Steady Flows in Annular Wedge Lid-Driven Cavities

Anandan Ganapathy
anandanganapathy1@gmail.com

Follow this and additional works at: https://huskiecommons.lib.niu.edu/allgraduate-thesesdissertations

Part of the Aerodynamics and Fluid Mechanics Commons, and the Mechanical Engineering Commons

Recommended Citation
https://huskiecommons.lib.niu.edu/allgraduate-thesesdissertations/7056

This Dissertation/Thesis is brought to you for free and open access by the Graduate Research & Artistry at Huskie Commons. It has been accepted for inclusion in Graduate Research Theses & Dissertations by an authorized administrator of Huskie Commons. For more information, please contact jschumacher@niu.edu.
ABSTRACT

STEADY FLOWS IN ANNULAR WEDGE LID-DRIVEN CAVITIES

Anandan Ganapathy, MS
Department of Mechanical Engineering
Northern Illinois University, 2019
John Shelton, Director

The lid-driven cavity is a well-known canonical problem in the field of analytical and computational fluid dynamics. It is comprised of a square domain of an incompressible fluid with its upper lid of a specified velocity and the remaining wall boundaries subjected to fixed, no-slip, zero-velocity constraints. There are numerous examples found in the literature that address the well-known stable behavior of this system, which includes a primary recirculating vortex, secondary corner vortices, and a velocity profile through the center midpoint y-axis that is shown to be dependent on the Reynolds number. In this investigation, the shape of the square cavity is modified into an annular wedge-shaped cavity (W-LDC) with varying center angle and its fluid behavior is analyzed numerically over a range of Reynolds numbers: 100, 400, and 1000. The primary and secondary vortices of these W-LDCs with different wedge angles and different lid lengths provide a benchmark for comparison with the square lid-driven cavity. In order to obtain these results, open-source CFD software called OpenFOAM is used to solve the Navier-Stokes equations using the PISO algorithm.

A grid-independent study was performed that determined an optimal grid size of 240*240 cells in both the radial and angular directions. An optimal simulation time study was also carried out that checked the minimum stream function value ($\Psi_{min}$) for sufficiently calculated simulation time. The purpose of these studies and analyses will aid in the future findings of stable-flow structures and how the vortexes are formed in an annular wedge-shaped cavity and will aid in the
future findings of advantageous methodologies to determine the flow characterization of drilling mud.
ACKNOWLEDGEMENTS

I express my sincere gratitude to my advisor, Dr. John Shelton, for his constant support and encouragement throughout the tenure of this thesis. I deem it a great privilege to have been able to seek knowledge from him; the entire journey of learning would have been impossible without his guidance. I would also like to thank my committee members, Dr. Nicolas Pohlman and Dr. Kyu Taek Cho, who have extended their support by giving some valuable ideas to improve my thesis.

I would like to give my special thanks to my parents for their love, support and blessings that have enabled me to complete this master’s degree. I would like to thank my girlfriend, Vandanaa Rajendran, for all her support so far in my life.
TABLE OF CONTENTS

LIST OF TABLES ................................................................................................................................. v

LIST OF FIGURES ............................................................................................................................... vi

CHAPTER 1. INTRODUCTION ............................................................................................................. 1

1.1 Lid-Driven Cavity .............................................................................................................................. 2

1.2 Other geometries of lid-driven cavity ............................................................................................. 4

1.2.1 Rectangular LDC ......................................................................................................................... 5

1.2.2 Trapezoidal LDC ......................................................................................................................... 7

1.2.3 Triangular LDC ........................................................................................................................... 10

1.2.4 Semi-circular LDC ...................................................................................................................... 11

1.2.5 Annular wedge LDC ................................................................................................................... 15

1.3 Problem statement .......................................................................................................................... 18

1.4 Scope of thesis ................................................................................................................................. 18

CHAPTER 2. THEORY ........................................................................................................................... 21

2.1 Governing equation .......................................................................................................................... 21

2.1.1 Conservation of mass .................................................................................................................. 21

2.1.2 Conservation of momentum ....................................................................................................... 22

2.2 Stream function ............................................................................................................................... 23

2.3 Flow development and the vortexes .............................................................................................. 24
<table>
<thead>
<tr>
<th>Chapter</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>3.1 OpenFOAM</td>
<td>26</td>
</tr>
<tr>
<td></td>
<td>3.2 Discretization scheme</td>
<td>27</td>
</tr>
<tr>
<td>4</td>
<td>4.1 Cases evaluated</td>
<td>30</td>
</tr>
<tr>
<td></td>
<td>4.2 Grid-independent study</td>
<td>34</td>
</tr>
<tr>
<td></td>
<td>4.3 Time-independent study</td>
<td>35</td>
</tr>
<tr>
<td></td>
<td>4.4 Flow structures for Reynolds number of 100</td>
<td>37</td>
</tr>
<tr>
<td></td>
<td>4.5 Flow structures for Reynolds number of 400</td>
<td>39</td>
</tr>
<tr>
<td></td>
<td>4.6 Flow structures for Reynolds number of 1000</td>
<td>42</td>
</tr>
<tr>
<td></td>
<td>4.7 Comparison of stream function values</td>
<td>45</td>
</tr>
<tr>
<td></td>
<td>4.8 Stokes flow</td>
<td>46</td>
</tr>
<tr>
<td></td>
<td>4.9 Velocity profiles</td>
<td>48</td>
</tr>
<tr>
<td>5</td>
<td>5. CONCLUSIONS</td>
<td>51</td>
</tr>
<tr>
<td></td>
<td>BIBLIOGRAPHY</td>
<td>52</td>
</tr>
</tbody>
</table>
LIST OF TABLES

Table 4.1: Specifications of Each Annular Wedge Cavity......................................................... 31

Table 4.2: Ψ_{min} and Ψ_{max} Values with Their Locations for Re=100 ...................................... 38

Table 4.3: Ψ_{min} and Ψ_{max} Values with Their Locations for Re=400 ........................................ 41

Table 4.4: Ψ_{min} and Ψ_{max} Values with Their Locations for Re=1000 ................................. 44

