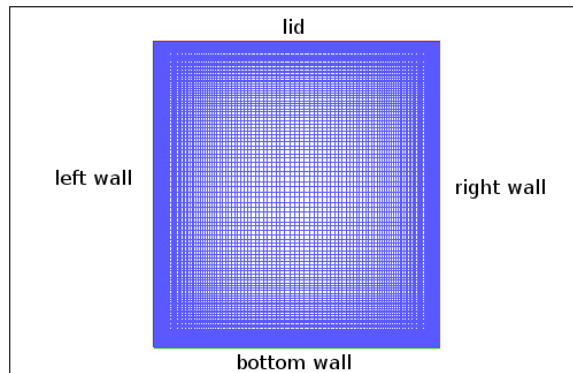


Figure 1.2: Grid Display

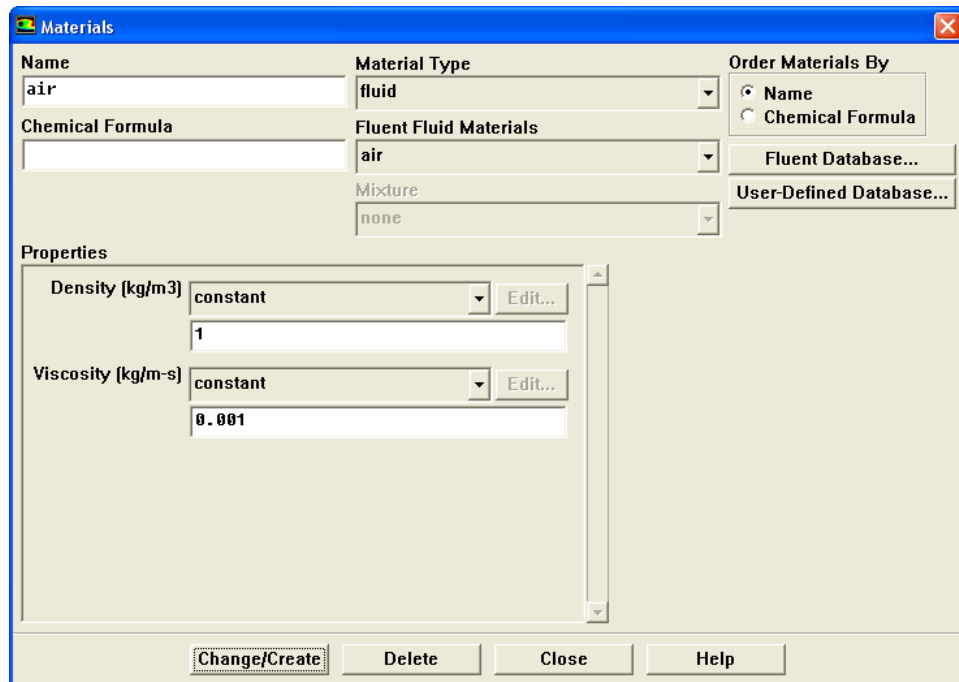
The grid adjacent to the walls is finer compared to that in the central region. The purpose of such fine mesh is to capture sharp gradients near the walls correctly.

Step 2: Models

Do not alter the default settings as the problem is to be solved in steady state with two dimensional laminar conditions.

Step 3: Materials

Define → Materials...



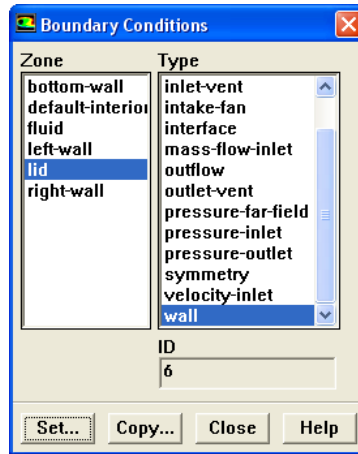
1. Enter 1 kg/m³ for Density.
2. Enter 0.001 kg/m-s for Viscosity.
3. Click Change/Create and close the Materials panel.

As Reynolds number is defined as $Re = \frac{\rho UH}{\mu}$, velocity will be set to 1 m/s to have $Re = 1000$.

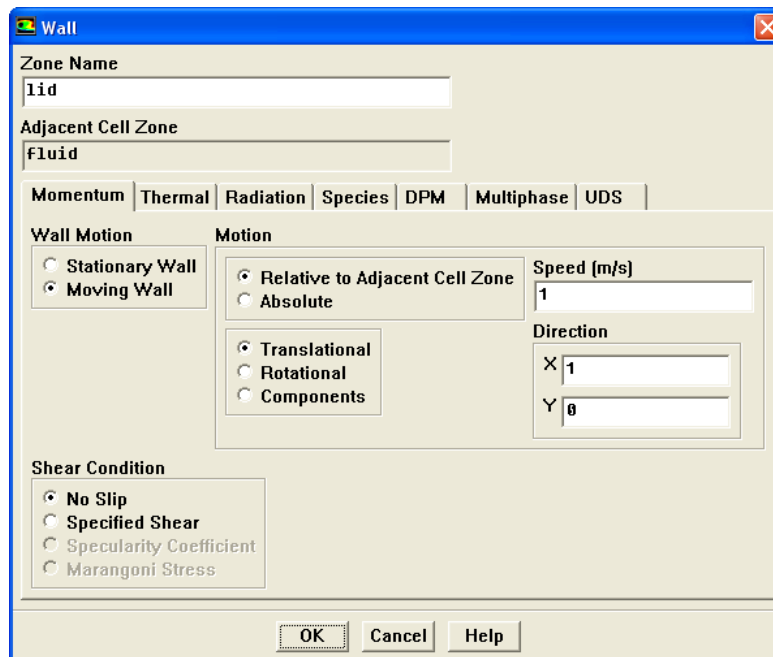
Step 4: Boundary Conditions

1. Set the boundary condition for lid.

Define → Boundary Conditions...



- (a) Select lid from Zone list.
- (b) Click Set... to open the Wall panel.

