

COMPARATIVE STUDY ON THE ORIFICE INDUCED VELOCITY FIELD

A THESIS SUBMITTED TO  
THE GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES  
OF  
MIDDLE EAST TECHNICAL UNIVERSITY

BY  
SONAT ÇELEBİ

IN PARTIAL FULFILLMENT OF THE REQUIREMENTS  
FOR  
THE DEGREE OF MASTER OF SCIENCE  
IN  
CIVIL ENGINEERING

JUNE 2014



Approval of the thesis:

**COMPARATIVE STUDY ON THE ORIFICE INDUCED VELOCITY FIELD**

submitted by **SONAT ÇELEBİ** in partial fulfillment of the requirements for the degree of **Master of Science in Civil Engineering Department, Middle East Technical University** by,

Prof. Dr. Canan Özgen

Dean, Graduate School of **Natural and Applied Sciences**

\_\_\_\_\_

Prof. Dr. Ahmet Cevdet Yalçın

Head of Department, **Civil Engineering**

\_\_\_\_\_

Assoc. Prof. Dr. Şahnaz Tiğrek

Supervisor, **Civil Engineering Dept., METU**

\_\_\_\_\_

**Examining Committee Members:**

Prof. Dr. A. Melih Yanmaz

Civil Engineering Dept., METU

\_\_\_\_\_

Assoc. Prof. Dr. Şahnaz Tiğrek

Civil Engineering Dept, METU

\_\_\_\_\_

Prof. Dr. A. Burcu Altan Sakarya

Civil Engineering Dept, METU

\_\_\_\_\_

Assoc. Prof. Dr. Mete Köken

Civil Engineering Dept., METU

\_\_\_\_\_

Dr. Taylan Ulaş Evcimen

Projects Manager, Koza Construction

\_\_\_\_\_

Date: 11.06.2014

**I hereby declare that all information in this document has been obtained and presented in accordance with academic rules and ethical conduct. I also declare that, as required by these rules and ethical conduct, I have fully cited and referenced all material and results that are not original to this work.**

Name, Last Name: Sonat Çelebi  
Signature:

## **ABSTRACT**

### **COMPARATIVE STUDY ON THE ORIFICE INDUCED VELOCITY FIELD**

Çelebi, Sonat

M.Sc., Department of Civil Engineering

Supervisor: Assoc. Prof. Dr. Şahnaz Tiğrek

June 2014, 118 pages

Orifices are important devices used in many hydraulic engineering applications. Therefore, investigation of flow characteristics upstream of orifices is a quite important phenomenon. One of the solution approaches is the potential flow solution. In this study, two different potential flow solutions presented by Shammaa et. al. and Bryant et. al are investigated and compared with the solution obtained by using computational fluid dynamics software, FLUENT. Both inviscid and volume of fluid (VOF) methods are used for the FLUENT solution. Solutions for the orifice induced velocity field upstream of the orifice are presented.

Keywords: orifice flow, volume of fluid, flow measurement, potential flow, computational fluid dynamics.

## ÖZ

### ORİFİS KAYNAKLI HIZ ALANININ KARŞILAŞTIRMALI İNCELENMESİ

Çelebi, Sonat

Yüksek Lisans, İnşaat Mühendisliği Bölümü

Tez Yöneticisi: Doç. Dr. Şahnaz Tiğrek

Haziran 2014, 118 sayfa

Orifisler, bir çok hidrolik mühendisliği uygulamalarında kullanılan önemli elemanlardır. Bu nedenle, orifis gerisindeki akımın incelenmesi çok önemli bir konudur. Bu inceleme için kullanılan metodlardan birisi de potansiyel akım metodudur. Bu çalışmada, Shammaa ve Bryant tarafından önerilen iki farklı potansiyel akım metodu çalışılmış ve bir hesaplamalı akışkanlar dinamiği yazılımı olan FLUENT programından elde edilen sonuçlar ile karşılaştırılmıştır. FLUENT çözümünde hem yapışkan olmayan (sürtünmesiz) hem de akışkan hacmi yöntemi kullanılmıştır. Orifis kaynaklı hız alanı için bulunan sonuçlar sunulmuştur.

Anahtar kelimeler: orifis akımı, akışkan hacmi yöntemi, debi ölçümü, potansiyel akım, hesaplamalı akışkanlar dinamiği.

To my family & friends

## **ACKNOWLEDGMENTS**

First of all, I would like to express my sincere gratitude to Assoc. Prof. Dr. Şahnaz Tiğrek, for her support, patience, encouragement and confidence on me in every stage of this study.

Secondly, I would like to thank Mr. Nevzat Çakmak for his useful help during this study.

I would like to present my deepest gratitude to my parents for their endless support during this study and my entire lifetime.

## TABLE OF CONTENTS

|                                                  |      |
|--------------------------------------------------|------|
| ABSTRACT.....                                    | v    |
| ÖZ .....                                         | vi   |
| ACKNOWLEDGEMENTS .....                           | viii |
| TABLE OF CONTENTS.....                           | ix   |
| LIST OF TABLES.....                              | xi   |
| LIST OF FIGURES .....                            | xii  |
| LIST OF SYMBOLS .....                            | xvii |
| CHAPTERS                                         |      |
| 1. INTRODUCTION.....                             | 1    |
| 1.1. Introduction and Scope of the Study .....   | 1    |
| 2. THEORETICAL BACKGROUND .....                  | 3    |
| 2.1. General Information.....                    | 3    |
| 2.2. Potential Flow .....                        | 5    |
| 2.2.1. Source or Sink Flows.....                 | 13   |
| 2.2.2. 3D Potential Flow – Shammaa Solution..... | 15   |
| 2.2.3. 3D Potential Flow – Bryant Solution.....  | 17   |
| 2.3. Fluent CFD Solution.....                    | 19   |
| 2.3.1. About ANSYS Fluent Software.....          | 19   |
| 2.3.2. ANSYS Fluent Inviscid Flow Solution ..... | 20   |
| 2.3.3. Modeling.....                             | 21   |
| 2.3.4. Meshing .....                             | 21   |
| 2.3.5. Volume of Fluid (VOF) Method.....         | 27   |
| 2.3.6. $k - \epsilon$ Method .....               | 29   |
| 3. RESULTS AND DISCUSSION .....                  | 33   |
| 3.1. General Information.....                    | 33   |
| 3.2. 2D Potential Flow Solution .....            | 35   |
| 3.3. 3D Potential Flow Solution – Shammaa.....   | 37   |
| 3.4. 3D Potential Flow Solution – Bryant.....    | 44   |
| 3.5. Fluent CFD Solution.....                    | 48   |

|                                                                        |     |
|------------------------------------------------------------------------|-----|
| 3.5.1. Inviscid Solution of the Çobanoğlu Model .....                  | 49  |
| 3.5.2. Inviscid Solution of the Enlarged Model .....                   | 58  |
| 3.5.3. Laminar Solution of the Enlarged Model .....                    | 67  |
| 3.5.4. $k - \varepsilon$ Solution of the Enlarged Model .....          | 76  |
| 3.5.5. VOF Solution of the Enlarged Model .....                        | 86  |
| 4. COMPARISON OF OBTAINED RESULTS .....                                | 97  |
| 4.1. Comparison of Obtained Results .....                              | 97  |
| 5. CONCLUSION.....                                                     | 103 |
| 5.1. Conclusion .....                                                  | 103 |
| REFERENCES.....                                                        | 104 |
| APPENDICES                                                             |     |
| A. MATLAB CODES .....                                                  | 106 |
| B. FLUENT RESIDUALS .....                                              | 109 |
| C. VOLUME OF FLUID $k - \varepsilon$ SOLUTION FOR ÇOBANOĞLU MODEL .... | 112 |

## LIST OF TABLES

### TABLES

|                                                                                            |    |
|--------------------------------------------------------------------------------------------|----|
| Table 2.1. Mesh size information.....                                                      | 25 |
| Table 2.2. Mass flux information for different mesh sizes .....                            | 26 |
| Table 2.3. Empirical constants of the $k - \epsilon$ method, Launder and Sharma(1974). ... | 31 |

## LIST OF FIGURES

### FIGURES

|                                                                                     |    |
|-------------------------------------------------------------------------------------|----|
| Figure 2.1. Orifice types according to shape of upstream edge .....                 | 3  |
| Figure 2.2. Orifice types according to nature of discharge .....                    | 4  |
| Figure 2.3. Orifice types according to size.....                                    | 4  |
| Figure 2.4. Orifice types according to size.....                                    | 8  |
| Figure 2.5. Orifice types according to size.....                                    | 9  |
| Figure 2.6. Orifice types according to size.....                                    | 11 |
| Figure 2.7. Streamlines and potentials representing source flows .....              | 14 |
| Figure 2.8. Flow axes used at Shammaa's solution .....                              | 16 |
| Figure 2.9. Flow axes used at Shammaa's solution .....                              | 17 |
| Figure 2.10. Flow axes used at Bryant's solution .....                              | 18 |
| Figure 2.11. Flow axes used at Bryant's solution .....                              | 19 |
| Figure 2.12. Front view of the coarse model.....                                    | 22 |
| Figure 2.13. Closer view of the orifice area of the coarse model .....              | 23 |
| Figure 2.14. Front view of the fine model.....                                      | 23 |
| Figure 2.15. Closer view of the orifice area of the fine model .....                | 24 |
| Figure 2.16. Front view of the finer model .....                                    | 24 |
| Figure 2.17. Closer view of the orifice area of the fine model .....                | 25 |
| Figure 2.18. Front view of the model setup .....                                    | 27 |
| Figure 3.1. Front view of the model setup .....                                     | 34 |
| Figure 3.2. Side view of the model setup .....                                      | 35 |
| Figure 3.3. Potential flow sink solution velocity potential vs. $x/d$ .....         | 36 |
| Figure 3.4. $V_r/U_0$ vs. $x/d$ .....                                               | 37 |
| Figure 3.5. Velocity potential $\phi$ vs. $r$ , for different $\theta$ values ..... | 38 |