Table 4.5: Ψ_{min} and Ψ_{max} Values with Their Locations for Stokes Flow .............................. 48
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>Schematic diagram of lid-driven cavity</td>
<td>2</td>
</tr>
<tr>
<td>1.2</td>
<td>Comparison of u-velocity along vertical lines through geometric center</td>
<td>4</td>
</tr>
<tr>
<td>1.3</td>
<td>Flow structures for different Reynolds number with $\Psi_{\text{min}}$ values</td>
<td>6</td>
</tr>
<tr>
<td>1.4</td>
<td>The velocity profile for rectangular cavities in y-direction</td>
<td>7</td>
</tr>
<tr>
<td>1.5</td>
<td>Streamline contour plots of different Reynolds number for $\theta = 60$</td>
<td>8</td>
</tr>
<tr>
<td>1.6</td>
<td>The contour plot for different angle $\theta$</td>
<td>9</td>
</tr>
<tr>
<td>1.7</td>
<td>Streamline contour plots and isotherm plots for $\text{Re} = 100$</td>
<td>11</td>
</tr>
<tr>
<td>1.8</td>
<td>Streamline contour plots for different Reynolds number</td>
<td>12</td>
</tr>
<tr>
<td>1.9</td>
<td>The velocity profile for Reynolds number 500 and 1000</td>
<td>13</td>
</tr>
<tr>
<td>1.11</td>
<td>Contour plots for different time seconds for Reynolds number 1000</td>
<td>14</td>
</tr>
<tr>
<td>1.12</td>
<td>Different flow patterns for different opening angle and different radii ratio</td>
<td>15</td>
</tr>
<tr>
<td>1.13</td>
<td>Typical flow structures for different opening angle</td>
<td>16</td>
</tr>
<tr>
<td>1.14</td>
<td>Schematic of wedge angle</td>
<td>17</td>
</tr>
<tr>
<td>1.15</td>
<td>Effect on flow structures for different wedge angle in Stokes flow</td>
<td>17</td>
</tr>
</tbody>
</table>
Figure 1.1: Different annular wedge cavities with wedge angle and radii ratio .................. 19

Figure 4.1: Contour plots of square lid-driven cavity ......................................................... 30

Figure 4.2: Annular wedge cavity ....................................................................................... 32

Figure 4.3: Annular wedge cavities with preliminary simulation of Re=100 ....................... 33

Figure 4.4: Grid-independent study ..................................................................................... 35

Figure 4.5: Time-independent study ..................................................................................... 36

Figure 4.6: Flow structures for Reynolds number of 100 ................................................... 37

Figure 4.7: Flow structures for Reynolds number of 400 ................................................... 40

Figure 4.8: Flow structures for Reynolds number of 1000 ............................................... 43

Figure 4.9: Stream function values for Re=100 ................................................................. 45

Figure 4.10: Stream function values for Re=400 ............................................................... 45

Figure 4.11: Stream function values for Re=1000 ............................................................. 46

Figure 4.12: Flow structures for Stokes flow ..................................................................... 47

Figure 4.13: Velocity Profile in y-direction for Re=100 ...................................................... 49

Figure 4.14: Velocity Profile in y-direction for Re=400 ...................................................... 49

Figure 4.15: Velocity Profile in y-direction for Re=1000 ..................................................... 50
CHAPTER 1. INTRODUCTION

Due to the wide range of physical applications and rapid industrial revolutions, the flow problems in lid-driven cavity has been a very popular research subject for decades. It has been a benchmark problem to test various numerical schemes and to evaluate the accuracy and efficiency of numerical methods for incompressible Navier-Stokes equations. The main reason for such a value and popularity in the topic is because of the simplicity in the geometry, its two-dimensional flow region and simple boundary condition which leads to various approximations and solution methods, such as multigrid method [1, 24], finite element method [21], finite volume method [23] and the lattice Boltzmann method [15]. This regular geometry and the simple boundary condition have made this flow cavity a test case for many computational schemes and has provided a proper database to check and validate these computational schemes. The flow problems in lid-driven cavity have numerous industrial applications, such as the wet clutches [2], gas lubrication system [25], solar collectors [27], bearing and lubrication systems [25, 26] and in enhancement for thermal performance of heat exchangers [28]. With all these advantages, the lid-driven cavities have been used to test and validate various results.

With its simplicity in geometry and boundary conditions, various studies have been conducted for different geometries and for different aspect ratios. Even though there are huge advancements in the flow problems, the standard case is the fluid contained in a square domain with a moving lid which is used to analyze complex fluid characteristics; this standard case is named Lid-Driven Cavity.
1.1 Lid-Driven Cavity

In computational fluid dynamics for the validation of numerical methods and techniques, the square lid-driven cavity flow is considered as a benchmark problem. The motion of the fluid inside the square or any lid-driven cavity is induced by the translation of the top wall. The regular square lid-driven cavity is comprised of a square-shaped box which has the top wall moving with a constant velocity and the other three walls stationary; these three fixed walls are subjected to a no-slip, zero-velocity condition. In three-dimensional analysis, for all the additional walls, the boundary condition remains the same and the geometry is extended to an additional z – direction. The geometry of the lid-driven cavity is as shown in the Figure 1.1

![Figure 1.1: Schematic diagram of lid-driven cavity](image)

The Reynolds number and the aspect ratio are the two-dimensional parameters that are used in governing the Newtonian fluids in these cavities.

Boundary condition of the lid-driven cavity:
• Top wall- (u=1) and (V=0) is applied
• Left, right and the bottom walls- (u=v=0) with no-slip velocity boundary condition

A benchmark work for this square lid-driven cavity is done by Ghia et al. [1] for a two-dimensional case using stream function vorticity formulation of Navier-Stokes equation, and the results for the flow at various Reynolds number from the range of 100 to 10000 is calculated (Table 1.1). The position, strength and location of the primary vortex and the corner eddies are also given [1]. This paper provides the comparison of u-velocity along vertical lines through geometric center and primary vortex center in order to understand the flow characteristics and compare and validate the results (Figure 1.2).

Table 1.1 $\Psi_{\text{min}}$ Values and Locations of Primary and Secondary Vortexes [1].

<table>
<thead>
<tr>
<th>Reynolds number</th>
<th>Primary</th>
<th>Secondary</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$\Psi_{\text{min}}$</td>
<td>Location(x,y)</td>
</tr>
<tr>
<td>100</td>
<td>-0.103243</td>
<td>0.6172, 0.7344</td>
</tr>
<tr>
<td>400</td>
<td>-0.113909</td>
<td>0.5547, 0.6055</td>
</tr>
<tr>
<td>1000</td>
<td>-0.117929</td>
<td>0.5313, 0.5625</td>
</tr>
</tbody>
</table>
Figure 1.2 Comparison of u-velocity along vertical lines through geometric center [1].

1.2 Other geometries of lid-driven cavity

Other than the most standard testing and flows in the square lid-driven cavity, there are many studies that have been carried out for different shapes of geometries, and many changes have been done to the boundary conditions due to many research needs; some of the many different geometries are

- Rectangular LDC
- Triangular LDC
- Trapezoidal LDC
- Semi-circular LDC
- Annular wedge LDC
Even though many techniques are used, this investigation has been limited to the moving lid being on the top of the cavity.

1.2.1 Rectangular LDC

The rectangular cavity is the second most used research topic for fluid flow in lid-driven cavity so far. There are many aspect ratios that are made with many changes in the boundary conditions, such as cavity with additional heat on the lid surface and also on the bottom surface; the top lid is kept fixed and the bottom layer is driven. There have been many remarkable studies carried out for a rectangular cavity; one such is presented in a paper by Chen and Cheng [12]. That paper carried out numerical study to investigate the effect of heat transfer characteristics and the shape of the cavity in a lid-driven cavity. Based on the that, many geometries along with many aspect ratios were considered, where the importance is given to the rectangular-shaped cavity. In that paper, the value of Reynolds number varies from 100 to 1800, and also on comparing all the other geometries, the rectangular geometry produces the strongest flow and triangular is the weakest when the Reynolds number is fixed [12]. The Nusselt number average is also very high in the rectangular cavities when compared to other geometries such as circular and triangular [12].