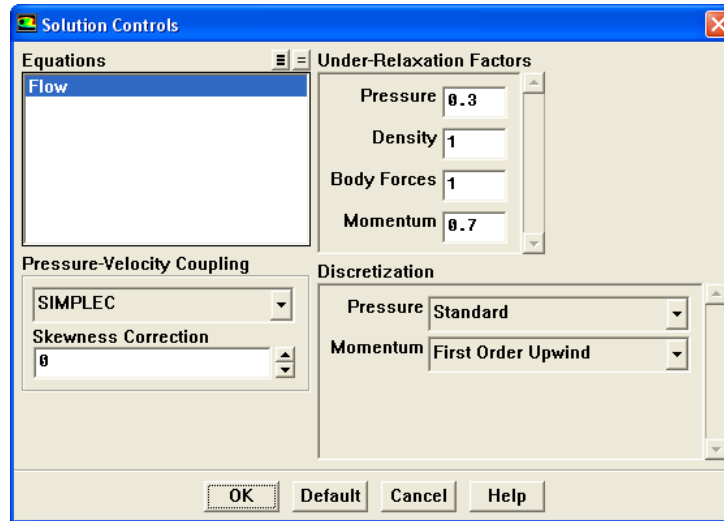


- (c) Click the Momentum tab.
 - (d) Select Moving Wall in the Wall Motion options.
 - (e) Enter 1 m/s for Speed.
 - (f) Click OK to close the Wall panel.
2. Close the Boundary Conditions panel.

Step 5: Solution

1. Set the solution controls.

Solve → Controls → Solution...

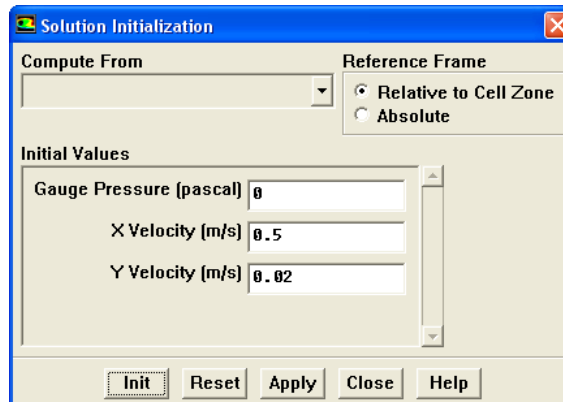


- (a) Select **SIMPLEC** from the Pressure-Velocity Coupling drop-down list.
- (b) Click **OK** to close the Solution Controls panel.

SIMPLEC is a better option for uncomplicated problems, where convergence depends on pressure-velocity coupling. In SIMPLEC, the pressure-correction under-relaxation factor is generally set to 1.0, which helps speed up convergence.

2. Initialize the flow.

Solve → Initialize → Initialize...



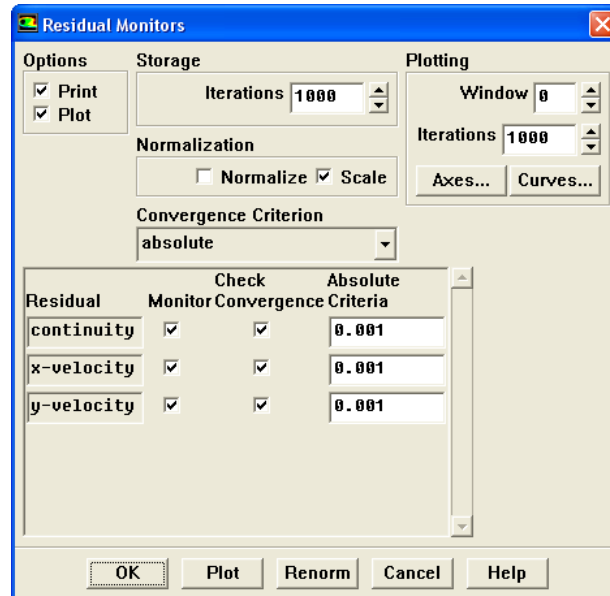
- (a) Enter 0.5 m/s for X Velocity.
- (b) Enter 0.02 m/s for Y Velocity.

(c) Click Init and close the Solution Initialization panel.

To have a good convergence, provide an initial estimate for the velocity field. As the wall is moving with a velocity of 1 m/s, in X direction, X-velocity can be set as some fraction of this value say, 0.5 m/s. Very low velocities such as 0.02 m/s can be set in Y direction.

3. Enable the plotting of residuals during the calculation.

Solve → Monitors → Residuals...



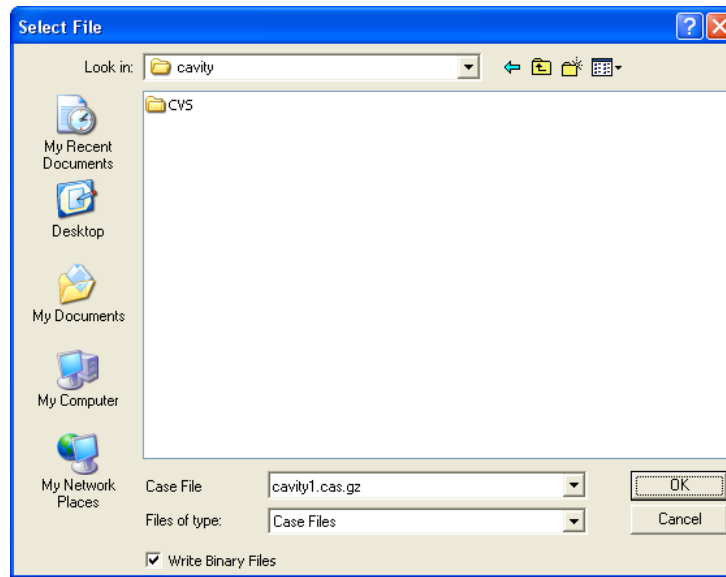
(a) Enable Plot in the Options list.