|                                                                                                        |    |
|--------------------------------------------------------------------------------------------------------|----|
| Figure 3.6. $V_r/U_0$ vs. $r/d$ , with fixed $\theta$ value .....                                      | 39 |
| Figure 3.7. $V_r/U_0$ vs. $r/d$ , with fixed $\theta$ value .....                                      | 40 |
| Figure 3.8. $V_r/U_0$ vs. $r/d$ , with fixed $\theta$ value .....                                      | 40 |
| Figure 3.9. $V_r/U_0$ vs. $r/d$ , for different $\theta$ values.....                                   | 41 |
| Figure 3.10. Velocity potential $\phi$ vs. $r/d$ , with different $x$ values .....                     | 42 |
| Figure 3.11. $V_r/U_0$ vs. $z/d$ , with different $x$ values.....                                      | 43 |
| Figure 3.12. $V_r/U_0$ vs. $r/d$ , with different $x$ values .....                                     | 44 |
| Figure 3.13. Velocity potential $\phi$ vs. $x$ , with different $\theta$ values.....                   | 45 |
| Figure 3.14. $V_r/U_0$ vs. $x/d$ .....                                                                 | 46 |
| Figure 3.15. $V_r/U_0$ vs. $y/d$ , with fixed $x$ values .....                                         | 47 |
| Figure 3.16. $V_r/U_0$ vs. $z/d$ , with fixed $x$ values .....                                         | 48 |
| Figure 3.17. Pathlines by velocity magnitude .....                                                     | 50 |
| Figure 3.18. Close up of the orifice area for the pathlines by velocity magnitude ...                  | 50 |
| Figure 3.19. Contours of velocity magnitude.....                                                       | 51 |
| Figure 3.20. Close up of the orifice area for the contours of velocity magnitude ....                  | 51 |
| Figure 3.21. Contours of velocity magnitude.....                                                       | 52 |
| Figure 3.22. Close up of the orifice area for the contours of velocity magnitude ....                  | 52 |
| Figure 3.23. Velocity magnitude contours of the orifice area only .....                                | 53 |
| Figure 3.24. Pressure contours .....                                                                   | 54 |
| Figure 3.25. Pressure contours of the orifice area.....                                                | 54 |
| Figure 3.26. Velocity magnitude readings of the centerline velocity .....                              | 55 |
| Figure 3.27. $y$ - $z$ plane velocity magnitude on $z$ -axis for different $x$ values.....             | 55 |
| Figure 3.28. $y$ - $z$ plane velocity magnitude on $y$ -axis for different $x$ values .....            | 56 |
| Figure 3.29. $V_r$ vs. $z$ , for different $x$ values .....                                            | 57 |
| Figure 3.30. Velocity vectors for $x=0, 0.375\text{cm}, 0.75\text{cm}, 1.5\text{cm}, 3\text{cm}$ ..... | 57 |
| Figure 3.31. Pathlines by velocity magnitude .....                                                     | 58 |
| Figure 3.32. Close up of the orifice area for the pathlines by velocity magnitude ...                  | 59 |

|                                                                                                    |    |
|----------------------------------------------------------------------------------------------------|----|
| Figure 3.33. Contours of velocity magnitude.....                                                   | 60 |
| Figure 3.34. Close up of the orifice area for the contours of velocity magnitude.....              | 60 |
| Figure 3.35. Contours of velocity magnitude.....                                                   | 61 |
| Figure 3.36. Close up of the orifice area for the contours of velocity magnitude.....              | 61 |
| Figure 3.37. Velocity magnitude contours of the orifice area only.....                             | 62 |
| Figure 3.38. Pressure contours .....                                                               | 63 |
| Figure 3.39. Pressure contours of the orifice area.....                                            | 63 |
| Figure 3.40. Velocity magnitude readings of the centerline velocity .....                          | 64 |
| Figure 3.41. y-z plane velocity magnitude on z-axis for different x values.....                    | 64 |
| Figure 3.42. y-z plane velocity magnitude on y-axis for different x values .....                   | 65 |
| Figure 3.43. $V_r$ vs. $z$ , for different x values .....                                          | 66 |
| Figure 3.44. Velocity vectors for $x=0, 0.375\text{m}, 0.75\text{m}, 1.5\text{m}, 3\text{m}$ ..... | 66 |
| Figure 3.45. Pathlines by velocity magnitude .....                                                 | 67 |
| Figure 3.46. Close up of the orifice area for the pathlines by velocity magnitude ...              | 68 |
| Figure 3.47. Contours of velocity magnitude.....                                                   | 69 |
| Figure 3.48. Close up of the orifice area for the contours of velocity magnitude.....              | 69 |
| Figure 3.49. Contours of velocity magnitude.....                                                   | 70 |
| Figure 3.50. Close up of the orifice area for the contours of velocity magnitude.....              | 70 |
| Figure 3.51. Velocity magnitude contours of the orifice area only.....                             | 71 |
| Figure 3.52. Pressure contours .....                                                               | 72 |
| Figure 3.53. Pressure contours of the orifice area.....                                            | 72 |
| Figure 3.54. Velocity magnitude readings of the centerline velocity .....                          | 73 |
| Figure 3.55. y-z plane velocity magnitude on z-axis for different x values.....                    | 74 |
| Figure 3.56. y-z plane velocity magnitude on y-axis for different x values .....                   | 74 |
| Figure 3.57. $V_r$ vs. $z$ , for different x values .....                                          | 75 |
| Figure 3.58. Velocity vectors for $x=0, 0.375\text{m}, 0.75\text{m}, 1.5\text{m}, 3\text{m}$ ..... | 76 |
| Figure 3.59. Pathlines by velocity magnitude .....                                                 | 77 |

|                                                                                       |    |
|---------------------------------------------------------------------------------------|----|
| Figure 3.60. Close up of the orifice area for the pathlines by velocity magnitude ... | 78 |
| Figure 3.61. Contours of velocity magnitude.....                                      | 78 |
| Figure 3.62. Close up of the orifice area for the contours of velocity magnitude .... | 79 |
| Figure 3.63. Contours of velocity magnitude.....                                      | 80 |
| Figure 3.64. Close up of the orifice area for the contours of velocity magnitude .... | 80 |
| Figure 3.65. Velocity magnitude contours of the orifice area only .....               | 81 |
| Figure 3.66. Pressure contours .....                                                  | 82 |
| Figure 3.67. Pressure contours of the orifice area.....                               | 82 |
| Figure 3.68. Velocity magnitude readings of the centerline velocity .....             | 83 |
| Figure 3.69. z plane velocity magnitude on z-axis for different x values .....        | 83 |
| Figure 3.70. y-z plane velocity magnitude on y-axis for different x values .....      | 84 |
| Figure 3.71. $V_r$ vs. z, for different x values .....                                | 85 |
| Figure 3.72. Velocity vectors for $x=0, 0.375m, 0.75m, 1.5m, 3m$ .....                | 85 |
| Figure 3.73. Pathlines by velocity magnitude .....                                    | 86 |
| Figure 3.74. Close up of the orifice area for the pathlines by velocity magnitude ... | 87 |
| Figure 3.75. Contours of velocity magnitude.....                                      | 87 |
| Figure 3.76. Close up of the orifice area for the contours of velocity magnitude .... | 88 |
| Figure 3.77. Contours of velocity magnitude.....                                      | 89 |
| Figure 3.78. Close up of the orifice area for the contours of velocity magnitude .... | 89 |
| Figure 3.79. Velocity magnitude contours of the orifice area only .....               | 90 |
| Figure 3.80. Pressure contours .....                                                  | 91 |
| Figure 3.81. Pressure contours of the orifice area only.....                          | 91 |
| Figure 3.82. Velocity magnitude readings of the centerline velocity .....             | 92 |
| Figure 3.83. y-z plane velocity magnitude on z-axis for different x values.....       | 93 |
| Figure 3.84. y-z plane velocity magnitude on y-axis for different x values .....      | 93 |
| Figure 3.85. $V_r$ vs. z, for different x values.....                                 | 94 |
| Figure 3.86. Velocity vectors for $x=0, 0.375m, 0.75m, 1.5m, 3m$ .....                | 95 |

|                                                                                                                   |     |
|-------------------------------------------------------------------------------------------------------------------|-----|
| Figure 4.1. Cd vs Re from Yıldırım's study, Scrutinization of Flow Characteristics Through Orifices, 2010 .....   | 97  |
| Figure 4.2. Centerline velocity values for different FLUENT solutions .....                                       | 98  |
| Figure 4.3. Closer view of the orifice region for centerline velocity values for different FLUENT solutions ..... | 99  |
| Figure 4.4. $V_r/U_0$ vs. $x/d$ for different solutions.....                                                      | 100 |
| Figure 4.5. $V_r/U_0$ vs. $x/d$ for different solutions.....                                                      | 101 |
| Figure 4.6. Closer view of the orifice area for $V_r/U_0$ vs. $x/d$ for different solutions .....                 | 102 |
| Figure B.1. Residuals of the inviscid flow solution .....                                                         | 109 |
| Figure B.2. Residuals of the inviscid flow solution .....                                                         | 109 |
| Figure B.3. Residuals of the laminar flow solution.....                                                           | 110 |
| Figure B.4. Residuals of the k- $\epsilon$ solution .....                                                         | 110 |
| Figure B.5. Residuals of the vof solution.....                                                                    | 111 |
| Figure C.6. Pathlines by velocity magnitude .....                                                                 | 112 |
| Figure C.7. Close up of the orifice area for the pathlines by velocity magnitude...                               | 112 |
| Figure C.8. Contours of velocity magnitude .....                                                                  | 113 |
| Figure C.9. Close up of the orifice area for the contours of velocity magnitude ....                              | 113 |
| Figure C.10. Contours of velocity magnitude .....                                                                 | 114 |
| Figure C.11. Close up of the orifice area for the contours of velocity magnitude ..                               | 114 |
| Figure C.12. Velocity magnitude contours of the orifice area only .....                                           | 115 |
| Figure C.13. Pressure contours .....                                                                              | 115 |
| Figure C.14. Pressure contours of the orifice area .....                                                          | 116 |
| Figure C.15. Velocity magnitude readings of the centerline velocity.....                                          | 116 |
| Figure C.16. y-z plane velocity magnitude on z-axis for different x values .....                                  | 117 |
| Figure C.17. y-z plane velocity magnitude on y-axis for different x values.....                                   | 117 |
| Figure C.18. $V_r$ vs. z, for different x values .....                                                            | 118 |
| Figure C.19. Velocity vectors for $x=0, 0.375\text{cm}, 0.75\text{cm}, 1.5\text{cm}, 3\text{cm}$ .....            | 118 |

