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.

In this tutorial you will learn how to:

- 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 $Re = 1000$.

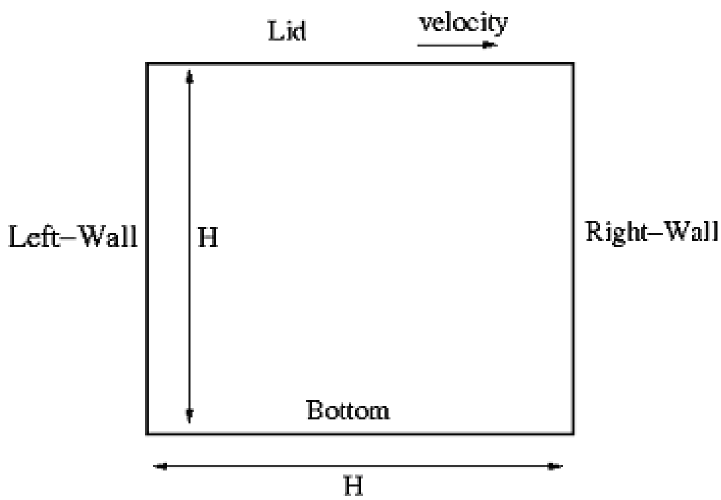


Figure 1.1: Problem Schematic

Preparation

1. Copy the following files to your working directory:
   - `cavity.msh`
   - `data-uvel.xy`
   - `data-vvel.xy`
2. Start the 2D double precision solver of FLUENT.

Setup and Solution

Step 1: Grid

1. Read the grid file, `cavity.msh`.

   File ⟶ Read ⟶ Case...

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

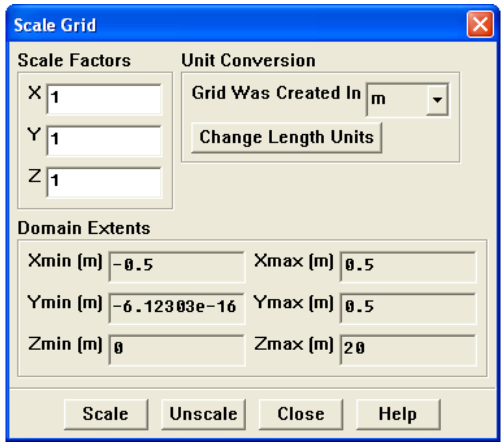
2. Check the grid.

   Grid ⟶ Check

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

3. Check the scale of the grid.

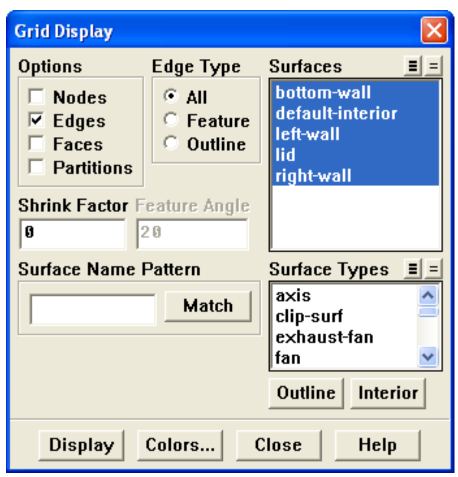
   Grid ⟶Scale...



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

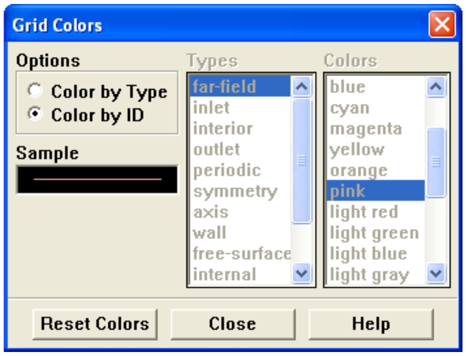
4. Display the grid (Figure 1.2).

   Display ⟶Grid...



(a) Click Colors....

The Grid Colors *panel opens.*



i. Under Options, enable Color by ID.

ii. Click Close.