The $\Psi_{\text{min}}$ value change for different Reynolds numbers and the typical flow structures are published. From the below Figure 1.3, it is evident that in this type of method there is no formation of secondary or corner eddies. This can also be confirmed through the marking of stream function maximum value as 0 for all the different geometries of rectangular cavities. The flow structures for different geometry of the rectangular cavity are listed below from [12].
6

Figure 1.3: Flow structures for different Reynolds number with Ψ_min values [12].

The next research paper is Migeon et al. [13], where novel hydrodynamic simulations were carried out for the cavities with the aspect ratio of 2:1 and with the Reynolds number of 1000 was tested and compared with the standard square cavity [13]. In that paper the velocity profiles in y and x directions along the cavity are also published for rectangular geometry in order to validate the results (Figure 1.4).
1.2.2 Trapezoidal LDC

The next geometry that has been reviewed and studied is the trapezoidal-shaped cavities. Extensive research work has been done for different geometries of trapezoids; for instance, in the paper by Ting Zhang et al. [15], a two-dimensional isosceles trapezoidal cavity is taken and simulated with an incompressible lattice Bhavnagar–Gross–Krook model. Different trapezoidal cavities with different angle of $\theta$ are considered for Reynolds numbers varying from 100 to 15000.
and simulated [15]. The time sec is done as long as \( t=4500000 \) and the stream function minimum and the maximum values for different geometries are mentioned. The velocity profile in \( x \) and \( y \) directions are also published for validation [15]. It is concluded that by increasing the Reynolds number after a certain level the flow inside the cavity becomes very complex and the number of vortexes inside the cavity increases (Figure 1.5 and 1.6).

Figure 1.5: Streamline contour plots of different Reynolds number for \( \theta = 60 \) [15].
Figure 1.6: The contour plot for different angle \( \theta \) [15].
The paper by Hasib et al. [14] is a numerical study done in order to investigate the effects of tilt angles in trapezoidal cavity with a moving lid. In addition to that, the wall is heated either on top or the bottom and a series of Reynolds numbers from 0.1 to 10000 is conducted and the typical flow structures are published.

1.2.3 Triangular LDC

There are many triangular cavities as well based on different angles of triangle and aspect ratio. In the paper Chen and Cheng [17] the study is concerned with the lid oscillation and buoyancy effects on the periodic flow pattern with mixed convection in a triangular cavity [17]. In this, the stream function–vorticity formulation like in Ghia et al. [1] is used; the focus is given to the flow structure differences by oscillating cavity mainly. The direction of movement of the lid is varied in some cases and the values of stream function values are given for different Reynolds numbers. It is concluded that the oscillation does not give any changes in the calculated average Nusselt number [17].

The paper by Gonzalez et al. [16] published the theoretical and experimental results of the three-dimensional model instability of steady laminar two-dimensional states that are developed in isosceles triangular lid-driven cavity [16], and the results are plotted (Figure 1.7)
1.2.4 Semi-circular LDC

A few research topics are focused on a semi-circular cavity when compared to other geometries seen so far. Stokes and viscous flows are tested and published for different angles of semi-circular cavities, for example, the paper by Glowinski et al. [18], in which the focus is on the investigation of the capability of a finite element bases methodology in handling incompressible viscous flow for a series of large Reynolds numbers for curved boundaries and geometries.
The formation of primary, secondary and corner eddies is captured and published; results are calculated for Reynolds number around 6600 by numerical experiments for steady-state condition.

Figure 1.8: Streamline contour plots for different Reynolds number [18].
The stream function values are plotted and the change in stream function given for different Reynolds number, and the velocity profile is plotted (Figure 1.9)

\[ \text{Re}=500 \ (h_y = 1/128) \quad \text{Re}=1000 \ (h_y = 1/128) \]

Figure 1.9: The velocity profile for Reynolds number 500 and 1000 [18].

The paper by Mercan and Atalik [19] explores the vorticity transport formulation, and the second-order finite difference numerical method is adopted for high Reynolds numbers. Arc-shaped lid-driven flow is taken for a series of Reynolds numbers up to 8000 [19]. In the paper the effect of aspect ratio and angle ratio on the flow structure is studied and results are published.

The stream function minimum and maximum values are mentioned for every cavity and the vortex strength is discussed for cavities with different angles. The time seconds is increased and the changes in the formation of the stable vortexes are observed and results are given. The velocity along the x and y directions along the length of the cavity is examined and published in order to validate the results not given in these papers; the different streamline contour plots for different angles are given in figure 1.10 [19]
Figure 1.10: Contour plots for different angles and different Reynolds numbers.

Figure 1.11 shows the results for different cavity angles and for different Reynolds numbers for the semi-circular shape; some results are carried out for a long time in order to see the variation in the flow structures and vortices for different geometries [19].

Figure 1.11: Contour plots for different time seconds for Reynolds number 1000 [19].
1.2.5 Annular wedge LDC

Among all the other cavity geometries the least literature available is for the cavity with the annular wedge shape. Some papers are available for Stokes and viscous flow, but the Reynolds number for the flow structure is not mentioned. The paper by Niida and Yoshida [4], uses a relaxation technique to obtain the stream function and velocity components numerically in annular sectors, which are bounded by a pair of co-axial circular walls. In that research, the outer wall is rotating with an angular velocity, which is different from other regular lid-driven cavities, and the results are obtained and given as the typical flow pattern for different angle and different opening angle in annular wedge cavities with no Reynolds number (Figure 1.12) and variation in stream function is mentioned [4].

![Figure 1.12: Different flow patterns for different opening angle and different radii ratio [4].](image-url)
Rotem et al. [10] consider the various cross-sectional shapes of circular angle sector for viscous flow from a complete range of $\pi/12$ to $\pi$; this is done numerically and compared to the results with the experimental data [10]. The paper is not concerned about the radii ratio of the annular; instead, it tells about the typical flow structure for different angles (Figure 1.13).

![Diagram](image)

**Figure 1.13:** Typical flow structures for different opening angle [10].

Bilgil and Dolek [11] analyze the two-dimensional Stokes flow in annular wedge cavities for different cavity angles and for different speed ratio (Figure 1.4). The speed ratio of top and bottom lids and the cavity angle are considered as two important factors of the flow structures inside the cavity and results are published (Figure 1.15) [11]. The same Stokes flow is done in “Steady Stokes Flow in an Annular Cavity” by T. S. Krasnopolskaya et al. [29] where the results are done for top lid moving for different angle ratios.
Figure 1.14: Schematic of wedge angle [11].

Figure 1.15: Effect on flow structures for different wedge angle in Stokes flow [11].
1.3 Problem statement

The comprehensive study on effect of different angles and different Reynolds numbers on stream function values and the strength of the vortexes are not explained for annular-shaped cavities in any of the literature available. There are really no papers available for different Reynolds numbers for annular wedge lid-driven cavities in order to understand the flow structure when you increase the Reynolds number. The cavity angle and inside angle for different cavities are not mentioned in order to validate the typical flow structures for different angles. No velocity profiles along the x and y directions are available in order to compare and validate the results. The Navier-Stokes equation is not introduced in any of the results available in the literature. The simulation time and the mesh used are not mentioned in any of the papers for verifying purposes. No stable and steady flows are mentioned according to the different angles.