## LIST OF SYMBOLS

|                     |   |                                        |
|---------------------|---|----------------------------------------|
| $\omega$            | : | Vorticity                              |
| $V$                 | : | Velocity Field                         |
| $\nabla$            | : | Laplace Operator                       |
| $\emptyset$         | : | Velocity Potential                     |
| $\rho$              | : | Fluid Density                          |
| $p$                 | : | Pressure Field                         |
| $H$                 | : | Bernoulli Constant                     |
| $v_x$               | : | x-axis Velocity Component              |
| $v_y$               | : | y-axis Velocity Component              |
| $v_r$               | : | Radial Velocity Component              |
| $v_\theta$          | : | Azimuthal velocity component           |
| $\Psi$              | : | Stream Function                        |
| $\theta$            | : | Azimuthal Angle                        |
| $Q$                 | : | Volumetric Discharge                   |
| $W$                 | : | Work                                   |
| $m$                 | : | Sink Strength                          |
| $U_0$ ( $V_0$ )     | : | Average Velocity                       |
| $u_{max}$ $v_{max}$ | : | Maximum Velocity                       |
| $r_0$               | : | Radial Coordinate of Orifice Center    |
| $\theta_0$          | : | Azimuthal Coordinate of Orifice Center |
| $d$                 | : | Diameter of Orifice                    |



## CHAPTER 1

### INTRODUCTION

#### 1.1. Introduction and Scope of the Study

Detailed study of orifice-induced flow finds place in many engineering applications, e.g. fish entrainment study, flow in sedimentation tanks and flow induced by sluice gates, selective withdrawal from a reservoir, analyzing the flow upstream of intakes (Gerges and McCorquodale, 1997; Göbelez (2005), Shamma et al. (2005) and Bryant et al. (2008). Considering the design of a fish repulsion system, if the acceleration zone behind the intake can be identified (Islam and Zhu, 2010), the proper screen placement can be chosen in order to prevent fish sticking (Shamma et al., 2005). Furthermore, the movement of sediment behind orifices gains importance when they are placed in reservoirs and ponds and sediment accumulation can be predicted with the understanding of flow behind orifices.

In addition, calculation of sediment flushing can be possible by studying the movement of sediment due to orifice induced velocity field within a reservoir. Initiation of motion under the effect of a bottom outlet in a reservoir was investigated previously in the study of Göbelez (2008).

Orifices were widely examined in order to understand and predict the relationship between pressure drop or head loss and flow rate through orifices (Çobanoğlu, 2008 and Yıldırım, 2010). However, the number of studies on the flow characteristics behind the orifice is limited.

There are several experimental and numerical studies carried out recently in Hydromechanics Lab. of Civil Engineering Department, Middle East technical University (Göbelez, 2008, Kemalli, 2009, Çobanoğlu, 2008 and Yıldırım, 2010). In these studies, the main focus was orifice tubes instead of sharp crested orifices due to practical applications. In the present study, a sharp crested orifice is preferred in

order to compare two potential flow solutions proposed recently by Shammaa et al. (2005) and Byrant et al. (2008).

There are several experimental and numerical studies carried out recently in Hydromechanics Lab. of Civil Engineering Department, Middle East technical University (Göbelez, 2008, Kemalli, 2009, Çobanoğlu, 2008 and Yıldırım, 2010). In these studies, the main focus was orifice tubes instead of sharp crested orifices due to practical applications. In the present study, a sharp crested orifice is preferred in order to compare two potential flow solutions proposed recently by Shammaa et al. (2005) and Byrant et al. (2008).

There are several experimental and numerical studies carried out recently in Hydromechanics Lab. of Civil Engineering Department, Middle East technical University (Göbelez, 2008, Kemalli, 2009, Çobanoğlu, 2008 and Yıldırım, 2010). In these studies, the main focus was orifice tubes instead of sharp crested orifices due to practical applications. In the present study, a sharp crested orifice is preferred in order to compare two potential flow solutions proposed recently by Shammaa et al. (2005) and Byrant et al. (2008).

The scope of this study is to compare these potential solutions along with the CFD solution obtained by means of the commercially available software ANSYS FLUENT in order to understand advantages and disadvantages of available solution approaches.

Therefore, in Chapter 1 a brief explanation is made on the problem. Chapter 2 provides a theoretical background and solution approaches to the problem. Chapter 3 presents the results of the solutions, while the comparison of the results can be found at Chapter 4. Conclusion is presented at Chapter 5. Calculation tables can be found in the Appendices section.

## CHAPTER 2

### THEORETICAL BACKGROUND

#### 2.1. General Information

An orifice can be defined as a small opening of any shape, at the bottom or on side of a storage structure (such as tank, reservoir). Orifices can be classified by various properties according to their shape (rectangular, circular, triangular, square), size (large, small), shape of upstream edge (sharp edged, round edged), nature of discharge (partially submerged, submerged, free discharging). Various orifice types can be seen at Figure 2.1, Figure 2.2 and Figure 2.3.

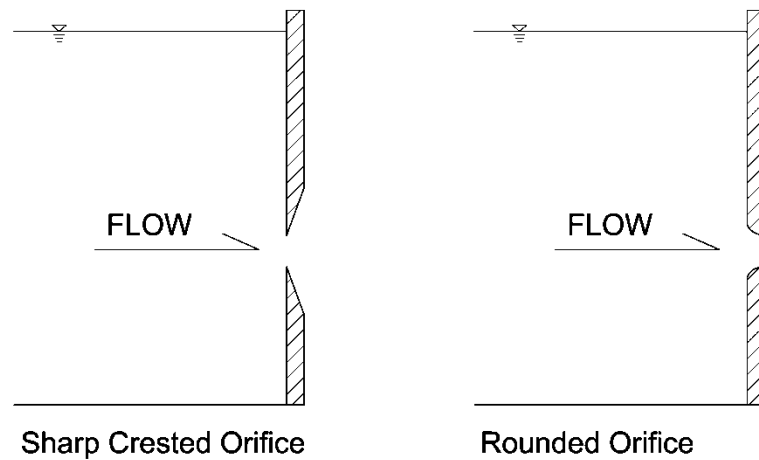


Figure 2.1. Orifice types according to shape of upstream edge

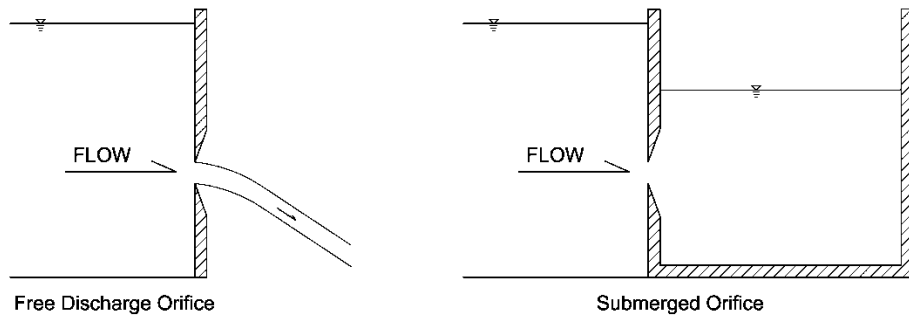


Figure 2.2. Orifice types according to nature of discharge

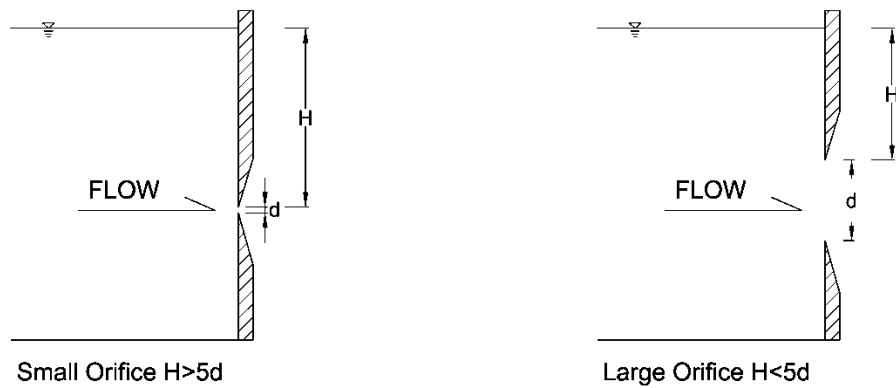


Figure 2.3. Orifice types according to size

Orifices have a large area of application both as measurement instruments like flow meters or as flow control devices like inlets or bottom outlets. It is also possible to view many types of valves or flow devices as orifices, however this area of application is outside of the scope of this thesis. The concept of flow upstream or through the orifices still needs more investigation as the concept is getting more popular recently.