(b) In the Grid Display panel, click Display

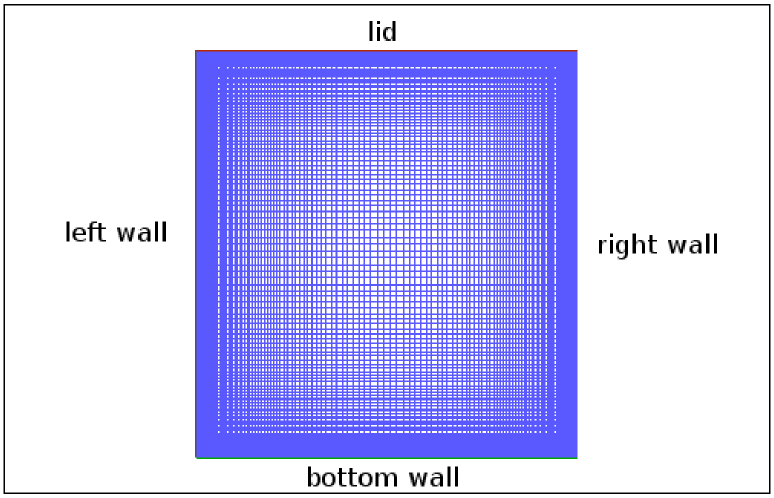


Figure 1.2: Grid Display

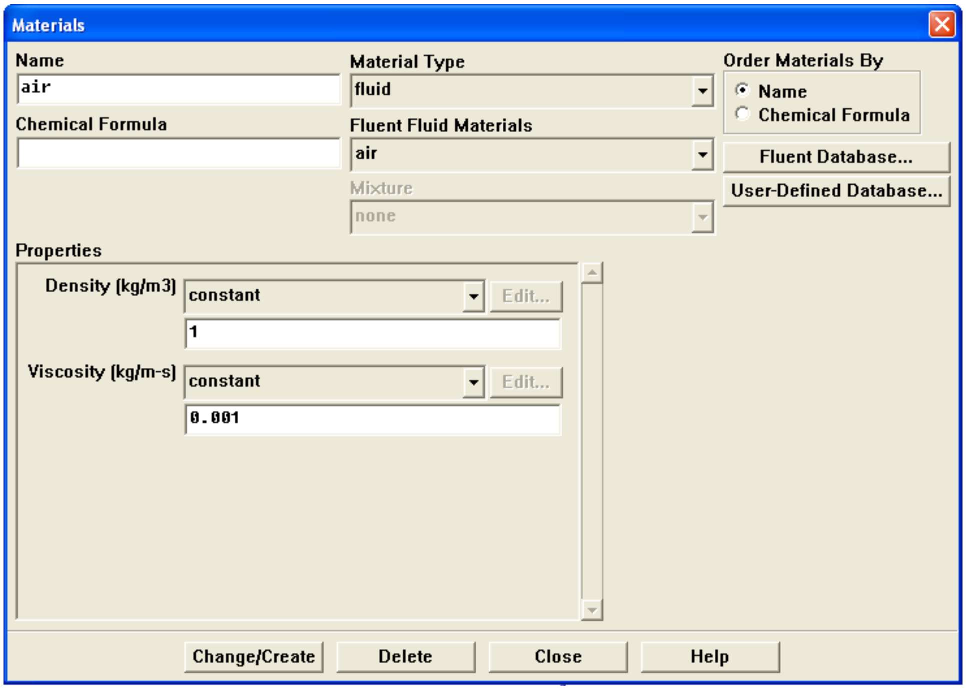
The grid adjacent to the walls is finer compared to that in the central region. The purpose of such fine mesh is to capture sharp gradients near the walls correctly.

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...



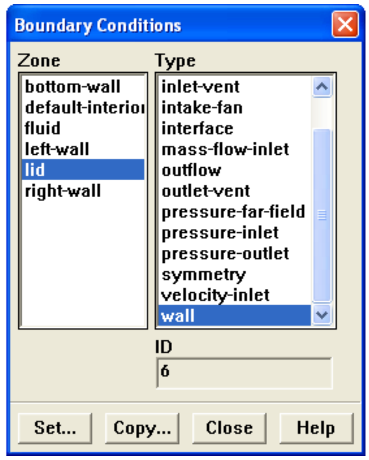
1. Set the Density (kg/m3) to 1.
2. Set Viscosity (kg/m-s) to 0.001.
3. Click on Change/Create and close the panel.

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

Step 4: Boundary Conditions

1. Set the boundary condition for lid.

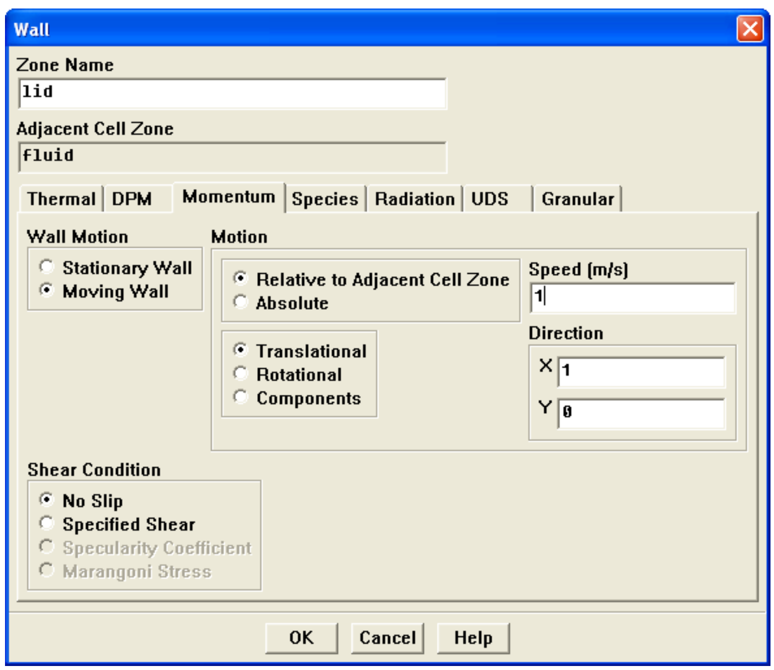
   Define ⟶Boundary Conditions...



   (a) Under Zone, select lid.

   (b) Click Set....

   The Wall *panel opens.*



   (c) Click on the Momentum tab.

   (d) Under Wall Motion, select Moving Wall

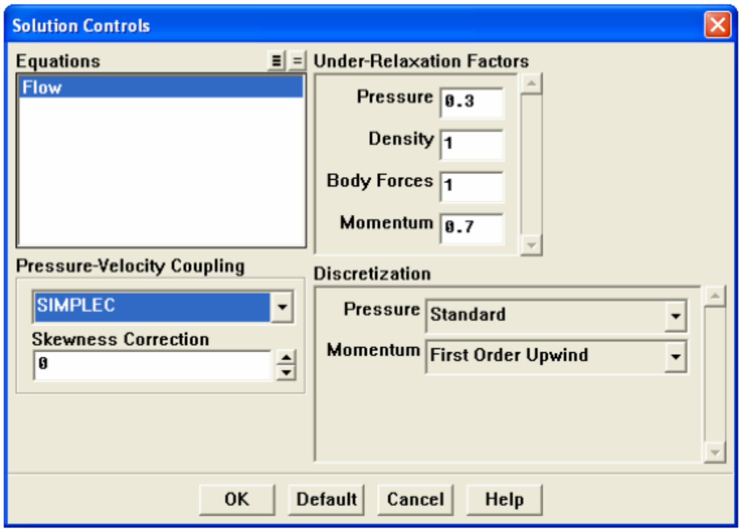
   (e) Set the Speed (m/s) as 1 and click OK.