1.4 Scope of thesis

Annular wedge cavities of different angles (θ) are introduced with the constraint of total domain area being 1, which is comparable to the square cavity taken. There are totally five different annular wedge cavities with different angles considered and simulations are carried out for Reynolds numbers 100, 400 and 1000. The Navier-Stokes equation is introduced in these cavities and the stream function values are calculated. The vortex strength is discussed for different cavity angle and for different Reynolds number, and the effect of different angles to the stream function is studied.

First the results are calculated in order understand the correctness of the analysis for Reynolds number of 100 and compared to the values of the benchmark results of Ghia et al. [1]. Next the formation of vortexes with stream function value and how they change with time is
discussed. In order to confirm the optimal simulation time and grid used, the time-independent study and grid-independent study are carried out and results are plotted. The Reynolds number calculation for each and every simulation is mentioned for references. Other than the viscous flow, Stokes flow results for all the five geometries are carried out and the stream function values are defined. The velocity profile of $u$-velocity along vertical lines through geometric center and primary vortex center is calculated for all the results. The different cavities are shown in Figure 1.16 with the increasing wedge angle and the radius ratio mentioned.

Figure 1.16: Different annular wedge cavities with wedge angle and radii ratio. (a) AW1 (b) AW2. Continued on following page.
Figure 1.16: Continued. (c) AW3 (d) AW4 and (e) AW5.
CHAPTER 2. THEORY

2.1 Governing equation

In this thesis, a lid-driven cavity flow with a Newtonian fluid with constant viscosity and constant density but with varying Reynolds number that is changed by different velocity of the lid is considered. For this type of incompressible, isothermal, and Newtonian fluid the fundamental equations of the fluid dynamics are really based on conservation laws [3]. The conservation laws are

- Conservation of energy
- Conservation of mass
- Conservation of momentum

As already mentioned, the investigation is done in isothermal fluid and that is why conservation of energy is not considered. The governing equations for this fluid behavior are well established and defined as the conservation of mass and Navier-Stokes equations [3].

2.1.1 Conservation of mass

This law states that the amount of mass that enters the system should be the same as the amount of mass leaving the system. This law states that the mass of any system is conserved throughout the process, as the mass cannot be created nor destroyed. In any closed system the total mass inside the system remains constant at any point of time [3]. The continuity equation is given as
The differential form of the continuity equation is stated as
\[
\frac{\partial \rho}{\partial t} + \mathbf{V} \cdot (\rho \mathbf{U}) = 0
\] (2)

In this investigation, the case is considered as an incompressible, steady-state flow, so the density \( \rho \) of the fluid reduces to
\[
\nabla \cdot \mathbf{u} 
\] (3)

2.1.2 Conservation of momentum

Conservation of momentum is nothing but a fundamental law which states that, if there are no external forces acted on the system, then the momentum of a system is constant throughout the whole system [3].

The X-momentum equation is given by
\[
\frac{\partial (\rho \mathbf{U})}{\partial t} + \frac{\partial (\rho \mathbf{U} \mathbf{U})}{\partial x} + \frac{\partial (\rho \mathbf{U} \mathbf{V})}{\partial y} + \frac{\partial (\rho \mathbf{U} \mathbf{W})}{\partial z} = - \frac{\partial p}{\partial x} + \frac{1}{Re} \left[ \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} \right] 
\] (4)

The Y-momentum equation is given by
\[
\frac{\partial (\rho \mathbf{V})}{\partial t} + \frac{\partial (\rho \mathbf{U} \mathbf{V})}{\partial x} + \frac{\partial (\rho \mathbf{V} \mathbf{V})}{\partial y} + \frac{\partial (\rho \mathbf{V} \mathbf{W})}{\partial z} = - \frac{\partial p}{\partial y} + \frac{1}{Re} \left[ \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} \right] 
\] (5)

where

\[
U = \frac{u}{U_\infty} \quad V = \frac{v}{U_\infty} \quad X = \frac{x}{L} \quad Y = \frac{y}{L} \quad \tau = \frac{t U_\infty}{L} \quad P = \frac{p}{\rho U_\infty} \quad Re = \frac{U_\infty L}{v}
\]
By applying Newton’s second law of motion to a fluid domain, the Navier-Stokes equations are obtained [3]. The Navier-Stokes equation for an incompressible fluid is given as

$$\frac{du}{dt} + u \cdot \nabla u = -\frac{p}{\rho} + g + \nu \nabla^2 u$$  \hspace{1cm} (6)

where

$u =$ velocity vector

$\nu =$ Kinematic viscosity

$g =$ Acceleration due to gravity

### 2.2 Stream function

For any incompressible (divergence-free) flows, stream function values are defined. The derivatives of the stream functions are derived from the flow velocity components and they are used to plot the streamlines of a system which in turn traces the path of the particles in a steady flow [20]. The mathematical derivation of a stream function from the known velocity components $u$ and $v$ are as follows;

To define a stream function, $\Psi(x, y)$, such that

$$U = \frac{\partial \Psi}{\partial y} \hspace{1cm} \text{and} \hspace{1cm} V = -\frac{\partial \Psi}{\partial x}$$  \hspace{1cm} (7)

By definition, $\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$  \hspace{1cm} (8)

So, the field is non-divergent (incompressible flow divergence of flow velocity is zero).

From (7) $U = \frac{\partial \Psi}{\partial y}$ ,

$$\Psi = \int_{y_0}^{y} u \, dy + a(x)$$  \hspace{1cm} (9)
and therefore

\[ \frac{\partial \psi}{\partial x} = \int_{y_0}^{y} \frac{\partial u}{\partial x} dy + \frac{\partial a}{\partial x} \]  

(10)

Substitute Equation (8) in (10) and we get

\[ \frac{\partial \psi}{\partial x} = \int_{y_0}^{y} -\frac{\partial v}{\partial y} dy + \frac{\partial a}{\partial x} \]  

(11)

\[ \frac{\partial \psi}{\partial x} = v(x, y) + v(x, y_0) + \frac{\partial a}{\partial x} \]  

(12)

But also, by definition, \( \frac{\partial \psi}{\partial x} = -v(x, y) \)

(13)

therefore \( \frac{\partial a}{\partial x} = -v(x, y_0) \)

(14)

From (9) and (14),

\[ \Psi = \int_{y_0}^{y} u(x, y) dy - \int_{x_0}^{x} v(x, y_0) dx \]  

(15)

2.3 Flow development and the vortexes

As already discussed, the flow inside a lid-driven cavity is developed by the driven lid. In this investigation the Reynolds number is defined by the velocity of the lid. The ratio of inertial forces to viscous forces is called as the Reynolds number, which is a dimensionless expression; in this particular annular wedge cavity the Reynolds number is defined as

\[ Re = \frac{\rho \Omega a (b-a)}{\mu} \]  

(16)
where

Re= Reynolds number

ρ= Density

Ω= Angular velocity

a= Radius of the inner circle

b= Radius of the outer circle

µ= Viscosity of the fluid

This Reynolds number calculation is used to define different values by changing the velocity. In a lid-driven cavity, the fluid is enclosed inside a cavity and the top wall is driven with a tangential velocity; as the top wall is driven with a velocity, the fluid near the wall exerts a viscous force and the flow takes a recirculation within the cavity. By increased shear stress inside the fluid it tends to form vortexes; a primary vortex is formed or built towards the center and as the primary eddy grows and begins recirculating, it gives rise to a secondary corner eddy. After a certain amount of time there won’t be any changes in the flow structure and the flow takes a steady state [1].