(b) Click OK to close the Residual Monitors panel.

4. Save the case file (cavity1.cas.gz).

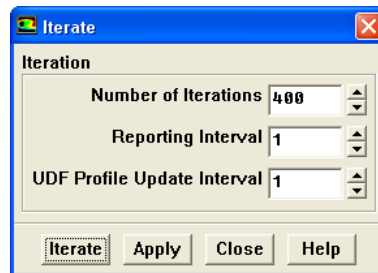
File → Write → Case...

Retain the default Write Binary Files option so that you can write a binary file. The .gz extension will save compressed files on both Windows and UNIX platforms.



5. Start the calculation by requesting 400 iterations.

Solve → Iterate...



- (a) Enter 400 for the Number of Iterations.
- (b) Click **Iterate** and close the **Iterate** panel.

The solution converges in about 320 iterations, with default convergence criteria. The residuals plot is shown in Figure 1.3.

Experimental results for X and Y -velocity along the two center lines passing through the flow domain are available. For comparison purpose, isosurfaces needs to be created at $X=0.5$ and $Y=0.5$.

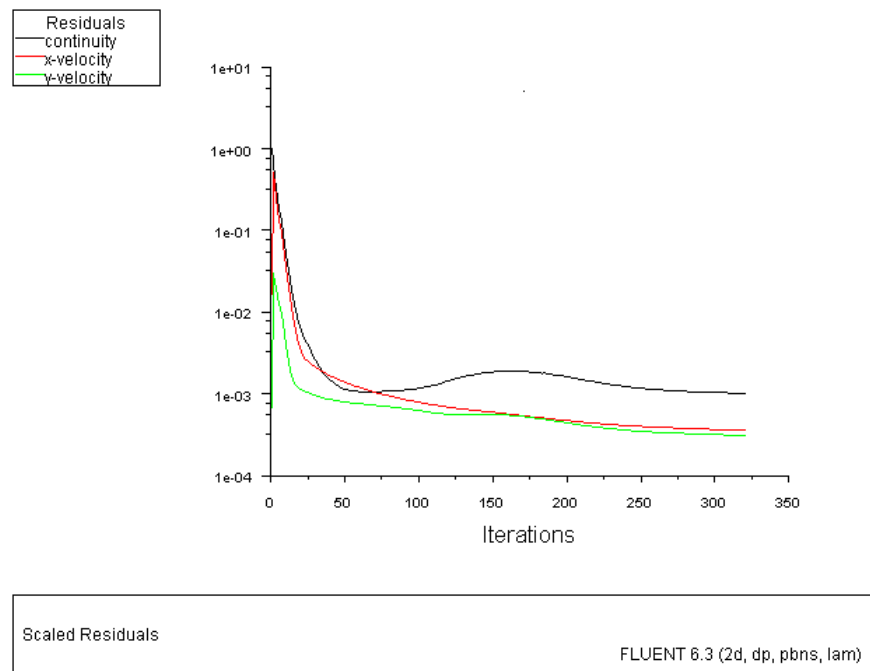
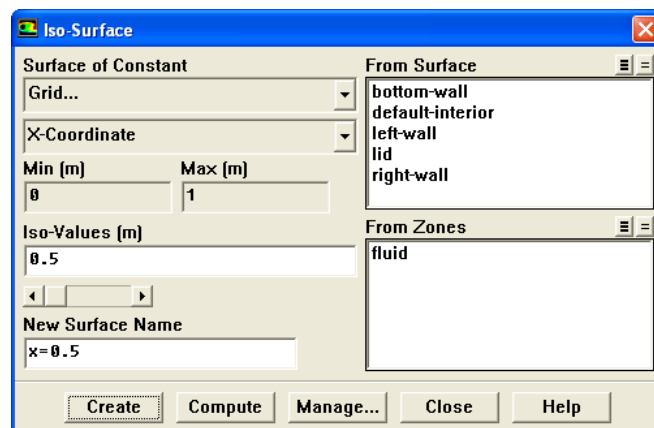


Figure 1.3: Scaled Residuals

6. Create an isosurface.

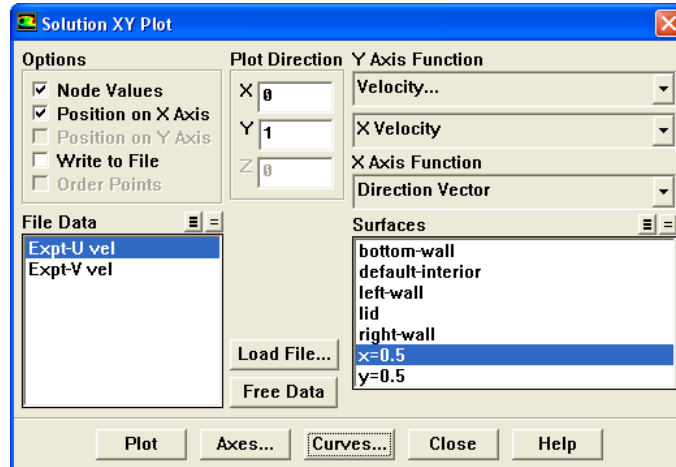
Surface → Iso-surface...



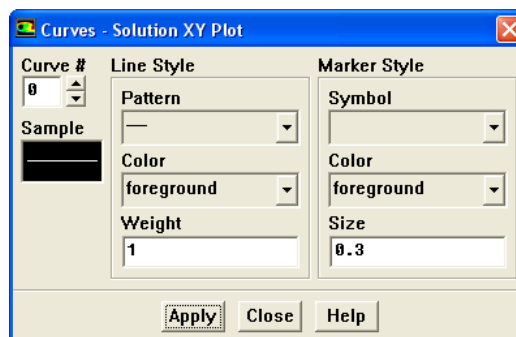
- Select Grid... and X-Coordinate from the Surface of Constant drop-down lists.
- Enter 0.5 m for the Iso-Values.
- Enter x=0.5 for the New Surface Name.
- Click Create.
- Similarly, create an isosurface for Y-coordinate, with the new surface name as y=0.5.