2. Close the Boundary Conditions panel.

Step 5: Solution

1. Set the solution controls.

   Solve ⟶ Controls ⟶ Solution...



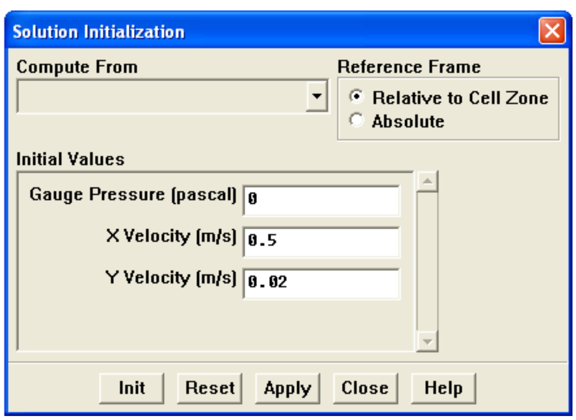
   (a) Select SIMPLEC for Pressure-Velocity Coupling.

   (b) Click OK to close the 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) Set the X Velocity (m/s) as 0.5.

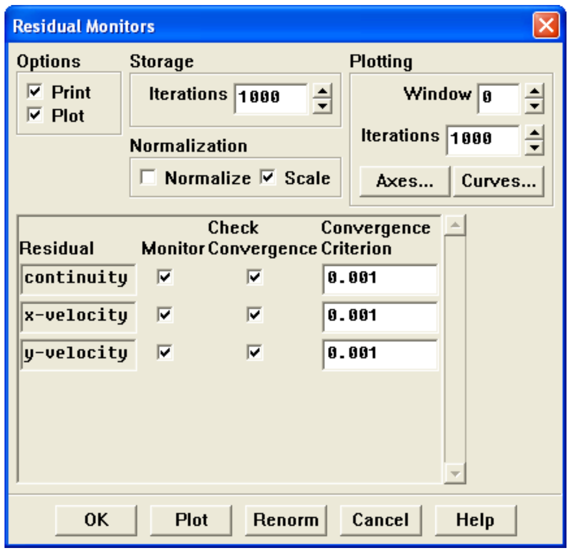
   (b) Set the Y Velocity (m/s) as 0.02.

(c) Click Init and close the 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.

$\boxed{\text{Solve}} \longrightarrow \boxed{\text{Monitors}} \longrightarrow$ Residuals...



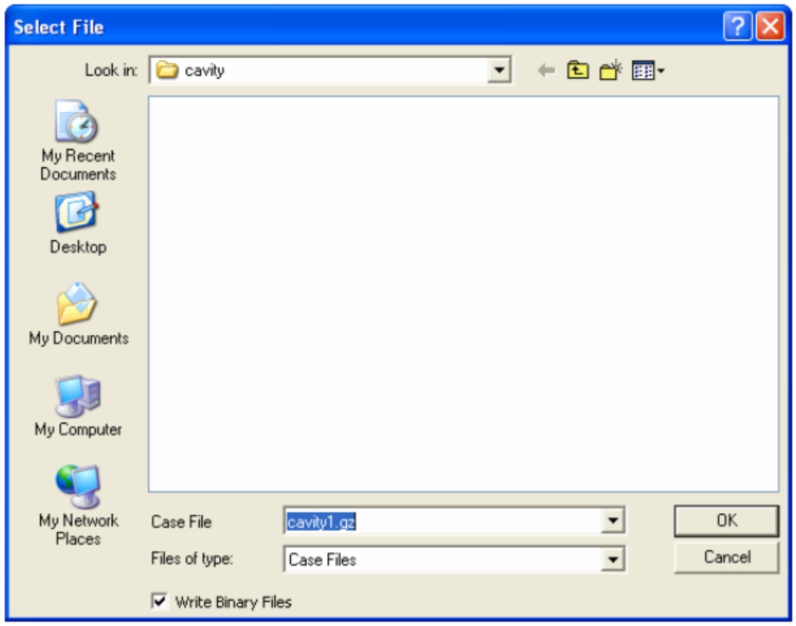
(a) Under Options, enable Plot.

(b) Click OK.

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

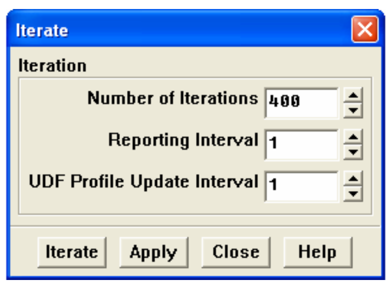
$\boxed{\text{File}} \longrightarrow \boxed{\text{Write}} \longrightarrow$ Case...

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) Set Number of Iterations to 400.

   (b) Click Iterate.

   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 for comparison at X=0.5 and Y=0.5.