As the orifice flow is quite important in terms of hydraulic engineering, it is also of equal importance from a medical point of view. There are a number of studies investigating the impact of size of the orifice on the flow characteristics, flow acceleration close to the orifice (Rodriguez et. al 1992, Anayiotos et. al 1995) as the concept of orifice flow is very important in determination of blood flow through the heart.

Numerous studies have been carried out in order to investigate flow upstream of orifices. Two dimensional (2D) potential flow theory approach used to describe the flow pattern was found inaccurate close to the orifice by Shammaa et. al, 2005. In order to solve the problem Shammaa et. al introduced a new 3D potential flow solution. Later at 2008, Bryant et. al suggested another 3D potential flow solution which they claim their solution better represents the real flow conditions in the very vicinity of the orifice.

## **2.2. Potential Flow**

Being the fact that all real fluids are viscous, concept of ideal fluids is purely theoretical since ideal fluids assumed to have no viscosity. Viscosity is an outcome of the momenta distribution in between the neighbouring fluid molecules as a result of intermolecular collisions, thus a viscous force is applied onto a fluid element where a velocity gradient is present (Fluid Mechanics, Sixth Edition, Douglas J.F. et. al., 2011).

Prandtl (1904) assumed that for very high Reynold's number flows, viscosity is limited to very thin boundary layer near the solid wall and flow outside of this region acts as if it possesses no viscosity. Prandtl's assumption proved later on and became a powerful tool to solve many engineering problems. As vortices are caused by viscous shear forces, assumption of no viscosity and absence of other rotational forces concludes the flow will be irrotational outside of the very thin boundary layer near the solid wall.

To summarize, ideal fluids are incompressible and they have no surface tension and vaporization pressure limit (Fluid Mechanics, Sixth Edition, Douglas J.F. et. al., 2011) and flows corresponding to ideal fluids are irrotational.

When viscous and other rotational forces are not present in the flow, the flow will remain irrotational as it originally is outside the boundary layer. Letting  $V$  as the velocity field, vorticity must be equal to zero, that is;

$$\omega = \nabla \times V = 0 \quad (2.1)$$

Eq. (2.1) is satisfied if a *velocity potential*  $\phi$  is introduced, such as;

$$V = \nabla \phi \quad (2.2)$$

Thus, irrotational flows are also called potential flows and vice versa. Velocity vector generally having three components now is defined by a single scalar quantity  $\phi$ .

Since ideal fluids are incompressible, the velocity field must satisfy the continuity equation for a fluid;

$$\nabla \cdot V = 0 \quad (2.3)$$

Substituting Eq. (2.2) into Eq. (2.3), Eq. (2.4) is obtained;

$$\nabla^2 \phi = 0 \quad (2.4)$$

where  $\nabla^2$  is the Laplace Operator and Eq. (2.4) is called the Laplace Equation.

Solution procedure for an incompressible, high Reynold's number fluid using Prandtl's assumption is to be carried out as follows : By solving the Laplace Eq. (2.4), a velocity potential should be found for the external region which satisfies both upstream boundary conditions and that the requirement that the fluid velocity must be tangent to the body surface. Fluid velocity is tangent to the body surface if the boundary layer around the body is thin. Following the determination of the velocity field for the external region using abovementioned kinematic considerations, the pressure field  $p$  can be computed using Euler equation;

$$\rho \frac{DV}{Dt} = -\nabla p \quad (2.5)$$

where  $\rho$  is density and  $t$  is time.

Eq. (2.5) gives the equation of motion for inviscid fluids with body forces ignored. In Eq. (2.5), the following operator;

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + \mathbf{V} \cdot \nabla \quad (2.6)$$

is the total derivative which gives rate of change by following a fluid particle.

For incompressible fluid and irrotational velocity field, Bernoulli Equation has the form;

$$\rho \frac{\partial \phi}{\partial t} + p + \frac{1}{2} \rho V^2 = H \quad (2.7)$$

where  $H$  is the Bernoulli constant (which can also be referred as stagnation pressure, a point where the fluid velocity is zero and all kinetic energy has been converted into pressure energy).

Eq. (2.2) and Eq. (2.4) can be expressed in Cartesian coordinates for 2D flows as;

$$v_x = \frac{\partial \phi}{\partial x}, v_y = \frac{\partial \phi}{\partial y} \quad (2.8)$$

and

$$\frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} = 0 \quad (2.9)$$

where  $v_x$  is the velocity component in x-axis and  $v_y$  is the velocity component in y-axis. Considering polar coordinates;

$$v_r = \frac{\partial \phi}{\partial r}, v_\theta = \frac{1}{r} \frac{\partial \phi}{\partial \theta} \quad (2.10)$$

and

$$\frac{\partial^2 \phi}{\partial r^2} + \frac{1}{r} \frac{\partial \phi}{\partial r} + \frac{1}{r^2} \frac{\partial^2 \phi}{\partial \theta^2} = 0, \quad (2.11)$$

where  $v_r$  is the velocity component in radial direction and  $v_\theta$  is the velocity component in azimuthal direction.

Streamlines and stream function can also be used for analyzing planar or axisymmetric incompressible flows. A streamline is defined as a purely theoretical line in space being tangential to velocity vectors at any given time. Considering the velocity and displacement vectors of a fluid at a point as shown in Figure 2.4.

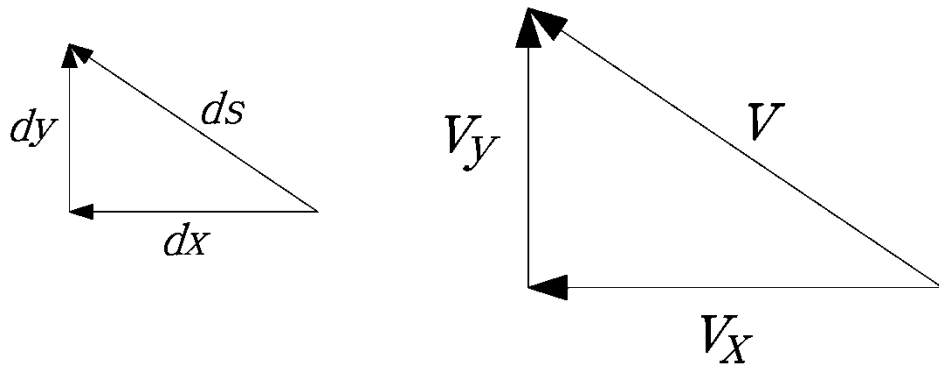


Figure 2.4. Orifice types according to size

here;

$$dy = v_y \text{ and } dx = v_x$$

Thus two triangles shown at Figure 2.4. are similar to each other. Therefore;

$$\frac{dx}{v_x} = \frac{dy}{v_y} \quad (2.12)$$

In order to determine streamlines by a numerical system, the following procedure was applied. In the following Figure 2.5, let  $aa'$  and  $bb'$  be two streamlines in a flow bounded by solid boundaries  $AA'$  and  $BB'$ .

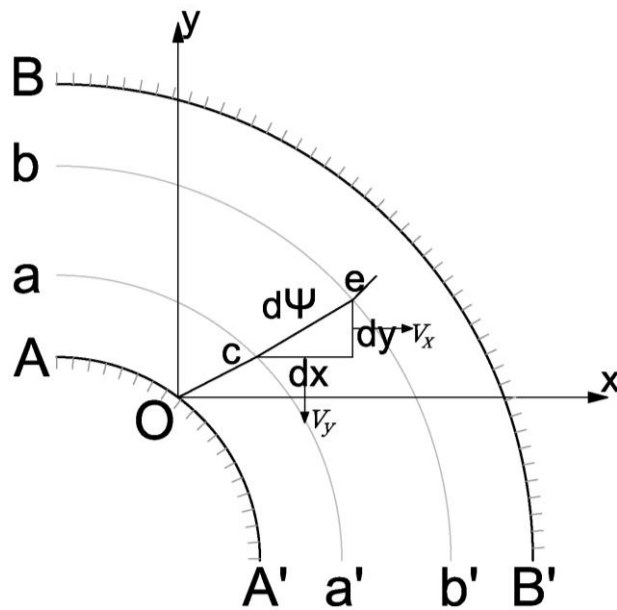


Figure 2.5. Orifice types according to size

If the streamline corresponding to the flow rate per unit depth between AA' and the streamline aa' would be denoted as  $\Psi_a$ ;

$$\Psi_a = Q_{Oc} \quad \text{and} \quad \Psi_b = Q_{Oe}$$

it follows;

$$d\Psi = \Psi_b - \Psi_a = Q_{ce}$$

therefore;

$$d\Psi = v_x dy - v_y dx \quad (2.13)$$

$\Psi$  which is the *stream function*, is given by Eq. (2.14) as;

$$\Psi = \int v_x dy - \int v_y dx \quad (2.14)$$

Therefore, the stream function depends on position coordinates

$$d\Psi = \frac{\partial \Psi}{\partial x} dx - \frac{\partial \Psi}{\partial y} dy \quad (2.15)$$

Relationship between the velocity components and the stream function is given by analyzing the Eq. (2.13) and Eq. (2.15);

$$v_x = \frac{\partial \Psi}{\partial y} ; v_y = -\frac{\partial \Psi}{\partial x} \quad (2.16)$$

For curled flows, it is more appropriate to use polar coordinates;  $\Psi = f(r, \theta)$  differentiating;

$$d\Psi = \frac{\partial \Psi}{\partial r} dr + \frac{\partial \Psi}{\partial \theta} d\theta \quad (2.17)$$

When using polar coordinates, positive direction for tangential velocity is the positive direction for azimuthal angle  $\theta$ , which is the counterclockwise direction and the positive direction for the radial velocity is outwards.