In order to analyze the flow structure inside the cavity which has been a subject of research for a long time, it is assumed that in the transverse direction the flow is considered as infinitely long, and the two-dimensional flow is steady inside the cavity.
CHAPTER 3. COMPUTATION METHODOLOGY

3.1 OpenFOAM

In this investigation in order to carry out the simulations, open-source CFD software was used; OpenFOAM stands for “Open-Source Field Operation and Manipulation,” which is benchmarked and verified by ESI Group and can solve complex scientific and engineering problems. It is designed for computational continuum mechanics [8]. OpenFOAM is capable of solving both laminar as well as turbulent flow types, heat transfer problems, electromagnetic as well as discrete particle problems [8]. In OpenFOAM the traditional monolithic software design and the user coding extensions are replaced by discretization operators and physical models in library form, where it implements all the components of mesh handling, solver support and linear systems. All these tools are built into this system is a big advancement in the CFD tools.

The mesh handling done in OpenFOAM is a polyhedral type of mesh; in this polyhedral type, a cell is considered by the faces closing the volume, which are vectors [7]. OpenFOAM uses the finite volume approach. From the solvers in OpenFOAM, icoFoam (which is designed to solve the incompressible Navier-Stokes equations with a finite volume approach) is employed in this investigation.
### 3.2 Discretization scheme

As OpenFOAM uses finite volume method, that approach is used in this thesis as it is a mostly used way for discretizing and solving viscous flows [7]. In this approach the domain is divided into finite discrete domain and cells that which are the control volumes and after that even the time interval is divided into many time steps and solved over the time [7]. By using the integral formulation of the Navier-Stokes and continuity equations, the governing equations are discretized. By using this integral form of governing equation for each cell and control volume, the global governing equations are obtained and later the algebraic equations are obtained by discretized governing equations [7].

For all this the Navier-Stokes equation has to solved; in order to solve the Navier-Stokes equation in OpenFOAM, it requires numerical technique for coupling pressure and momentum quantities [7]. This technique is done by namely three algorithms:

- SIMPLE
- PISO
- PIMPLE

We consider the momentum equation for this type due to the fact that the density is constant, so there is no need for solving the energy equation; instead, solving the momentum equation is enough. In this case there are four quantities in the momentum equation: the three velocity components (Ux, Uy and Uz) and the pressure, so as there are four unknowns and just three equations. Another equation has to be used, which is the mass conservation equation. As there is no pressure included in mass conservation equation, there is a need of some special method to solve momentum equation in this case, which is also known as pressure momentum coupling problem [7]. In order to get rid of this problem, the divergence operator is applied to the momentum
equation and for that to be done a semi-discretization is used. In this the time derivative is
discretized and the space derivatives are in partial differential form; by this method the mass
conservation equation is used to eliminate the terms and come up with the known Poisson equation
for the pressure P [7]. Now there is equation for momentum and pressure, and they are solved
sequentially.

In order to fulfill the mass conservation and also the case condition, a pressure and
momentum field should be defined; this is achieved by pressure momentum coupling algorithms:
SIMPLE, PISO, and PIMPLE


  In OpenFOAM this algorithm is used for steady-state analyses.

- PISO: Pressure-Implicit-Split-Operator.

  In OpenFOAM this algorithm is used for transient calculation. In this type, the calculation
  is limited in the time step based on the Courant number.

- PIMPLE: Merged PISO–SIMPLE.

  This algorithm is the combination of both the algorithms, which allows for bigger time
  steps (Co >> 1).

In this thesis, the type of algorithm used is PISO.

PISO algorithm: The main variations of PISO from SIMPLE are the time derivation term
and also the consistent pressure velocity coupling equation, as PISO is supported with these two
criteria. The only need is to fulfill the stability criterion and there is no need to relax the fields and
equations [7]. In this algorithm the Courant number, a dimensionless number, should never be
greater than 1, because if the Courant number is less than 1 then the information from one cell can
only be reached to the next neighbor cell within one time step. If not, the information can reach to
second or more neighboring cells, which is not acceptable based on explicit aspects. In general, for the Courant number is calculated by

$$Co = \frac{U \Delta t}{\Delta x}$$

where U= cell velocity, Δt= time step and Δx= distance between the cells.

But in OpenFOAM, Δx is calculated by cell volume instead of the cell distance. For cases like where the mesh is too refined, there is increased velocity and time step; then that increases the Courant number above 1 and that limits the whole simulation, so to satisfy it the time step must be adjusted based on the velocity and the mesh used in the simulation [7].
CHAPTER 4. RESULTS AND DISCUSSIONS

4.1 Cases evaluated

In this thesis the first work has been done with the standard square lid-driven cavity and the results are validated with the Ghia et al. paper [1] for the Reynolds numbers of 100, 400 and 1000 (Figure 4.1)

Figure 4.1: Contour plots of square lid-driven cavity. (a) Reynolds number 100 (b) Reynolds number 400 (c) Reynolds number 1000.
Later, after validating results for square cavity, as discussed before, five different cavities are taken into account for this investigation in annular wedge cavity flows, as AW1, AW2, AW3, AW4 and AW5. These cavities are considered with five different angles in radians which are 0.2, 0.4, 0.5, 0.67 and 1 respectively in order to analyze the flows in cavities which are least deviated from square cavity and all the way up to more curved cavity. These cavities were taken with two main constraints: the side length of the cavity is kept as 1 and also the total domain area of the cavities is kept as 1. The specifications of the different cavities are listed below in Table 4.1.

Table 4.1: Specifications of Each Annular Wedge Cavity

<table>
<thead>
<tr>
<th>Wedge Type</th>
<th>AW1</th>
<th>AW2</th>
<th>AW3</th>
<th>AW4</th>
<th>AW5</th>
</tr>
</thead>
<tbody>
<tr>
<td>α</td>
<td>90.64</td>
<td>91.43</td>
<td>91.91</td>
<td>92.87</td>
<td>96.72</td>
</tr>
<tr>
<td>β</td>
<td>88.96</td>
<td>88.09</td>
<td>87.71</td>
<td>87.14</td>
<td>86.18</td>
</tr>
<tr>
<td>Θ</td>
<td>11.46</td>
<td>22.91</td>
<td>28.63</td>
<td>38.21</td>
<td>57.32</td>
</tr>
<tr>
<td>Lid length</td>
<td>0.90</td>
<td>0.80</td>
<td>0.75</td>
<td>0.66</td>
<td>0.50</td>
</tr>
<tr>
<td>Outer wall length</td>
<td>1.1</td>
<td>1.2</td>
<td>1.25</td>
<td>1.33</td>
<td>1.5</td>
</tr>
</tbody>
</table>

Even though the cavities are considered with varying angles (θ), in this investigation they are characterized by the varying (α). For understanding the labels that are the used in the annular wedge cavities, see the notations in Figure 4.2.
Figure 4.2: Annular wedge cavity.