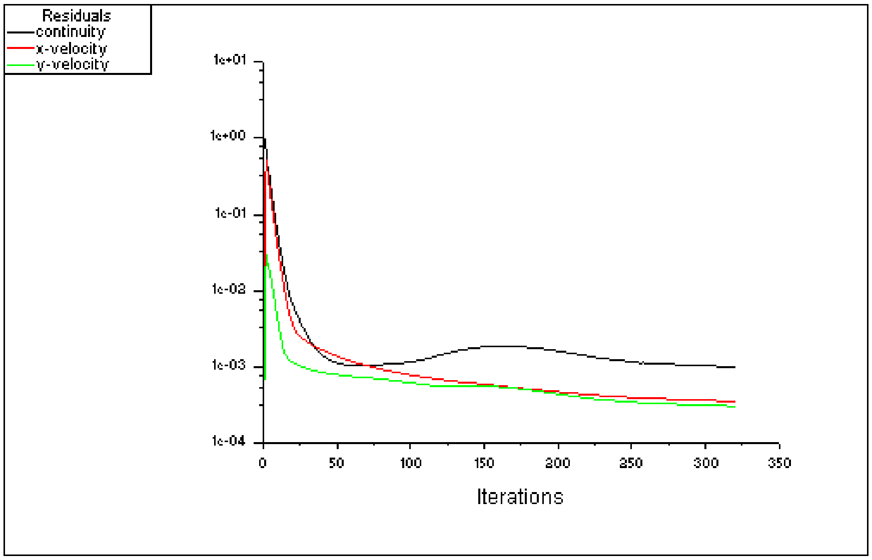
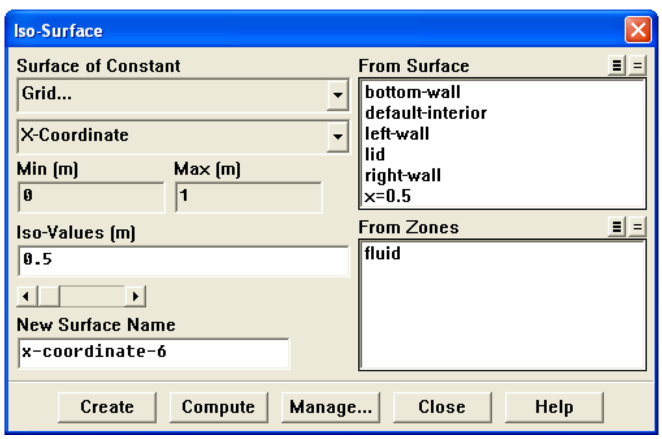


Figure 1.3: Scaled Residuals

6. Create an isosurface.

   $\boxed{\text{Surface}}$ ⟶ Iso-surface...



   (a) Under Surface of Constant, select Grid... and X-Coordinate.

   (b) Under Iso-Values, enter a value of `0.5`.

   (c) Under New Surface Name, enter `x=0.5`.

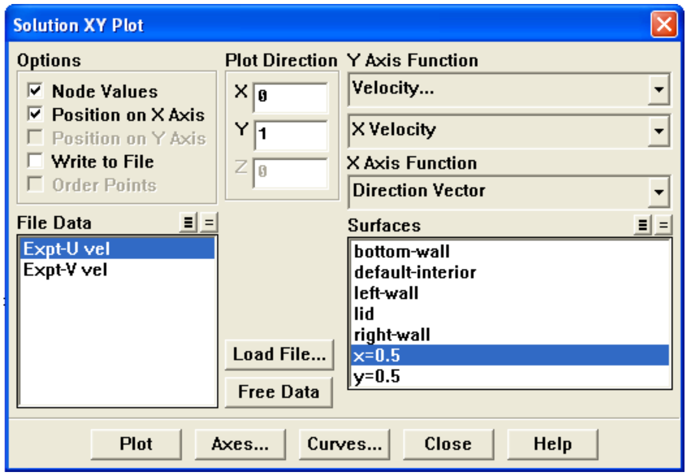
   (d) Click Create.

   (e) Similarly, create an isosurface for Y-coordinate, with the new surface name as `y=0.5`.

   (f) Close the panel.

7. Create a plot of X velocity (Figure 1.4).

   Plot ⟶XY Plot...



   (a) Click Load File....

   This opens the Select File *panel.*

   i. Select files `data-uvel.xy`, and `data-vvel.xy`.

   ii. Click OK to close the Select File panel.

   (b) Under File Data, select Expt-U-vel.

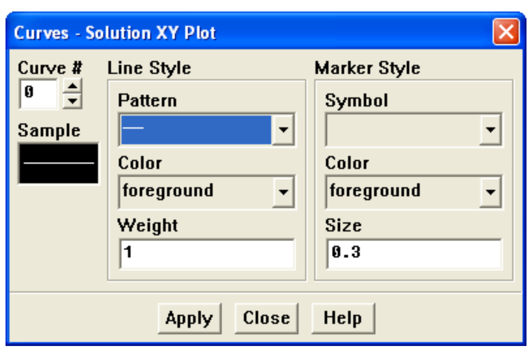
   (c) Under Surfaces, select x=0.5.

   (d) Set the Plot Direction for X as `0` and Y as `1`.

   (e) Under Y Axis Function, select Velocity... and X Velocity.

   (f) Click Curves....

   This opens the Curves - Solution XY Plot *panel.*



i. Under Pattern, select the line in the drop-down list.

ii. Under Symbol, select the blank in the drop-down list.

iii. Click Apply and close the panel

(g) Click Plot.

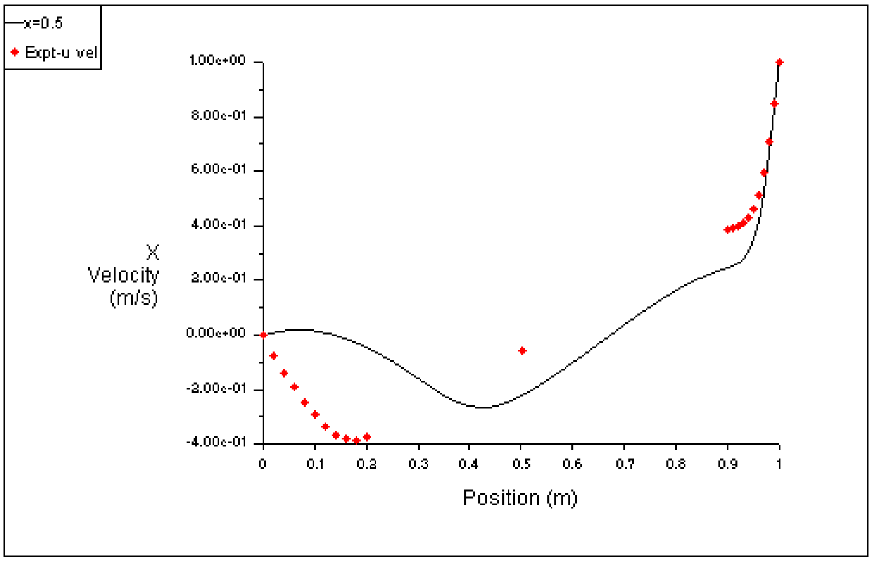


Figure 1.4: Calculated Vs. Experimental Data for X-velocity on x=0.5

8. Create a plot of Y-velocity.

(a) Under File Data, select Expt-V-vel.