- (f) Close the Iso Surface panel.
7. Create a plot of X velocity (Figure 1.4).

Plot → XY Plot...



- (a) Click Load File....
This opens the Select File panel.
- Select files data-uvel.xy, and data-vvel.xy.
 - Click OK to close the Select File panel.
- (b) Select Expt-U-vel in the File Data list.
- (c) Select x=0.5 from the Surfaces list.
- (d) Enter 0 for X and 1 for Y in the Plot Direction group box.
- (e) Select Velocity... and X Velocity from the Y Axis Function drop-down lists.
- (f) Click the Curves... button.



- i. Select line from the **Pattern** drop-down list.
 - ii. Select blank from the **Symbol** drop-down list.
 - iii. Click **Apply** and close the **Curves - Solution XY Plot** panel.
- (g) Click **Plot**.

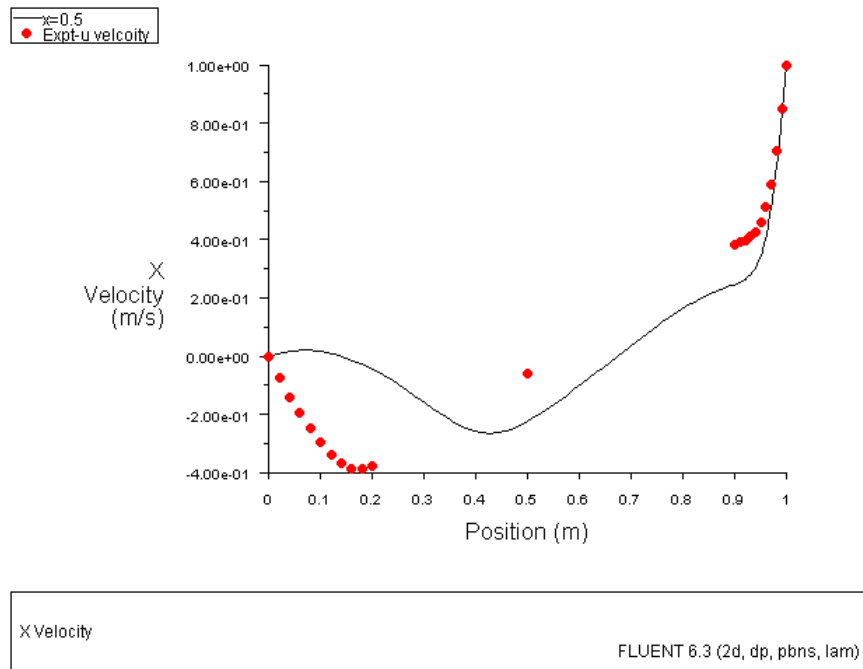


Figure 1.4: Calculated Vs. Experimental Data for X-velocity on $x=0.5$

8. Create a plot of Y-velocity.
 - (a) Select **Expt-V-vel** from the **File Data** list.
 - (b) Select $y=0.5$ from the **Surfaces** list.
 - (c) Enter 1 for X and 0 for Y in the **Plot Direction** group box.
 - (d) Select **Velocity...** and **Y Velocity** from the **Y Axis Function** drop-down lists.
 - (e) Click **Plot** and close the **Solution XY Plot** panel (Figure 1.5).

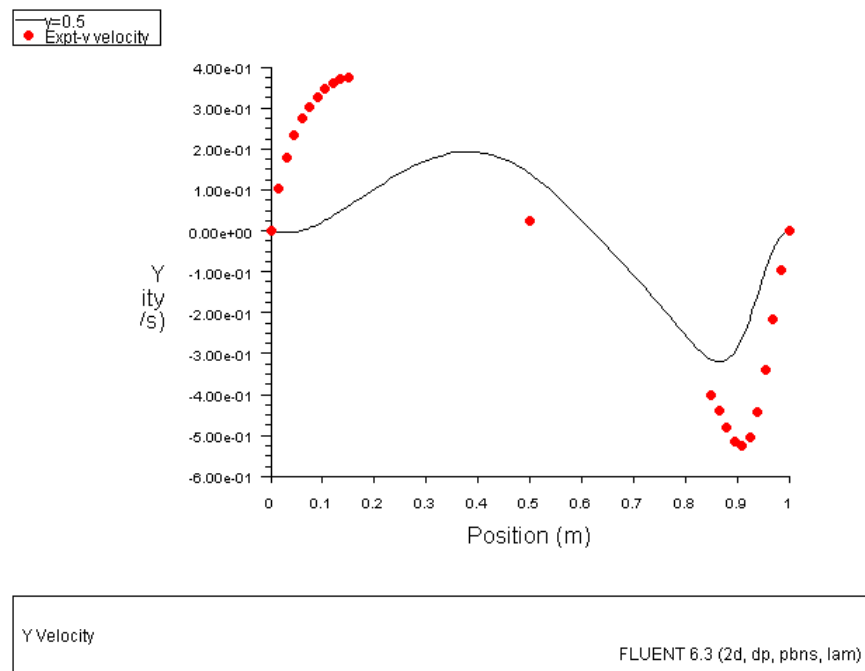


Figure 1.5: Calculated Vs. Experimental Data for Y-velocity on $y=0.5$

FLUENT results are very different from the experimental data. The results can be improved by using higher order discretization schemes, which will provide more accurate flow field.

9. Set the solution parameters.

Solve → **Controls** → Solution...

- (a) Select **Second Order Upwind** from the **Momentum** drop-down list.
- (b) Click **OK** to close the **Solution Controls** panel.

For a better match of results, the case has to converge till the residuals flatten out. In this case, convergence criteria can be set as $1e-5$ for all equations.

10. Set the convergence criteria for all equations.

Solve → **Monitors** → Residual...

- (a) Set the **Absolute Criteria** for continuity, x-velocity and y-velocity to $1e-5$.
- (b) Click **OK** to close the **Residual Monitors** panel.