Where

R2- Radius of the outer circle
R1- Radius of the inner circle
Θ- Wedge angle
α- Wedge inner angle
Β- Wedge outer angle

The annular wedge cavities are taken into account for the Reynolds number of 100 which are calculated by different calculations for each cavity. These simulations are done without any study about the optimal simulation time and grid size that has to be used in order to check the correctness of the analysis. The preliminary simulations are carried out to check the flow structures inside the cavities and to compare these cavities with square cavity and see the resulting flow structures (Figure 4.3).
Figure 4.3: Annular wedge cavities with preliminary simulation of Re=100. (a) AW1 (b) AW2 (c) AW3 (d) AW4 and (e) AW5.
The above results were carried out with the grid size of 60*60. Now that the simulations are stable, the mesh size and the simulation time are to be calculated in order to get the more stable flow structures, so the grid-independent and time-independent studies were carried out.

4.2 Grid-independent study

In this investigation, before running the simulations in all the cavities for different Reynolds numbers, a grid-independent study was carried out to check whether the grid size used will have the least error in the calculation of stream function ($\Psi_{\text{min}}$) values. For this to be achieved, simulations where done from the least grid size 60*60 and all the way up to the grid size of 240*240 in multiples of 30 for the most deviated angle from the square cavity, which is the angle of ($\alpha = 96.72$) for the Reynolds numbers 100, 400, and 1000 with uncalculated simulation time.

$\Psi_{\text{min}}$ error values are calculated for mesh 60*60 – mesh 90*90, mesh 90*90 – mesh 120*120, mesh 120*120 – mesh 150*150, mesh 150*150 – mesh 180*180, mesh 180*180 – mesh 210*210 and mesh 210*210 – mesh 240*240. These $\Psi_{\text{min}}$ error values are calculated by the formula:

$$\Psi_{\text{min}} \text{ Error} = \left| \frac{\Psi_{\text{min}}(\text{Smaller Mesh}) - \Psi_{\text{min}}(\text{Bigger Mesh})}{\Psi_{\text{min}}(\text{Bigger Mesh})} \right|$$

The calculated stream function error values of all the Reynolds numbers for different meshes are plotted in a graph to analyze which mesh gives the stable values and which can be used for all the simulation after this time (Figure 4.4).
From this study, the stream function error is more till the mesh size of 180*180 and the $\Psi_{\text{min}}$ error is very negligible from 180*180 to 240*240, so in order to get the exact stable results, all the simulations from now are carried out with the mesh of 240*240.

### 4.3 Time-independent study

After the grid-independent study, the suitable mesh was selected. Now the simulation time for the cavities was to be checked to see whether the values obtained converge and flow inside the cavity at a stable state. For this the simulations are executed for the worst-case scenario which is more deviated from the square cavity ($\alpha = 96.72$) for the Reynolds numbers 100, 400 and 1000 with the calculated mesh of 240*240 for 10 sec of simulation time. Stream function minimum is a value that explains the strength of the primary vortex formed in a cavity. A steady flow should
have the $\Psi_{\min}$ value almost constant after some simulation time, and stream function maximum value gives the strength of the corner eddy formed over the simulation time. Plotting the stream function value over the simulation time helps to understand the stability of the flow based on the $\Psi_{\min}$ value deviation over the increase in the simulation time. Figure 4.5 is plotted over the $\Psi_{\min}$ value of all the Reynolds numbers against the simulation time.

![Graph](image)

Figure 4.5: Time-independent study

From Figure 4.5, the error or change in the values over the time is completely negligible after 5 sec of simulation time and this clearly explains that the flow obtained steady state, which implies that the simulation time of 10s used in the simulations is enough to analyze the flow structure inside the cavity.
4.4 Flow structures for Reynolds number of 100

The simulations were executed for the calculated simulation time of 10sec and for the Mesh of 240*240 for all the annular wedge cavities AW1, AW2, AW3, AW4 and AW5 with the Reynolds number of 100; the streamline contour plots are in Figure 4.6.

Figure 4.6: Flow structures for Reynolds number of 100. (a) AW1 (b) AW2 (c) AW3 (d) AW4 and (e) AW5.
From the above flow structures, it is seen that the primary vortex is formed in all the wedge cavities and there is a formation of corner eddies. From the comprehensive study of the stream function values it is clear that the strength of the primary vortex is reduced with increase in the angle from AW1 to AW5 and there is increase in corner eddy size with the increasing angle. In order to understand the strength of the primary vortex and the corner eddy to analyze the results, the stream function values are plotted for all the cavities. In addition to that, the values are compared to the values of standard square cavity to understand the strength of the vortexes in square and annular wedge cavities (Table 4.2).

Table 4.2: $\Psi_{\text{min}}$ and $\Psi_{\text{max}}$ Values with Their Locations for Re=100

<table>
<thead>
<tr>
<th>Angle</th>
<th>R2/R1</th>
<th>Length 1</th>
<th>Length 2</th>
<th>Side length</th>
<th>$\alpha$</th>
<th>$\beta$</th>
<th>$\psi_{\text{min}}$</th>
<th>Location (x, y)</th>
<th>$\psi_{\text{max}}$</th>
<th>Location (x, y)</th>
</tr>
</thead>
<tbody>
<tr>
<td>11.46</td>
<td>5.5/4.5</td>
<td>0.90</td>
<td>1.1</td>
<td>1</td>
<td>90.64</td>
<td>88.96</td>
<td>-0.096956</td>
<td>0.657147, 0.721521</td>
<td>1.32883e-05</td>
<td>1.03016, 0.0600803</td>
</tr>
<tr>
<td>22.91</td>
<td>3/2</td>
<td>0.80</td>
<td>1.20</td>
<td>1</td>
<td>91.43</td>
<td>88.09</td>
<td>-0.090313</td>
<td>0.691497, 0.704602</td>
<td>1.40424e-05</td>
<td>1.10883, 0.0565775</td>
</tr>
<tr>
<td>28.63</td>
<td>2.5/1.5</td>
<td>0.75</td>
<td>1.25</td>
<td>1</td>
<td>91.91</td>
<td>87.71</td>
<td>-0.086795</td>
<td>0.713846, 0.689357</td>
<td>1.3661e-05</td>
<td>1.1524, 0.0557402</td>
</tr>
<tr>
<td>38.21</td>
<td>2/1</td>
<td>0.66</td>
<td>1.33</td>
<td>1</td>
<td>92.87</td>
<td>87.14</td>
<td>-0.08061</td>
<td>0.739222, 0.671924</td>
<td>1.36389e-05</td>
<td>1.21276, 0.0514554</td>
</tr>
<tr>
<td>57.32</td>
<td>1.5/0.5</td>
<td>0.50</td>
<td>1.5</td>
<td>1</td>
<td>96.72</td>
<td>86.18</td>
<td>-0.066051</td>
<td>0.788191, 0.64001</td>
<td>1.24737e-05</td>
<td>1.32948, 0.056001</td>
</tr>
<tr>
<td>square</td>
<td>-</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>90</td>
<td>90</td>
<td>-0.103423</td>
<td>0.6172, 0.7344</td>
<td>1.74877e-06</td>
<td>0.0313, 0.0391</td>
</tr>
</tbody>
</table>
From the stream function minimum values in the table, the primary vortex for the square cavity is strongest and the AW5, which is of ($\alpha = 96.72$), is having the weakest primary vortex values; other cavities have the strength which gradually decreases with the increase in the angle. There is no correlation for the values of stream function maximum values as there is an increase in the values from the transition from square cavity to the AW1 and AW2, but it again falls down.