(b) Under Surfaces, select y=0.5.

(c) Set the Plot Direction for X as 1 and Y as 0.

(d) Under Y Axis Function, select Velocity... and Y Velocity.

(e) Click Plot (Figure 1.5).

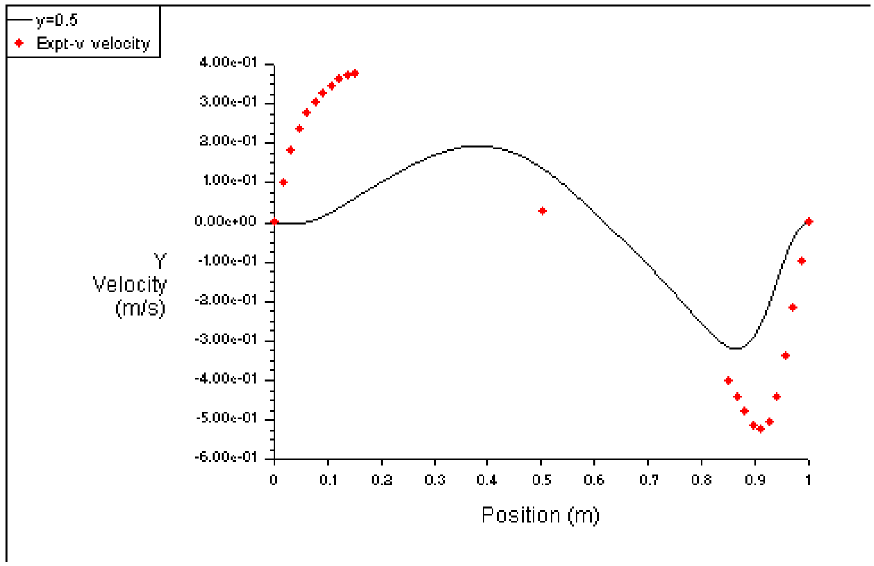


Figure 1.5: Calculated Vs. Experimental Data for Y-velocity on y=0.5

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 second order upwind scheme for momentum.

   Solve ⟶ Controls ⟶Solution...

   (a) Under Discretization, select Second Order Upwind for Momentum.

   (b) Click OK.

   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 ⟶Residuals...

    (a) Set Convergence Criterion for continuity, x-velocity, y-velocity to `1e-5`.

    (b) Click OK.

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...

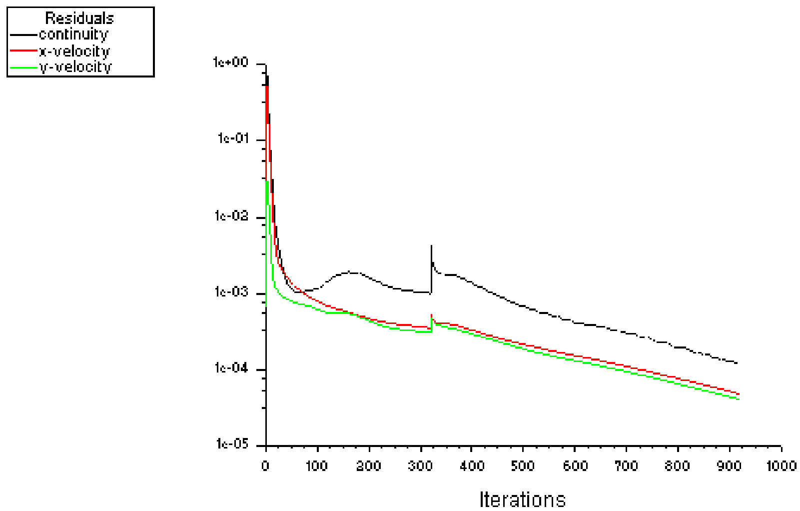


Figure 1.6: Scaled Residuals

13. Compare the new results again by repeating steps 5.7 and 5.8 (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.

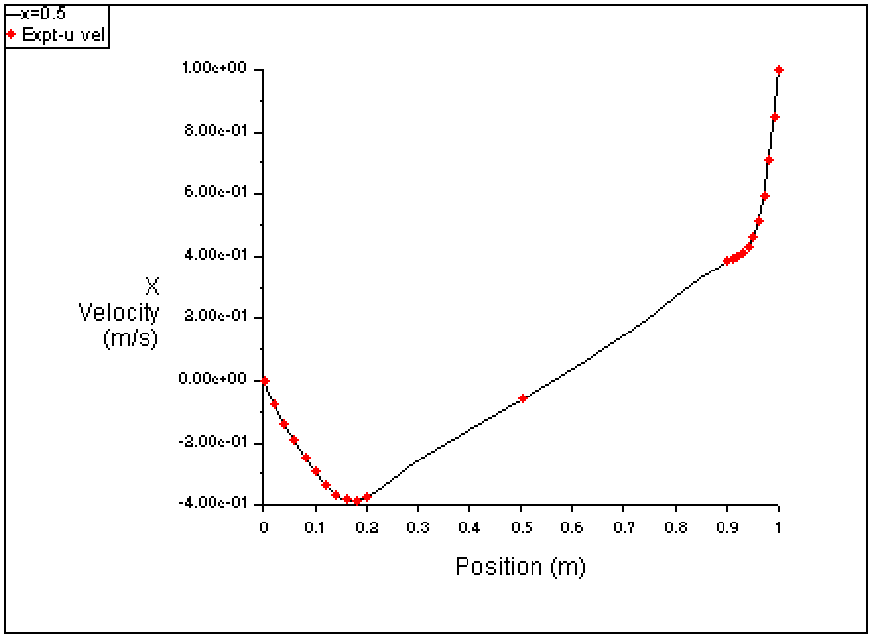


Figure 1.7: Calculated Vs. Experimental Data for X-velocity with Higher Order Solution