11. Iterate the case till convergence criteria is met (Figure 1.6).

12. Save the case and data files (cavity2.cas.gz and cavity2.dat.gz).

File → **Write** → Case & Data...

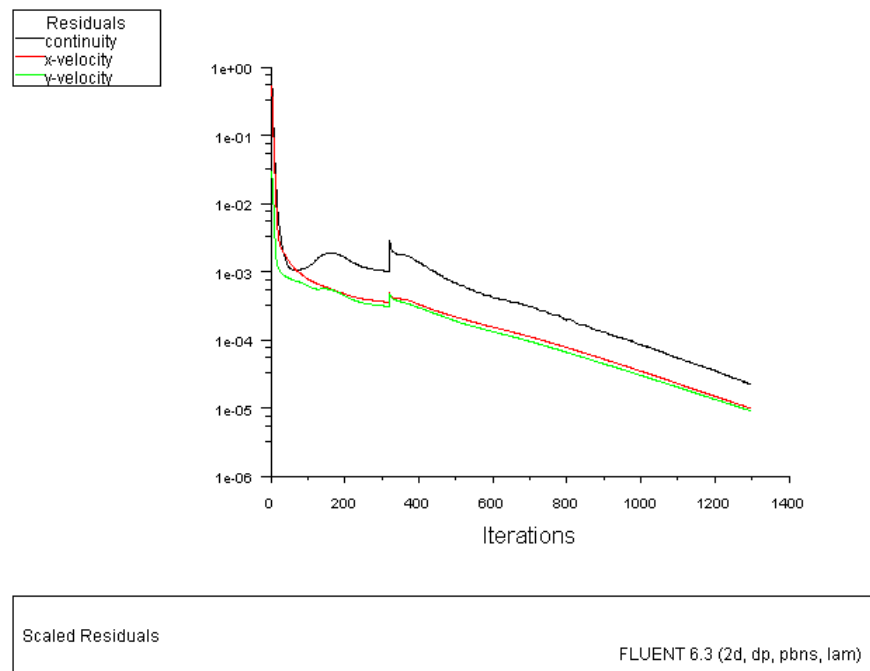


Figure 1.6: Scaled Residuals

13. Compare the new results again by repeating [steps 7 and 8](#) (Figures [1.7](#) and [1.8](#)).

Note: *Since the experimental data is already read, they can be used.*

The results for both the velocities are in very close agreement with the experimental values.

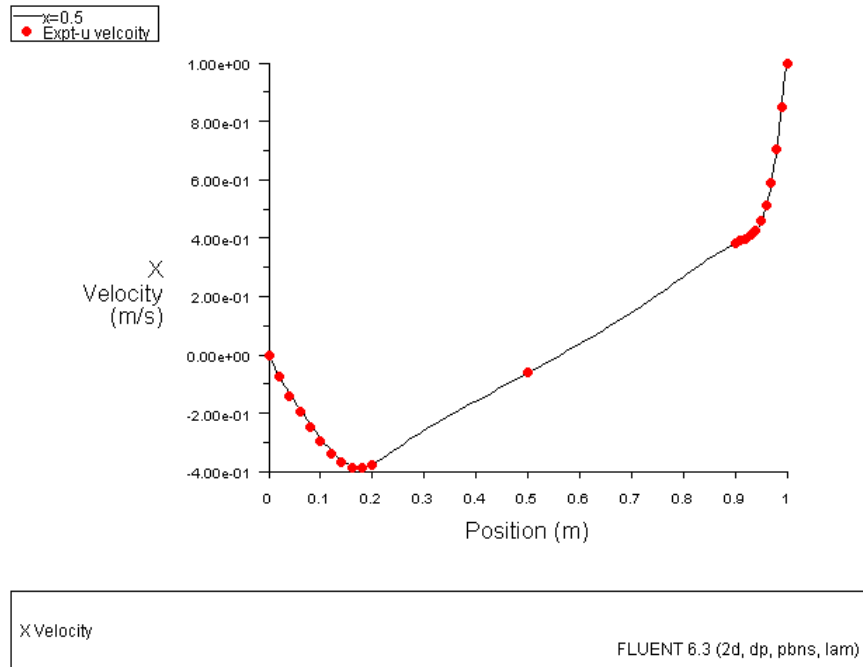


Figure 1.7: Calculated Vs. Experimental Data for X-velocity with Higher Order Solution

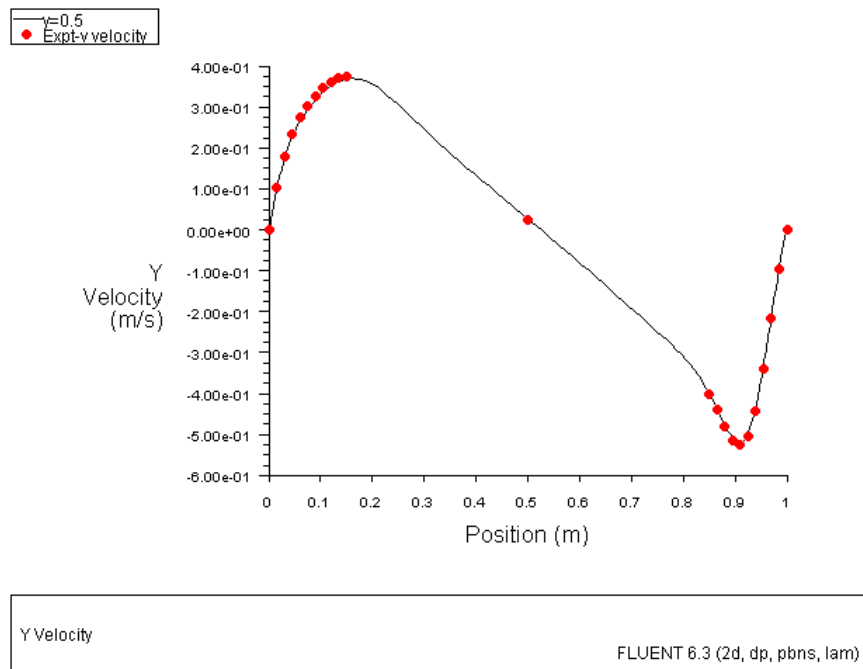
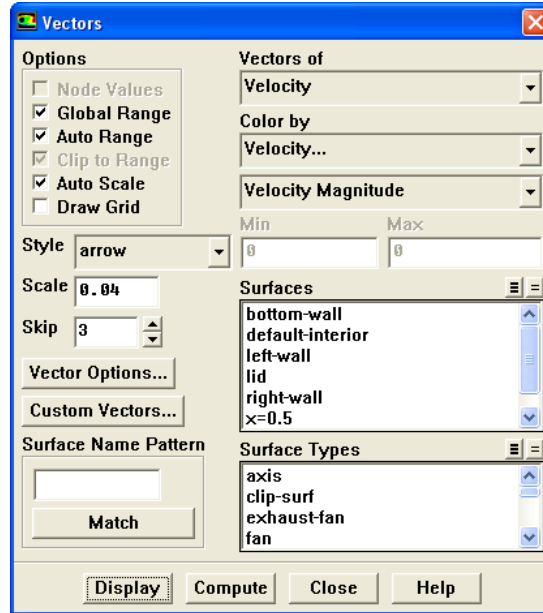