Let  $\Psi_x$  and  $\Psi_y$  be two streamlines as shown in Figure 2.6. Applying the continuity equation under the assumption of  $XY = dr$  when  $\approx 0$  ;

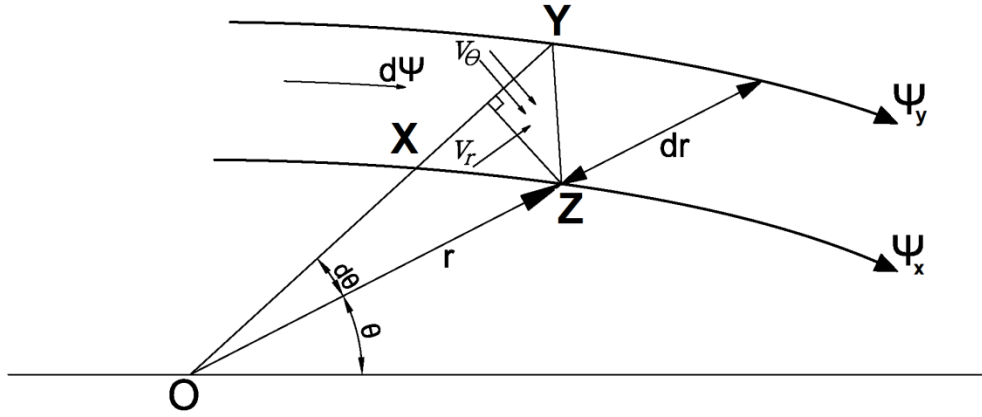


Figure 2.6. Orifice types according to size

$$v_r \, r d\theta - v_\theta dr = d\Psi \quad (2.18)$$

Investigating Eq. (2.17) and Eq. (2.18), Eq. (2.19) is obtained;

$$v_\theta = -\frac{\partial \Psi}{\partial r} \text{ and } v_r = \frac{1}{r} \frac{\partial \Psi}{\partial \theta} \quad (2.19)$$

Velocity *potential* can be defined as;

$$\phi = \int_X^Y v_s ds \quad (2.20)$$

where  $v_s$  is the velocity tangential to the elementary path  $s$ , and  $X$  and  $Y$  are two different points in a potential field where the work is to be carried out.

In order to clarify the meaning of potential which is a term often used in mechanics, consider  $P$  and  $P'$  in a gravitational field,  $P'$  having a greater potential than the point

P. Then, the work required to carry a particle from P to P' is defined as the difference in between their potential;  $\partial W$ .

If the force is F and the distance between P and P' is ds, then;

$$W = \int_P^{P'} F ds \quad (2.21)$$

Consider two points in the same potential field, namely A and B;

$$\int_A^B v ds$$

is independent of the path taken. Then;

$$\int_A^B v_m ds = \int_A^B v_n ds \quad (2.22)$$

In such a field, fluid flow is referred as potential flow.

Considering the round way around AnBm in Figure 2.3;

$$\begin{aligned} \Gamma_{AnBm} &= \int_A^B v_n ds + \int_B^A v_m ds \\ &= \int_A^B v_n ds - \int_B^A v_m ds \end{aligned} \quad (2.23)$$

For the field to be potential, two integrals in Eq. (2.23) must be equal, so  $\Gamma_{AnBm} = 0$ .

Since circulation is equal to zero than vorticity must be equal to zero, therefore;

$$\frac{\partial v_y}{\partial x} - \frac{\partial v_x}{\partial y} = 0 \quad (2.24)$$

which is exactly the condition for irrotational flow. Therefore, from Eq. (2.20) it could be concluded that, if  $v_x$  and  $v_y$  are the orthogonal components of  $v_s$ ;

$$v_x = \frac{\partial \phi}{\partial x} \text{ and } v_y = \frac{\partial \phi}{\partial y} \quad (2.25)$$

so that;

$$\phi = v_x dx + v_y dy \quad (2.26)$$

In polar coordinates, letting  $v_r$  and  $v_\theta$  as the radial and tangential components of  $v_s$ ;

$$\phi = v_r dr + v_\theta r d\theta \quad (2.27)$$

so that;

$$v_r = \frac{\partial \phi}{\partial r} \text{ and } v_\theta = \frac{1}{r} \frac{\partial \phi}{\partial \theta} \quad (2.28)$$

Comparing Eq. (2.16) and Eq. (2.28) and equating for  $v_x$  and  $v_y$  Eq. (2.29) is obtained;

$$\frac{\partial \Psi}{\partial y} = \frac{\partial \phi}{\partial x} \text{ and } \frac{\partial \phi}{\partial y} = -\frac{\partial \Psi}{\partial x} \quad (2.29)$$

which are called Cauchy-Riemann Equations and these functions enable to determine velocity potential or stream function if one of these equations are already known.

### 2.2.1. Source or Sink Flows

A source can be defined as an origin through which the flow is obtained. A sink can be defined as an outlet from which the flow is discharged. Velocity potential of a sink (or source) having the sink strength  $m$  centered at the origin of the spherical coordinate system  $(r, \theta)$  can be defined as;

$$\phi = m * \ln r \quad (2.30)$$

Taking the derivative of Eq. 2.30 with respect to  $r$ , one could obtain the radial velocity  $v_r$  ;

$$v_r = \frac{m}{r} \quad (2.31)$$

where the sink strength  $m$  can be defined as;

$$m = \mp \frac{Q}{2\pi} \quad (2.32)$$

As it can be clearly seen from the Eq. 2.31. as the velocity component  $v_r$  decreases with increasing  $r$ , it has a singularity point at  $r = 0$ .

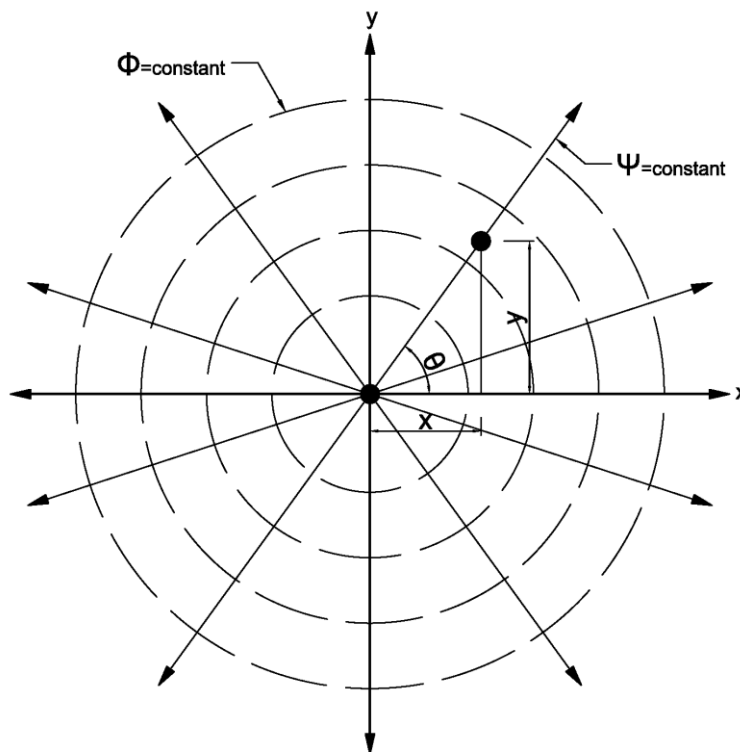


Figure 2.7. Streamlines and potentials representing source flows

For 3D source or sink flows, potential function can be defined as;

$$\phi = \frac{m}{2r} * \ln r \quad (2.33)$$

$$v_r = \frac{m}{2r^2} \quad (2.34)$$

### 2.2.2. 3D Potential Flow – Shammaa Solution

There is an attempt to investigate orifice induced velocity field by using a three dimensional potential flow solution by Shammaa et al. (2005). The authors suggested solutions for orifice and sluice gates however for the aim of this study, only orifice flow will be discussed here.

Shammaa et al. (2005) used a polar coordinate system ( $r, \theta, w$ ) in order to study the flow upstream of an orifice. For this study, researchers focused on the velocity on the x-z plane, therefore taking the azimuthal angle,  $w$ , as zero.

The author furthermore states that the sink solution presented at Chapter 2.2.1. can be distributed across a circular orifice centered at the origin. Selecting an infinitesimal area on the orifice area as  $dA$  and  $r_0$  and  $\theta_0$  as the coordinates of the orifice center,  $dA = r_0 dr_0 d\theta_0$ . Thus,  $dm = \frac{U_0 dA}{2\pi} = -\frac{2Q dA}{\pi^2 d^2}$  where  $U_0 = \frac{Q}{(\frac{\pi d^2}{4})}$  represents the uniform velocity through the orifice, and  $d$  is the diameter of the orifice. The velocity potential at a point  $N(r, \theta, 0)$  corresponding to such a sink is;

$$d\phi = -\frac{2Q}{\pi^2 d^2} * \left( \frac{r_0 dr_0 d\theta_0}{L} \right) \quad (2.35)$$

where  $L = (r_0^2 + r^2 - 2rr_0 * \cos \theta * \cos \theta_0)^{1/2}$  is the distance between the orifice center at  $(r_0, \theta_0)$  to the point  $N(r, \theta, 0)$ . He concludes that integrating over the orifice surface gives the velocity potential at point  $N$  as;

$$\phi = -\frac{2Q}{\pi^2 d^2} * \int_0^{2\pi} \int_0^{d/2} \frac{r_0 dr_0 d\theta}{(r_0^2 + r^2 - 2rr_0 \cos \theta \cos \theta_0)^{1/2}} \quad (2.36)$$

Eq. (2.36) is valid for the situations where the orifice center is far from both the reservoir bottom and the free surface, i.e.  $h \gg d$ ,  $h$  being the depth of the orifice center.

Furthermore, if Eq. (2.36) is differentiated with respect to  $r$  and letting  $\theta = \frac{\pi}{2}$ , then the flow velocity along x-axis will be obtained;

$$\frac{u_{\max}}{U_0} = 1 - \left(1 + \frac{d^2}{4x^2}\right)^{-1/2} \quad (2.37)$$

where  $x$  is the distance along the orifice centerline to the orifice on the x-axis.