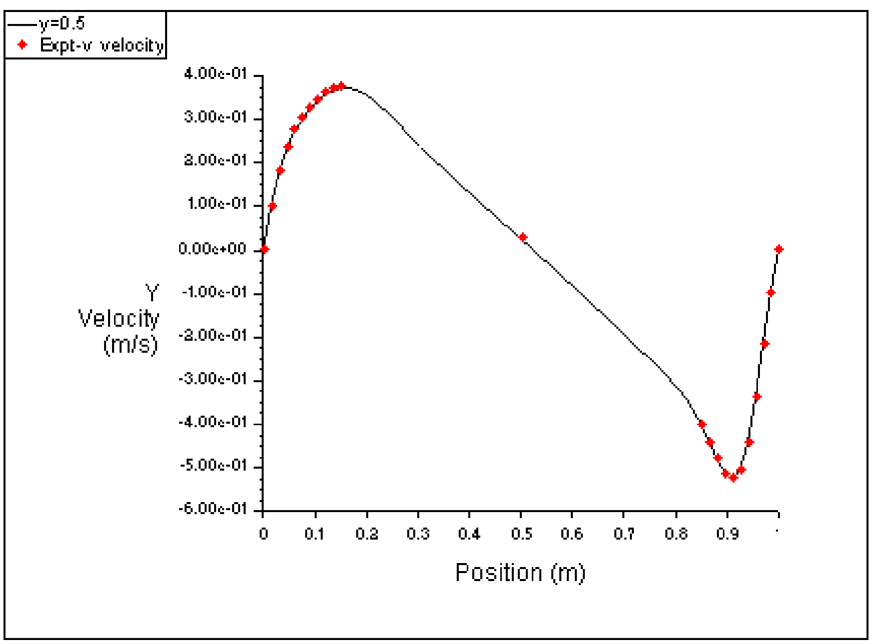
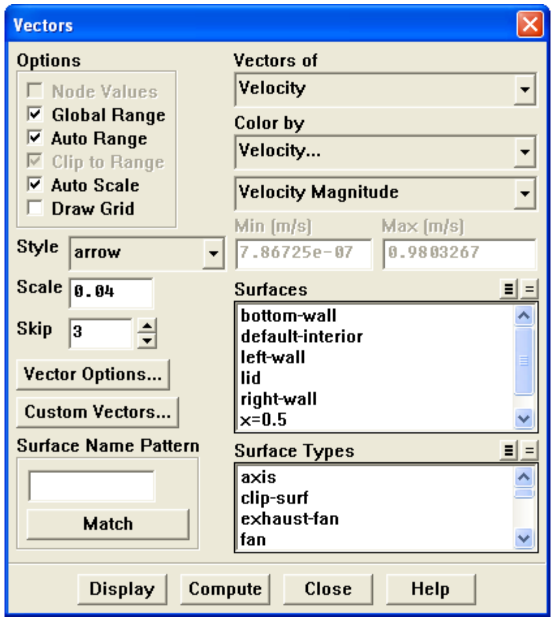


Figure 1.8: Calculated Vs. Experimental Data for Y-velocity with Higher Order Solution

Step 6: Postprocessing

1. Display velocity vectors. (Figure 1.9)

   Display ⟶Vectors...



   (a) Click Vector Options...

   (b) In the Vector Options panel, enable Fixed Length.

   (c) Click Apply and close the panel.

   (d) Set the Scale to `0.04` and increase Skip to `3`.

   (e) Click Display.

   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.*

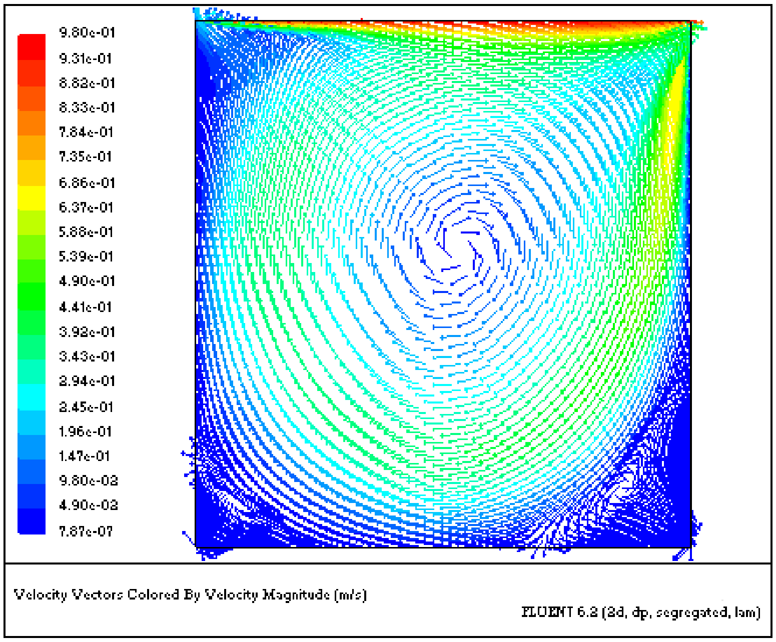
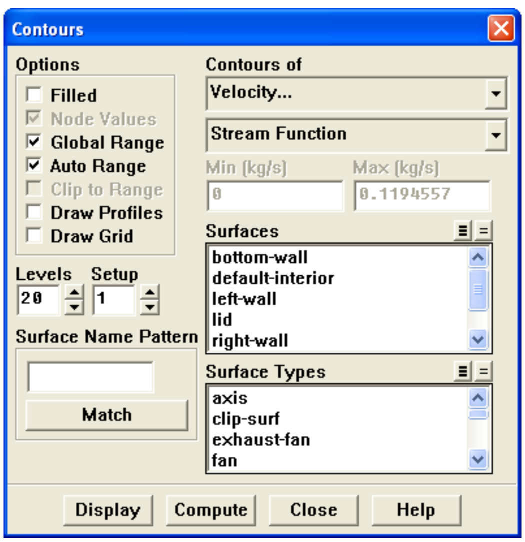