Figure 1.8: Calculated Vs. Experimental Data for Y-velocity with Higher Order Solution

Step 6: Postprocessing

1. Display velocity vectors (Figure 1.9).

Display → Vectors...



- (a) Click Vector Options....
- (b) Enable Fixed Length from the options.
- (c) Click Apply and close the Velocity Vectors panel.
- (d) Enter 0.04 for Scale and 3 for Skip.
- (e) Click Display and close Vectors panel.

Set the Fixed Length option to display all the vectors with the same length. Increase the Skip to reduce the number of vectors, so that the display does not have too many vectors.

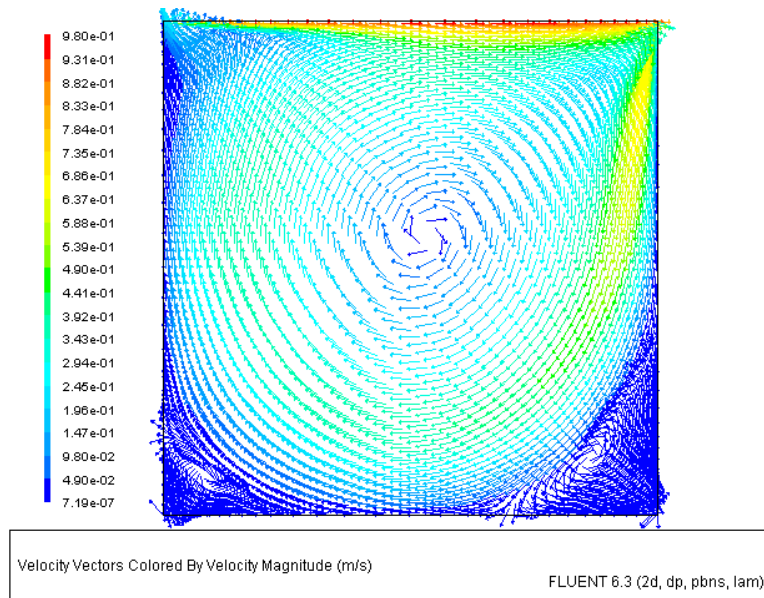
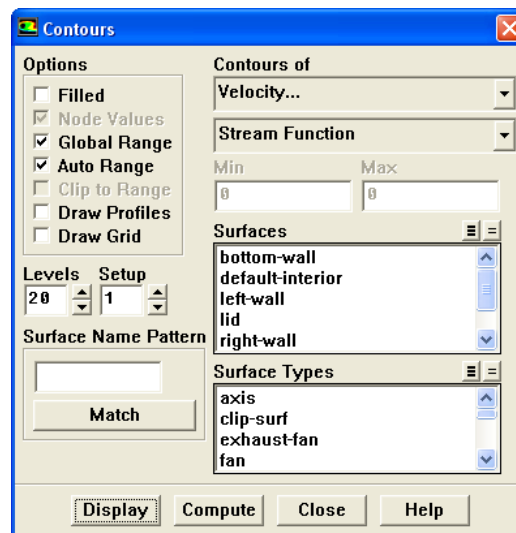


Figure 1.9: Velocity Vectors

2. Display stream function (Figure 1.10).

Display → Contours...



- (a) Select Velocity... and Stream Function from the Contours of drop-down lists.
 (b) Click Display.

Note: Right-click on a point in the domain to display the value of the corresponding contour in the console window.