4.5 Flow structures for Reynolds number of 400

The simulations are carried out for the cavities with the simulation time of 10sec and mesh of 240*240 for Reynolds number of 400. This Reynolds number is achieved by changing the velocity of the lid in the previously mentioned formula. To understand the flow structures of Reynolds number of 400 and to differentiate the flow structures from the Reynolds number of 100, the stream function contour plots are posted for all the wedge cavities (Figure 4.7). The stream function minimum and the stream function maximum values of all the cavities with their locations are plotted in the Table 4.3. From the contour plots it is seen that the center of the primary vortex has moved towards the geometric center, and the primary vortex size has increased in all the annular wedge cavities; size differences of the primary and the secondary cavities are negligible when compared to the Reynolds number of 100. The primary vortexes are fully formed. Figure 4.7 are the streamline contour plots of the different cavities respectively.
Figure 4.7: Flow structures for Reynolds number of 400. (a) AW1 (b) AW2 (c) AW3 (d) AW4 and (e) AW5.
Table 4.3: $\Psi_{\text{min}}$ and $\Psi_{\text{max}}$ Values with Their Locations for Re=400

<table>
<thead>
<tr>
<th>Angle</th>
<th>R2/R1</th>
<th>Length 1</th>
<th>Length 2</th>
<th>Side length</th>
<th>$\alpha$</th>
<th>$\beta$</th>
<th>Primary $\Psi_{\text{min}}$ Location (x, y)</th>
<th>Secondary $\Psi_{\text{max}}$ Location (x, y)</th>
</tr>
</thead>
<tbody>
<tr>
<td>11.46</td>
<td>5.5/4.5</td>
<td>0.90</td>
<td>1.1</td>
<td>1</td>
<td>90.64</td>
<td>88.96</td>
<td>-0.1072, 0.603171, 0.574519</td>
<td>0.0027094, 0.966587, 0.117193</td>
</tr>
<tr>
<td>22.91</td>
<td>3/2</td>
<td>0.80</td>
<td>1.20</td>
<td>1</td>
<td>91.43</td>
<td>88.09</td>
<td>-0.09953, 0.646804, 0.54056</td>
<td>0.0026952, 1.03665, 0.107814</td>
</tr>
<tr>
<td>28.63</td>
<td>2.5/1.5</td>
<td>0.75</td>
<td>1.25</td>
<td>1</td>
<td>91.91</td>
<td>87.71</td>
<td>-0.09598, 0.671499, 0.520732</td>
<td>0.0026662, 1.07989, 0.108247</td>
</tr>
<tr>
<td>38.21</td>
<td>2/1</td>
<td>0.66</td>
<td>1.33</td>
<td>1</td>
<td>92.87</td>
<td>87.14</td>
<td>-0.08876, 0.705513, 0.486703</td>
<td>0.0025236, 1.13377, 0.0981785</td>
</tr>
<tr>
<td>57.32</td>
<td>1.5/0.5</td>
<td>0.50</td>
<td>1.5</td>
<td>1</td>
<td>96.72</td>
<td>86.18</td>
<td>-0.07323, 0.776448, 0.417562</td>
<td>0.0021103, 1.23761, 0.0799966</td>
</tr>
<tr>
<td>square</td>
<td>-</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>90</td>
<td>90</td>
<td>-0.113909, 0.5547, 0.6055</td>
<td>6.42352e-04, 0.8906, 0.1250</td>
</tr>
</tbody>
</table>

From the above stream function minimum and the stream function maximum values, which in turn gives the strength of the primary vortex and the strength of the right corner eddy, the vortex strength is still low for the cavity with the angle $\alpha = 96.72$ when compared to the square cavity. The relation between the stream function values and angles is still the same for this Reynolds number as the value of the stream function reduces with the increases in the angle. Still there is no correlation found between the stream function values.
4.6 Flow structures for Reynolds number of 1000

Now that the flow structures of Reynolds numbers 100 and 400 are executed and obtained from the simulations, now the simulations are carried out for the Reynolds number of 1000 for all the annular wedge cavities with the calculated simulation time of 10 secs and with the calculated grid size of 240*240. This Reynolds number is achieved by changing the velocity of the lid for all the cavities; the velocity of each cavity is different based on the geometries.

The contour plots of stream function for different cavities shows that the center of the primary vortex is moved towards the geometric center and the size of the primary vortex is increased a little further than the previous Reynolds numbers. The size of the primary vortex and right corner eddy looks the same irrespective of the inner wedge angle. (Inner wedge angle $\alpha$ is used at this point). In this Reynolds number there is a clear initiation in the formation of an upper left corner eddy in all the Annular Wedge cavities. The strength of the primary vortex is more increased in this Reynolds number.

The contour plot of the streamlines of annular wedge cavities AW1, AW2, AW3, AW4 and AW5 are pictured in Figure 4.8 with the values of stream function minimum and the stream function maximum are plotted in a Table 4.4.
Figure 4.8: Flow structures for Reynolds number of 1000. (a) AW1 (b) AW2 (c) AW3 (d) AW4 and (e) AW5.
### Table 4.4: $\Psi_{\text{min}}$ and $\Psi_{\text{max}}$ Values with Their Locations for Re=1000