Figure 1.9: Velocity Vectors

2. Display stream function (Figure 1.10).

   Display ⟶Contours...



   (a) Under Contours of select Velocity... and Stream Function.

   (b) Click Display.

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

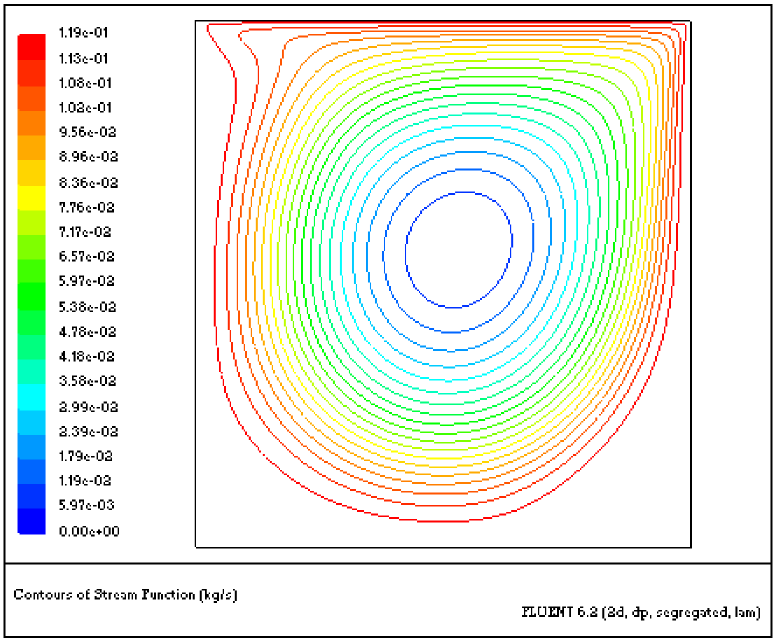


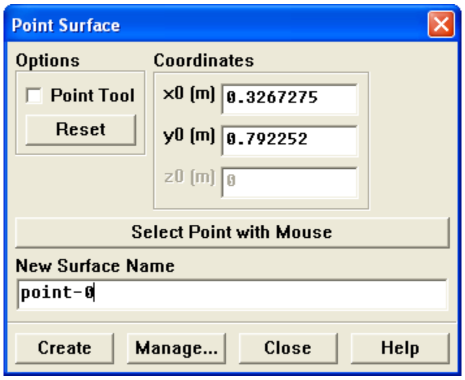
Figure 1.10: Contours of Stream Function

3. Change the range to view the vortices in the corners.
   (a) In Contours panel, under Options, deselect Auto Range.
   (b) Set the Min value to `0.111`.
   (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) Set x0 and y0 to `0.3267275` and `0.792252` respectively.

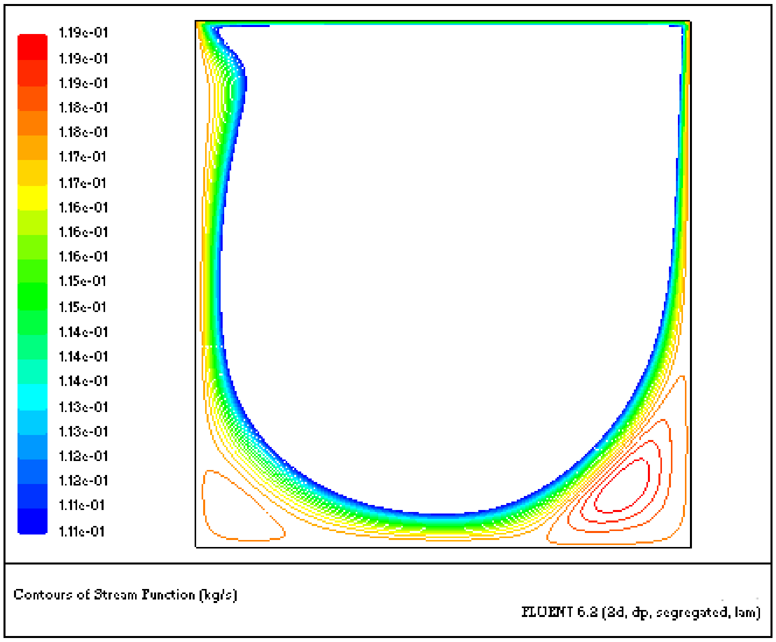


Figure 1.11: Contours of Stream Function with Vortices

(b) Under New Surface Name, enter `point-0`.