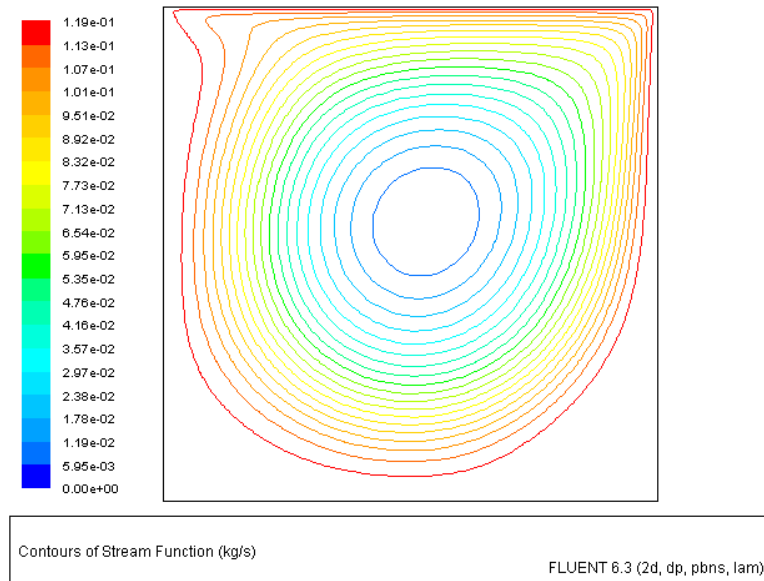


Figure 1.10: Contours of Stream Function

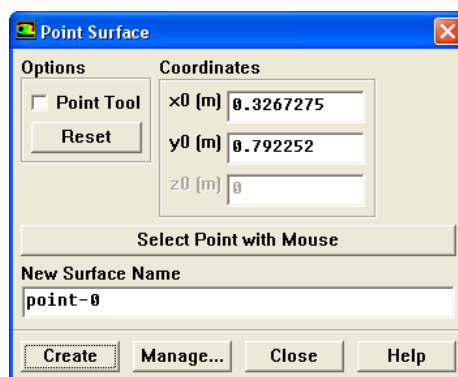
3. Change the range to view the vortices in the corners.

- (a) Deselect **Auto Range** from the **Options** list.
- (b) Enter 0.111 for the **Min**.
- (c) Click **Display** (Figure 1.11).

To visualize circulation in the cavity, display path lines by releasing from point surfaces. Create a point surface for doing this.

4. Create point surfaces.

Surface → **Point...**



- (a) Enter 0.3267275 for x_0 and 0.792252 for y_0 .

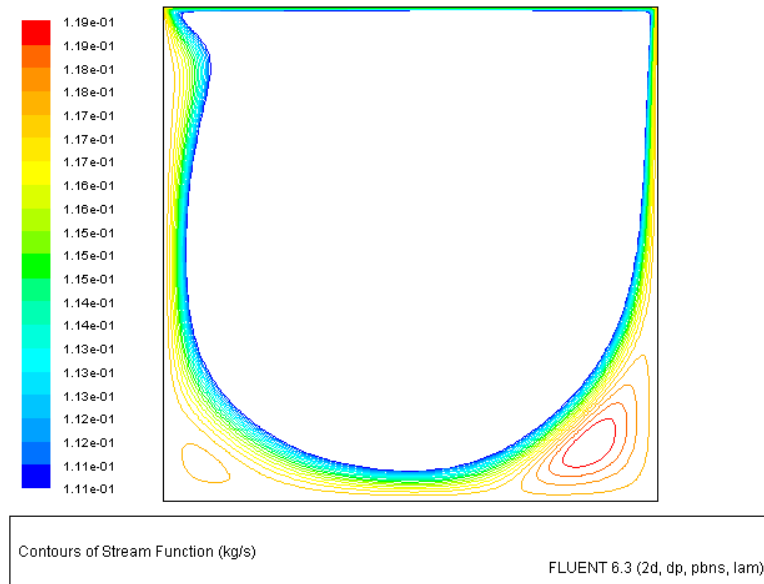


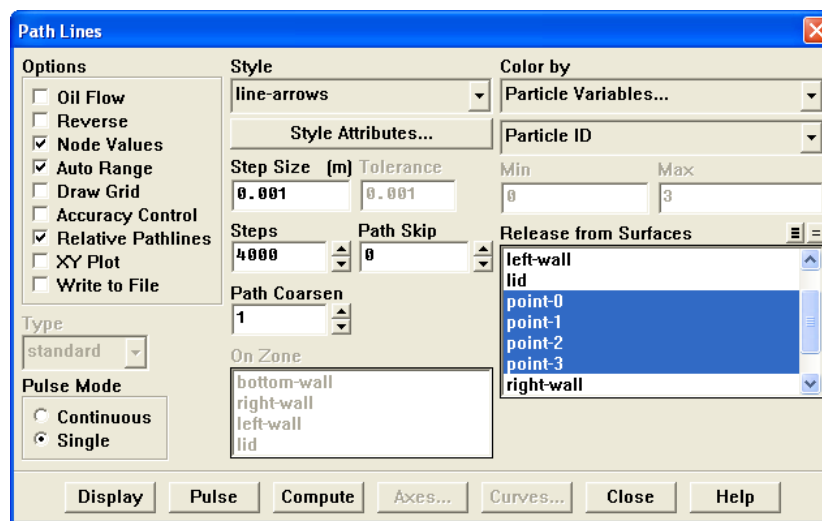
Figure 1.11: Contours of Stream Function with Vortices

- (b) Enter point-0 for New Surface Name.
- (c) Click Create.
- (d) Similarly, create three more points at $(x_0, y_0) = (0.0722, 0.17402)$, $(0.03889, 0.06426)$, and $(0.93554, 0.055343)$ respectively.

Display these four points by selecting all the point surfaces, under Surfaces and deselecting default-interior in the Grid Display panel.

5. Display path lines (Figure 1.12).

Display → Pathlines...



- (a) Select point-0, point-1, point-2 and point-3 from the Release from Surfaces list.
- (b) Enter 0.001 m for the Step Size and 4000 for the Steps.
- (c) Select line-arrows from the Style drop-down lists.
- (d) Click the Style Attributes... button.

- i. Set parameters in the Path Style Attributes panel as shown in the table:

Parameter	Value
Line Width	0.5
Spacing Factor	4
Scale	0.15

- ii. Click OK and close the panel.

- (e) Click Display.