A schematic representation of the problem is given in Figure 2.8 and Figure 2.9.

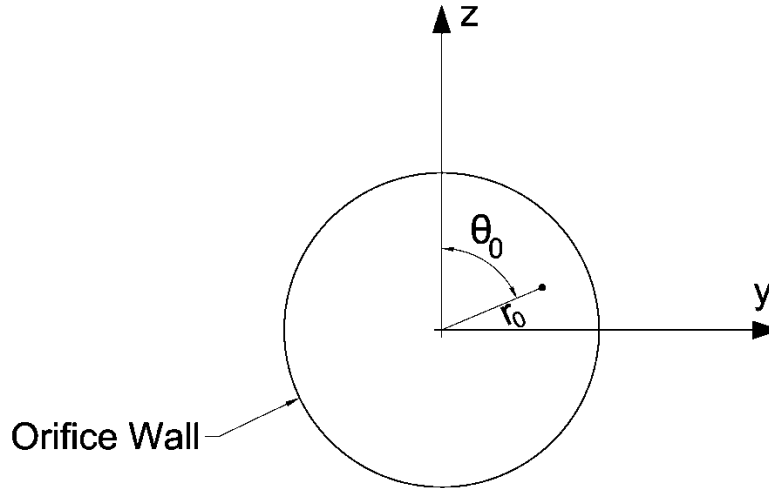


Figure 2.8. Flow axes used at Shammaa's solution

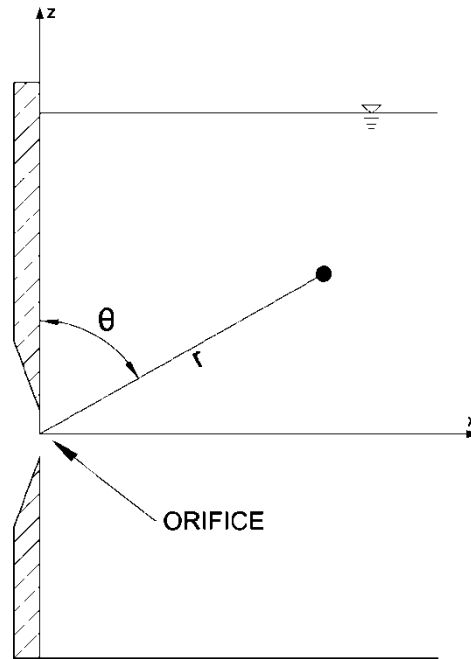


Figure 2.9. Flow axes used at Shammaa's solution

### 2.2.3. 3D Potential Flow – Bryant Solution

Bryant et al. (2008) suggested another three dimensional potential flow solution for an orifice induced velocity field problem, by changing the coordinate system used in Shammaa's study to the Cartesian coordinates. Bryant et al. investigated flow patterns of an ideal orifice using a physical model.

Authors claimed that Shammaa's solution cannot present accurate solutions for isovels or velocity profiles very close to the orifice, and for large orifices the pressure variation across the orifice cannot be ignored, as it was ignored at Shammaa's solution. They proposed a modification in order to take pressure gradient into account as it cannot be neglected for large orifices.

In Bryant's study, velocity potential is defined as;

$$\phi(x, y, z) = -\frac{2Q}{\pi^2 d^2} * \int_0^{2\pi} \int_0^{d/2} \frac{r_0 dr_0 d\theta}{(x^2 + (y - r_0 \sin \theta_0)^2 + (z - r_0 \sin \theta_0)^2)^{1/2}} \quad (2.38)$$

Velocity components in x, y and z directions are defined as;

$$\frac{V_x(x, y, z)}{U_0} = -\frac{1}{2\pi} * \int_0^{2\pi} \int_0^{d/2} \frac{x r_0 dr_0 d\theta}{(x^2 + (y - r_0 \sin \theta_0)^2 + (z - r_0 \sin \theta_0)^2)^{3/2}} \quad (2.39)$$

$$\frac{V_y(x, y, z)}{U_0} = -\frac{1}{2\pi} * \int_0^{2\pi} \int_0^{d/2} \frac{(y - r_0 \sin \theta_0) r_0 dr_0 d\theta}{(x^2 + (y - r_0 \sin \theta_0)^2 + (z - r_0 \sin \theta_0)^2)^{3/2}} \quad (2.40)$$

$$\frac{V_z(x, y, z)}{U_0} = -\frac{1}{2\pi} * \int_0^{2\pi} \int_0^{d/2} \frac{(z - r_0 \sin \theta_0) r_0 dr_0 d\theta}{(x^2 + (y - r_0 \sin \theta_0)^2 + (z - r_0 \sin \theta_0)^2)^{3/2}} \quad (2.41)$$

Therefore, it is possible to compute the radial velocity at any point upstream of the orifice. When computing the radial velocity in the x-z or x-y plane passing through the center of the orifice,  $V_y$  and  $V_z$  components can be ignored.

A schematic representation of the problem was given in Figure 2.10 and Figure 2.11.

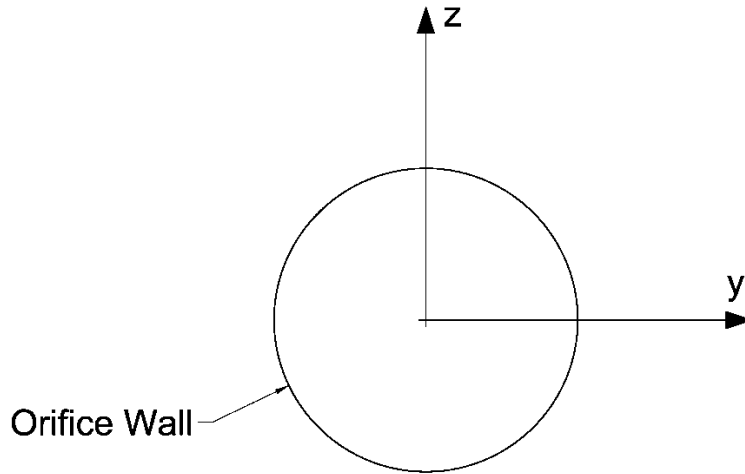


Figure 2.10. Flow axes used at Bryant's solution

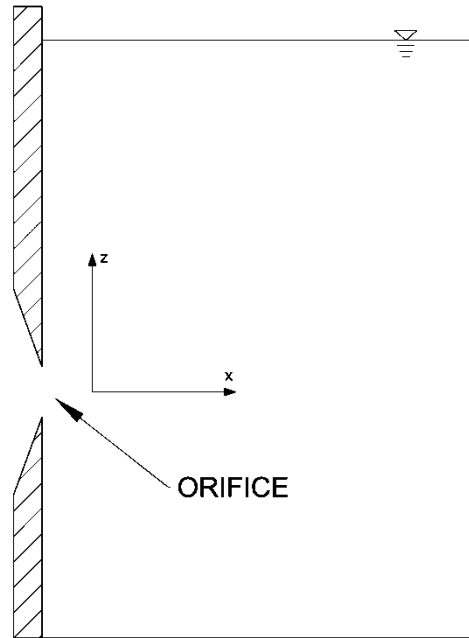


Figure 2.11. Flow axes used at Bryant's solution

## 2.3. Fluent CFD Solution

### 2.3.1 About ANSYS Fluent Software

The numerical computations for the comparison of analytical and numerical values are carried out using a computational fluid dynamics software; ANSYS Fluent.

ANSYS Fluent is a powerful computational fluid dynamics software using control volume approach, which is capable of modeling fluid flow, heat transfer and chemical reactions for complex geometries. ANSYS Fluent is used widely for automobile, aerospace, food and mechanical engineering areas (ANSYS Fluent, Yıldırım, 2010).

ANSYS Fluent has a flexible meshing structure, which can enable solution of complex geometries by unstructured meshes with ease. 2D triangular/quadrilateral, 3D tetrahedral/hexahedral/pyramid/wedge, and mixed (hybrid) meshes are in the type of meshes that the software could work with (ANSYS Fluent Theory Guide 6.3).

### 2.3.2 ANSYS Fluent Inviscid Flow Solution

ANSYS Fluent uses Euler Equations to carry out inviscid flow solutions. Although conservation of mass equation is the same as a laminar flow, absence of viscous forces and molecular diffusion reduces the conservation of momentum and energy equations.

Conservation of mass equation or alternatively the continuity equation;

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = S_m \quad (2.42)$$

This general form of continuity equation is valid for both compressible and incompressible flows. In Eq. (2.42),  $S_m$  reflects the mass added to the flow.

For 2D axisymmetric flows, continuity equation;

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x} \rho v_x + \frac{\partial}{\partial r} \rho v_r + \frac{\rho v_r}{r} = S_m \quad (2.43)$$

In Eq. (2.43),  $x$ ,  $r$ ,  $v_x$ ,  $v_r$  are the axial and radial coordinates and velocities respectively.

Conservation of momentum equation is given by;

$$\frac{\partial}{\partial t} \rho \mathbf{v} + \nabla \cdot \rho \mathbf{v} \mathbf{v} = -\nabla p + \rho \mathbf{g} + \mathbf{F} \quad (2.44)$$

In Eq. (2.44),  $p$ ,  $\rho \mathbf{g}$  and  $\mathbf{F}$  are the static pressure, gravitational body force and external body forces respectively.