<table>
<thead>
<tr>
<th>Angle</th>
<th>R2/R1</th>
<th>Length 1</th>
<th>Length 2</th>
<th>Side length</th>
<th>$\alpha$</th>
<th>$\beta$</th>
<th>$\Psi_{\text{min}}$</th>
<th>Location (x, y)</th>
<th>$\Psi_{\text{max}}$</th>
<th>Location (x, y)</th>
</tr>
</thead>
<tbody>
<tr>
<td>11.46</td>
<td>5.5/4.5</td>
<td>0.90</td>
<td>1.1</td>
<td>1</td>
<td>90.64</td>
<td>88.96</td>
<td>-0.11188</td>
<td>0.578873, 0.532602</td>
<td>0.01777</td>
<td>0.945157, 0.102808</td>
</tr>
<tr>
<td>22.91</td>
<td>3/2</td>
<td>0.80</td>
<td>1.20</td>
<td>1</td>
<td>91.43</td>
<td>88.09</td>
<td>-0.10441</td>
<td>0.623434, 0.494328</td>
<td>0.01785</td>
<td>1.01492, 0.0918466</td>
</tr>
<tr>
<td>28.63</td>
<td>2.5/1.5</td>
<td>0.75</td>
<td>1.25</td>
<td>1</td>
<td>91.91</td>
<td>87.71</td>
<td>-0.10095</td>
<td>0.648407, 0.474384</td>
<td>0.01771</td>
<td>1.05884, 0.0870474</td>
</tr>
<tr>
<td>38.21</td>
<td>2/1</td>
<td>0.66</td>
<td>1.33</td>
<td>1</td>
<td>92.87</td>
<td>87.14</td>
<td>-0.09392</td>
<td>0.683186, 0.436089</td>
<td>0.01719</td>
<td>1.11295, 0.0756402</td>
</tr>
<tr>
<td>57.32</td>
<td>1.5/0.5</td>
<td>0.50</td>
<td>1.5</td>
<td>1</td>
<td>96.72</td>
<td>86.18</td>
<td>-0.07881</td>
<td>0.752201, 0.353859</td>
<td>0.05101</td>
<td>1.22302, 0.0562057</td>
</tr>
</tbody>
</table>

square | - | 1 | 1 | 1 | 90 | 90 | - | 0.5313, 0.5625 | 1.751e-03 | 0.8594, 0.01094 |

The values of the stream function minimum are further increased for the annular wedge cavities; the angles $\alpha=90.64$, 91.43 and 91.91 have almost the strength of the standard square cavity. The relation between the angle and stream function minimum value is the same for this Reynolds number, as the value is decreasing as the angle is increased. Still there is no correlation between the different stream function maximum values.
4.7 Comparison of stream function values

The stream function minimum values for all the cavities are compared with the standard square cavity for all the Reynolds numbers such as 100, 400 and 1000 (Figure 4.9–4.11).

![Figure 4.9: Stream function values for Re=100.](image)

![Figure 4.10: Stream function values for Re=400.](image)
A parabolic curve is observed in all the Reynolds numbers for the different annular wedge cavities according to their different angles $\alpha$. The cavity with the angle $\alpha = 90.64$ forms the strongest vortex and the angle with $\alpha = 96.72$ forms the weakest vortex when compared to the other cavities in all the Reynolds numbers. The curve is valid with the equation given and regression value very closer to 1. By this it is concluded that the change in the stream function Value is not only by the different angles but also by the different Reynolds numbers too.

### 4.8 Stokes flow

As Stokes flow is addressed in many studies of other geometries of lid-driven cavities, the Stokes flow for annular wedge cavities is addressed. It is seen that there is a formation of primary vortex and two fully sized corner eddies in all the annular wedge cavities (Figure 4.12). The values of the stream function minimum and the maximum with the locations are calculated and plotted in Table 4.5 for the understanding of Stokes flows in the considered cavities. The relation between the cavity angle and stream function minimum value is still related for this type of flow too.
Figure 4.12: Flow structures for Stokes flow. (a) AW1 (b) AW2 (c) AW3 (d) AW4 and (e) AW5.
Table 4.5: $\Psi_{\text{min}}$ and $\Psi_{\text{max}}$ Values with Their Locations for Stokes Flow

<table>
<thead>
<tr>
<th>Angle</th>
<th>R2/R1</th>
<th>Length 1</th>
<th>Length 2</th>
<th>Side length</th>
<th>$\alpha$</th>
<th>$\beta$</th>
<th>$\psi_{\text{min}}$ Location (x, y)</th>
<th>$\psi_{\text{max}}$ Location (x, y)</th>
</tr>
</thead>
<tbody>
<tr>
<td>11.46</td>
<td>5.5/4.5</td>
<td>0.90</td>
<td>1.1</td>
<td>1</td>
<td>90.64</td>
<td>88.96</td>
<td>-9.3047e-04, 0.55, 0.75333</td>
<td>3.38596e-08, 1.0588, 0.0461368</td>
</tr>
<tr>
<td>22.91</td>
<td>3/2</td>
<td>0.80</td>
<td>1.20</td>
<td>1</td>
<td>91.43</td>
<td>88.09</td>
<td>-8.7621e-06, 0.595, 0.723327</td>
<td>3.55914e-10, 1.14148, 0.0414173</td>
</tr>
<tr>
<td>28.63</td>
<td>2.5/1.5</td>
<td>0.75</td>
<td>1.25</td>
<td>1</td>
<td>91.91</td>
<td>87.71</td>
<td>-8.4419e-06, 0.62, 0.72</td>
<td>3.56624e-10, 1.18781, 0.0384308</td>
</tr>
<tr>
<td>38.21</td>
<td>2/1</td>
<td>0.66</td>
<td>1.33</td>
<td>1</td>
<td>92.87</td>
<td>87.14</td>
<td>-7.8520e-06, 0.655, 0.68999</td>
<td>3.53619e-10, 1.24721, 0.0491131</td>
</tr>
<tr>
<td>57.32</td>
<td>1.5/0.5</td>
<td>0.33</td>
<td>1</td>
<td>1</td>
<td>95.73</td>
<td>86.18</td>
<td>-6.511e-06, 0.72, 0.65333</td>
<td>3.4644e-10, 0.0744958, 0.036886</td>
</tr>
</tbody>
</table>

4.9 Velocity profiles

Figures 4.13-4.15 show the velocity profiles of Reynolds numbers 100, 400 and 100 in the y-direction through the geometric center compared to the standard square cavity which is denoted by the angle $\alpha=90$. These velocity profiles can be used for the validation of the cavities with this type of angles and geometries. The flow characteristics inside the cavity is understood for different Reynolds number by the transition from negative to positive magnitude. The differences in
velocity profiles due to different cavity angles are reduced in the Reynolds number of 1000; this can be due to the effect of increased velocity.

Figure 4.13: Velocity Profile in y-direction for Re=100.

Figure 4.14: Velocity Profile in y-direction for Re=400.
Figure 4.15: Velocity Profile in y-direction for Re=1000.
CHAPTER 5. CONCLUSIONS

Stream function value ($\Psi_{\text{min}}$) depends on the cavity angle ($\alpha$) and the used Reynolds number for all the simulations. A parabolic curve in stream function minimum values is observed for all the annular wedge cavities according to their angle $\alpha$. There is formation of new vortexes as the Reynolds number is increased. Flow structures of Reynolds numbers 100, 400 and 1000 for annular wedge cavities look almost like the flow structures of standard square cavity, only the strength of the vortexes is different. In Stokes flow, all the cavities have a primary vortex and two fully formed corner eddies for all cavities.

Future work in this research can be in flow structures in deeper cavities, which is, a side length more than 1 can be analyzed and studied to understand the flow structures inside it. Particle perturbations can be introduced to the current cavity and the multi-phase flow can be analyzed for this type of annular wedge cavities. Vortex formation topic can be studied in detail for how a vortex is formed for cavities with circular geometry. Flows can be analyzed for higher Reynolds number in order to compare the vortex strength after a certain Reynolds number.
BIBLIOGRAPHY