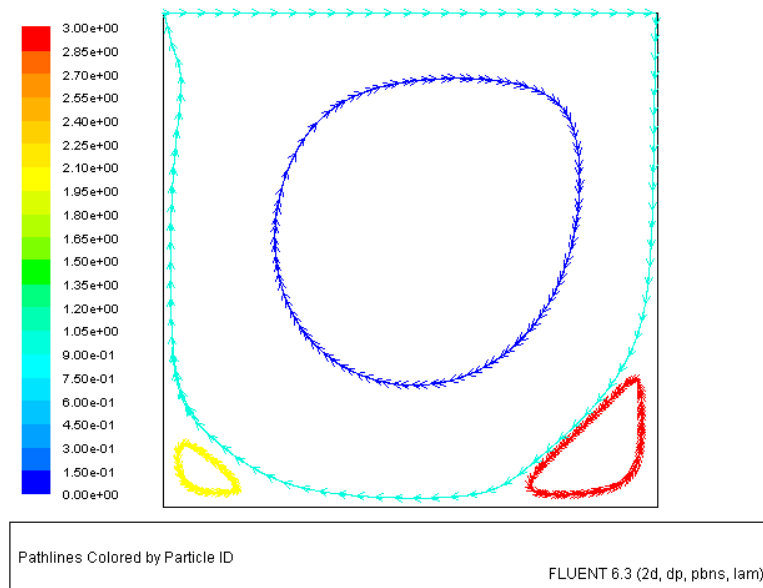


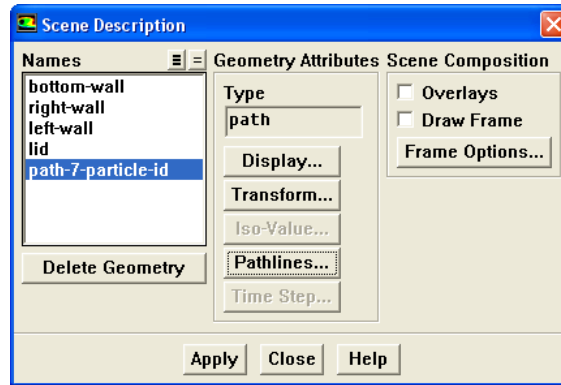
Figure 1.12: Path Lines

6. Create animations for path lines.

After displaying path lines, create an animation file so that the progress of path lines can be viewed clearly.

- (a) Change the display to have small path lines

Display → Scene...

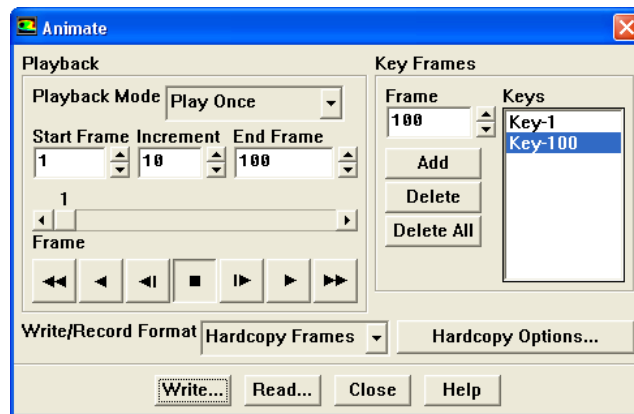


- i. Select path-7-particle-id from the Names list.
- ii. Click Path Lines....
- iii. Enter 2 for the Max Steps in the Pathline Attributes panel and click Apply.
- iv. Click Apply and close the Scene Description panel.

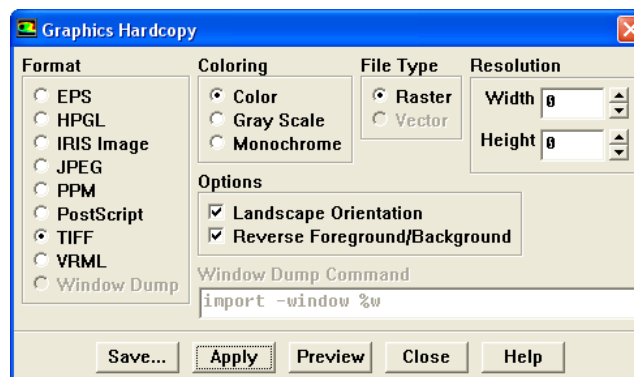
This will change the display in graphics window, with very small path lines

7. Display the scene animation.

Display → Scene Animation...



- (a) Click Add to create key-1 in the Keys list.
- (b) Enter 4000 for Max Steps in the Pathline Attributes panel and click Apply.
- (c) Enter 100 for Frame and click Add in the Animate panel.
- (d) Select Hardcopy Frames from the Write/Record Format drop-down list.
- (e) Click the Hardcopy Options... button.



- i. Select TIFF from the Format list.
 - ii. Select Color from the Coloring list.
 - iii. Click Apply and close the Graphics Hardcopy panel.
- (f) Click the Write... button in the Animate panel.
- (g) Enter `pic.tif` for the file name and click OK to close the Select File panel.
Files `pic0001.tif`, `pic0002.tif`, ... , and `pic0100.tif` will be written.
Create an animated .gif file with the .tif files using third party animation tools.

Summary

Comparison of center-line velocity for the first-order and second-order solution with the experimental values clearly indicates that second order schemes are essential for capturing correct flow features. First order scheme is the default scheme in **FLUENT**, it is a good practice to use first-order solution as a starting guess for further calculations.

References

E. Erturk, T.C. Corke and C. Gokcol, *Numerical Solutions of 2D Steady Incompressible Driven Cavity Flow at High Reynolds Numbers*, accepted for publication in International Journal for Numerical Methods in Fluids, 2005.

Exercises/Discussions

What will be the flow pattern in each of following situations:

1. The lid direction is reversed.
2. The bottom wall is given a velocity in same direction of the lid.
3. The bottom wall is given a velocity in opposite direction that of lid.
4. The domain height is increased.
5. What would be the vortex size if the lid speed is increased? Why?
6. How would you model the situation if normal motion is provided to the lid? Will the flow remain incompressible in that situation?
7. What other situation can be simulated using the same mesh file.

Links for Further Reading

- <http://www.cavityflow.com>
- http://www.csit.fsu.edu/~burkardt/datasets/cavity_flow/cavity_flow.html
- <http://nenes.eas.gatech.edu/CFD/1phase/PCCAV1/PCcav.htm>
- <http://www.unibas.it/utenti/bonfiglioli/node14.html>
- <http://wissrech.iam.uni-bonn.de/research/projects/NaSt3DGP/documentation/userguide/node35.html>