For 2D axisymmetric flows, conservation of momentum equation;

$$\frac{\partial}{\partial t} \rho v_x + \frac{1}{r} \frac{\partial}{\partial x} r \rho v_x v_x + \frac{1}{r} \frac{\partial}{\partial r} r \rho v_r v_x = -\frac{\partial p}{\partial x} + F_x \quad (2.45)$$

and

$$\frac{\partial}{\partial t} \rho v_r + \frac{1}{r} \frac{\partial}{\partial x} r \rho v_x v_r + \frac{1}{r} \frac{\partial}{\partial r} r \rho v_r v_r = -\frac{\partial p}{\partial r} + F_r \quad (2.46)$$

where

$$\nabla \cdot \mathbf{v} = \frac{\partial v_x}{\partial x} + \frac{\partial v_r}{\partial r} + \frac{v_r}{r} \quad (2.47)$$

### 2.3.3. Modeling

Geometry and mesh needed by ANSYS Fluent for the solution is done by using ANSYS Gambit software.

First of all, in order to have ANSYS Fluent to solve the problem, the basic geometric model is built in ANSYS Gambit and meshed afterwards. Following this procedure, boundary types are specified in ANSYS Gambit. Then, the final meshed geometry is transferred to ANSYS Fluent for the hydraulic analysis.

For the purpose of this study, calculations are carried out first by selecting inviscid flow, then continuing onwards to laminar,  $k - \varepsilon$  method and volume of fluid method in order to be able to compare the corresponding results.

### 2.3.4. Meshing

In the study, suggested meshing method by the FLUENT software, namely the Tet/Hybrid Tgrid meshing model is used with three different mesh sizes. Each of these meshes is studied at FLUENT inviscid flow solution in order to investigate the mesh dependence.

For coarse, fine and finer mesh models, the typical mesh sizes on orifice surface were selected as 0.05m, 0.01m and 0.005m respectively.

Errors calculated by taking into consideration the net mass flux difference and the residuals. Boundary conditions can be seen at Figure 2.15.

Representation of all the meshing models is presented here, along with the error margins of the corresponding runs.

Figures 2.12 to 2.17 present different mesh sizes.