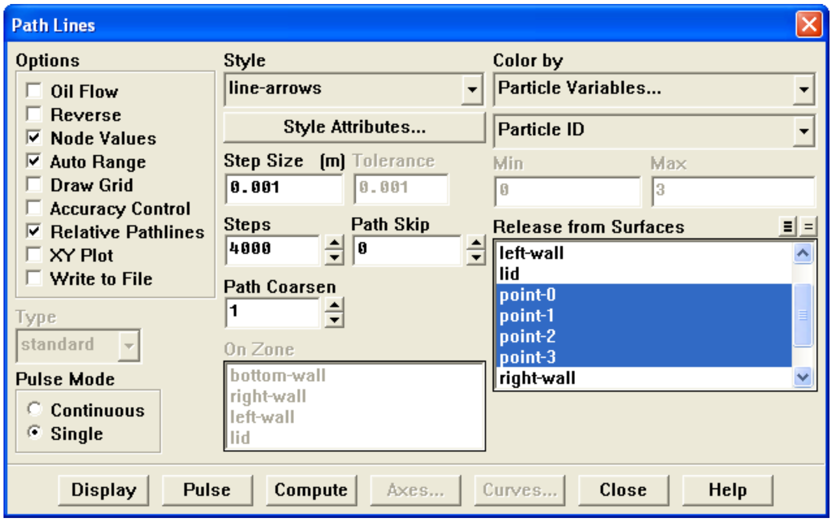
(c) Click Create.

(d) Similarly, create three more points at (x0, y0) = (`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) Under Release from Surfaces, select point-0, point-1, point-2, and point-3.

(b) Set Step Size to `0.001` and Steps to `4000`.

(c) Under Style, select line-arrows.

(d) Click Style Attributes....

  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.

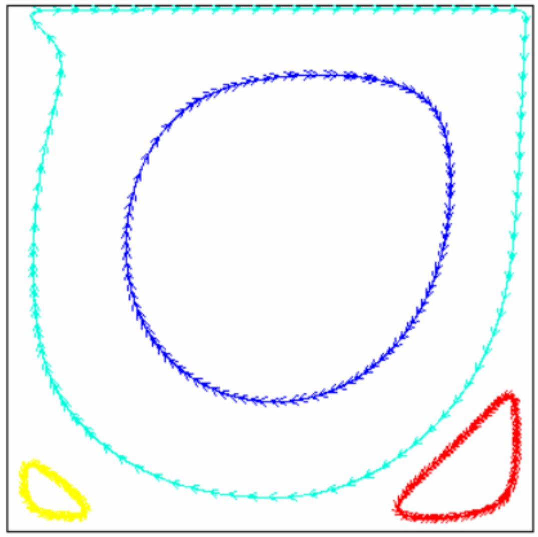


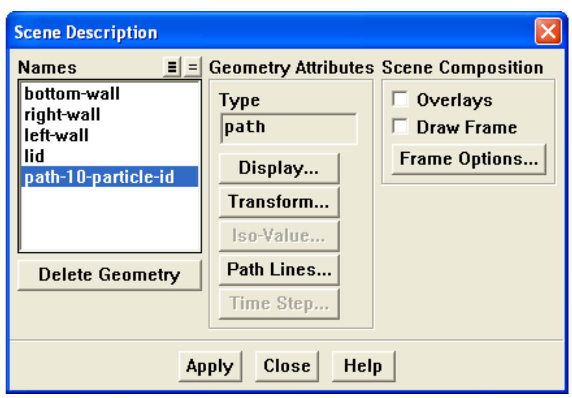
Figure 1.12: Path Lines

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. Under Names, select path-10-particle-id .

   ii. Click Path Lines...

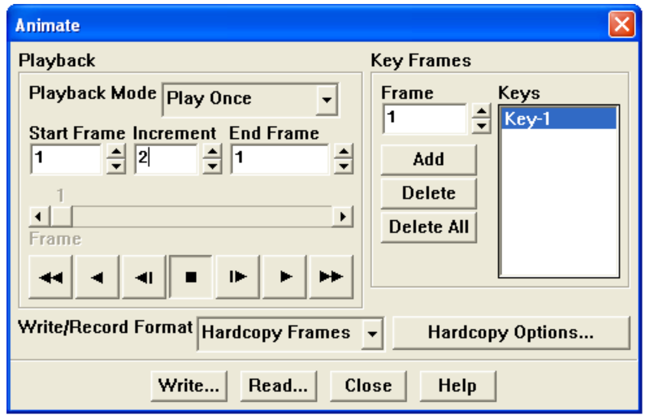
   iii. In the Pathline Attributes panel, set Max Steps to 2.

   iv. Click Apply.

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

(b) Display the scene animation.

Display ⟶Scene Animation...



i. Click Add.

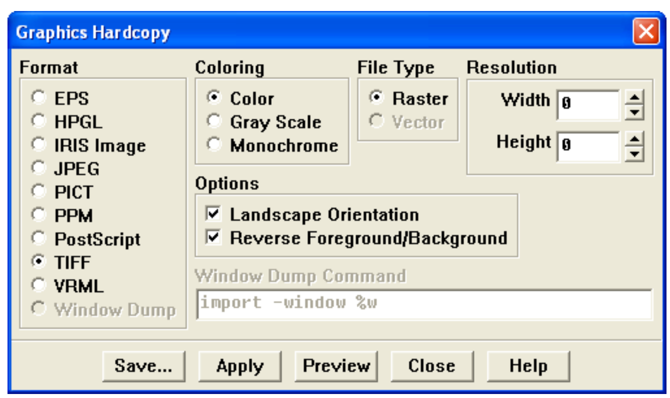
Under Keys, key-1 *will be created.*

ii. In the Pathline Attributes panel, set Max Steps to 4000, and click Apply.

iii. In the Animate panel, set Frame to 100 and click Add.

iv. Select Hardcopy Frames, as Write/Record Format.

v. Click Hardcopy Options...



A. Under Format and Coloring, select TIFF and Color respectively.

B. Click Apply and close the panel.

C. Click Write...

D. In the Select File panel, provide a file name with a `.tif` extension (`pic.tif`) and click OK.

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