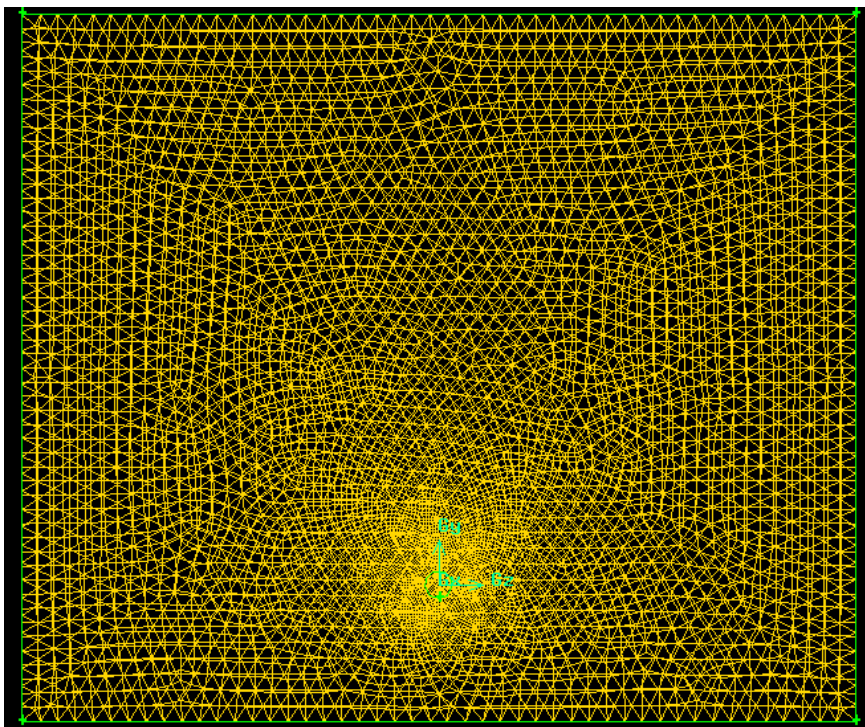


Figure 2.12. Front view of the coarse model

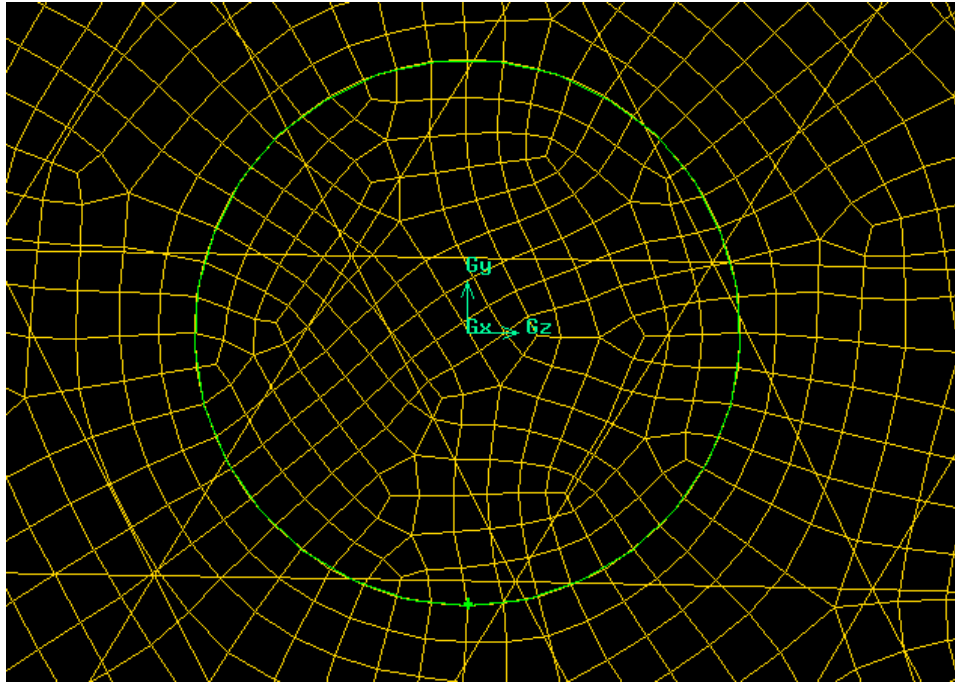


Figure 2.13. Closer view of the orifice area of the coarse model

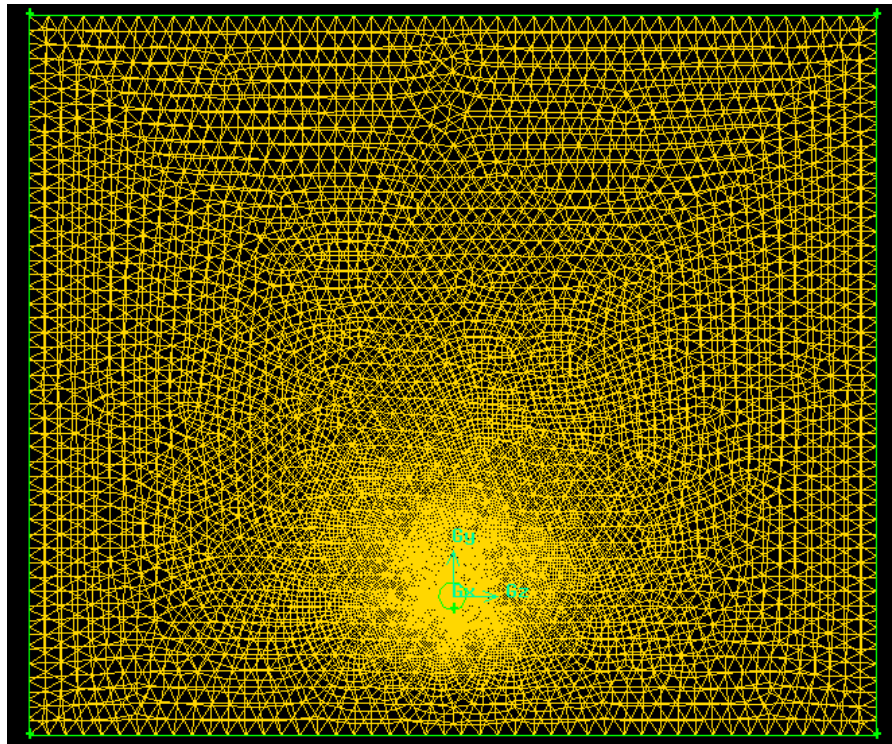


Figure 2.14. Front view of the fine model

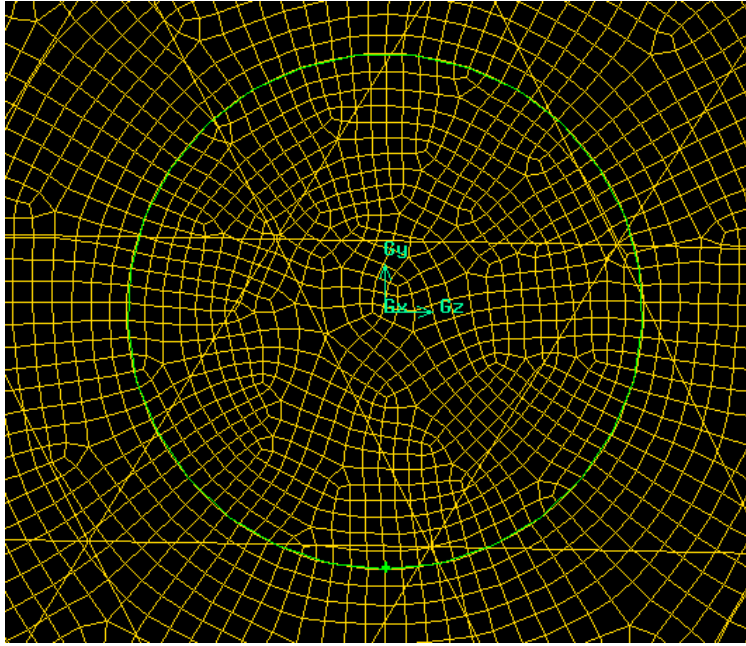


Figure 2.15. Closer view of the orifice area of the fine model

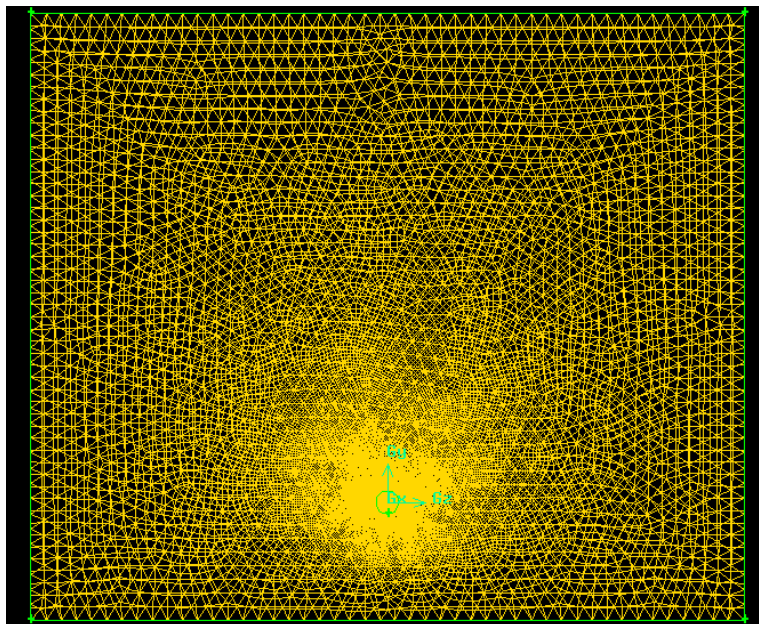


Figure 2.16. Front view of the finer model

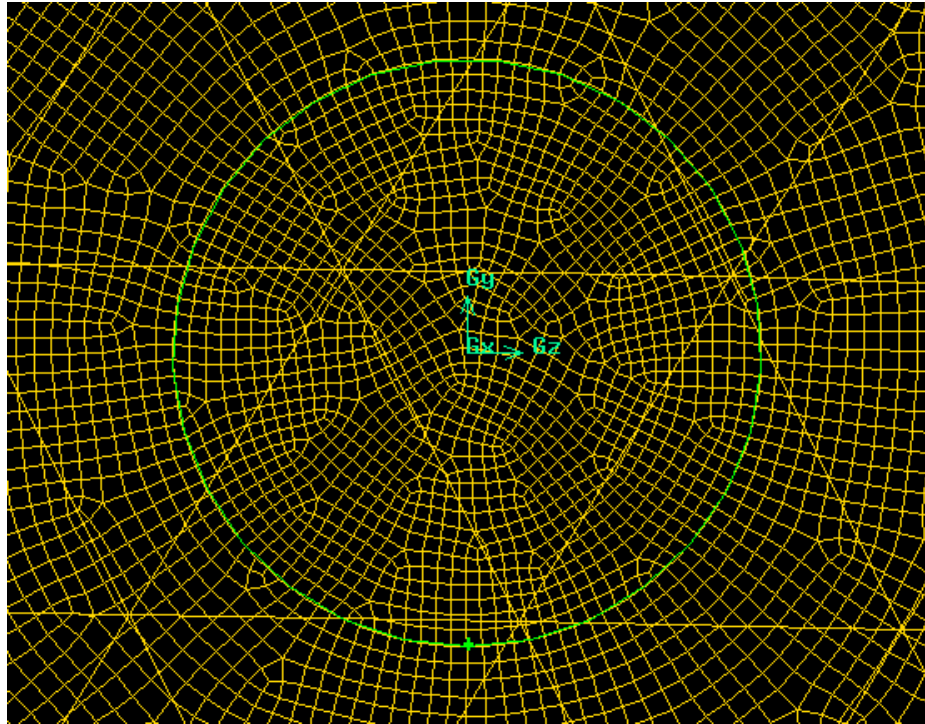


Figure 2.17. Closer view of the orifice area of the fine model

Table 2.1. Mesh size information

|              | Cells  | Faces   | Nodes  |
|--------------|--------|---------|--------|
| Coarse Model | 320067 | 656917  | 63125  |
| Fine Model   | 489304 | 1005611 | 98382  |
| Finer Model  | 588874 | 1212381 | 120239 |

At Table 2.1. the mesh information regarding cell, face and node numbers is shown.

Table 2.2. Mass flux information for different mesh sizes

|          |              | mass_flow_inlet<br>(kg/s) | orifice<br>(kg/s) | pressure_outlet<br>(kg/s) | net (kg/s) | error margin<br>(%) |
|----------|--------------|---------------------------|-------------------|---------------------------|------------|---------------------|
| Inviscid | Coarse Model | 35000.04                  | -34572.14         | -402.7121                 | 25.17462   | 0.071927            |
|          | Fine Model   | 35000.04                  | -34411.99         | -589.34058                | -1.2976    | 0.003707            |
|          | Finer Model  | 35000.04                  | -34290.08         | -517.00287                | 192.9463   | 0.551275            |
| Laminar  | Coarse Model | 35000.04                  | -34758.16         | -242.99999                | -1.124939  | -0.00321411         |
|          | Fine Model   | 35000.04                  | -34552.29         | -447.763                  | -0.0169067 | -0.0000483          |
|          | Finer Model  | 35000.04                  | -34443.01         | -555.46735                | 1.5599976  | 0.004457            |

After carrying out inviscid and laminar solutions for all the mesh sizes, a comparison of net flux difference between inlet and outlets shows that the fine mesh model results in lowest error margin, as can be seen at Table 2.2. Therefore after this point, fine mesh model was used for all other calculations.

In order to carry out the solutions, FLUENT needs boundary conditions such as inlets or outlets. Here, inlet of the box model is selected as mass\_flow\_inlet and the orifice is selected as pressure\_outlet. Top of the box model is selected as pressure\_outlet as it should be open to atmosphere and all the other parts of the box are selected as walls.

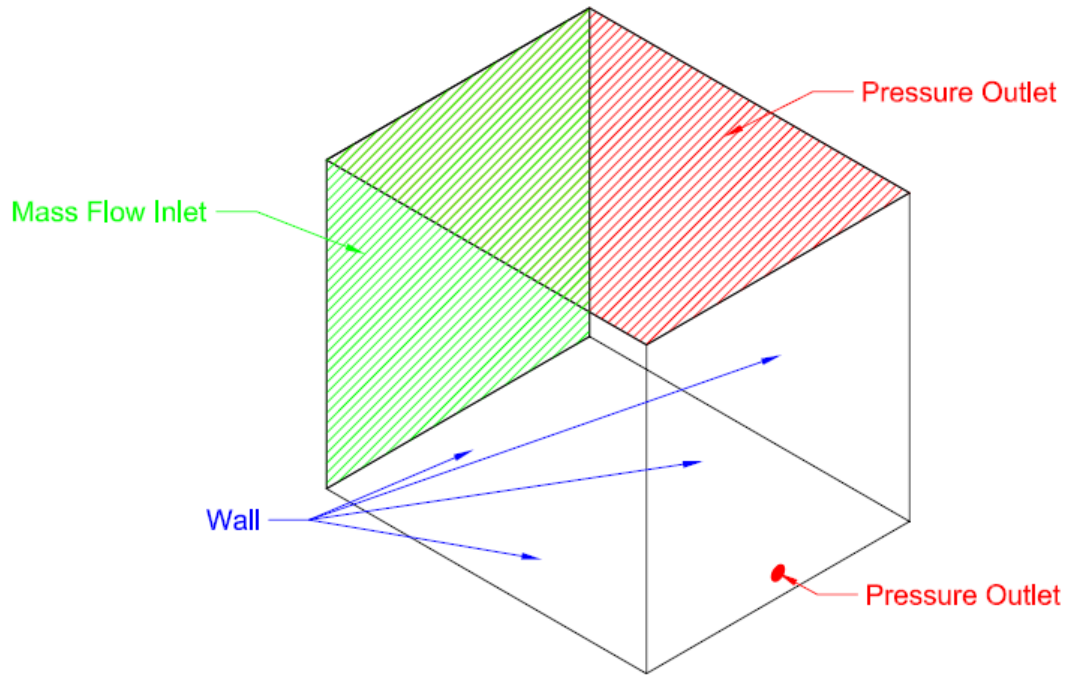


Figure 2.18. Front view of the model setup

### 2.3.5. Volume of Fluid (VOF) Method

The volume of fluid method is a free boundary (surface) method using Eulerian approach which was first mentioned by Hirt and Nichols, 1981.

Two or more immiscible fluids can be modeled by the VOF model by solving a single set of momentum equations and tracking each of the volume fractions. Applications of the method include prediction of jet breakup, bubble motion in a liquid, the liquid motion after dam break, etc.

The method depends on the fluids in each cell are not interpreting. Thus, each fluid occupies a certain volume fraction in each cell and sum of these fractions will be unity. The fields of all variables and properties are shared by the fluids, therefore the variables and properties are representatives of only one fluid or a mixture of fluids

which depends on the volume fraction values. Assuming  $\alpha_q$  is the volume fraction of the  $q^{th}$  fluid, three cases are possible;

- $\alpha_q = 0$ , means that the cell does not contain the  $q^{th}$  fluid
- $\alpha_q = 1$ , means that the cell is fully occupied by the  $q^{th}$  fluid
- $0 < \alpha_q < 1$ , means that the cell contains the  $q^{th}$  fluid as well as other liquid(s).

The continuity equation is solved for one or more volume fractions in order to track the interfaces between the fluid phases. This equation can be defined for the  $q^{th}$  fluid as follows;

$$\frac{1}{\rho_q} \frac{\partial \alpha_q \rho_q}{\partial t} + \nabla \cdot (\alpha_q \rho_q \mathbf{v}_q) = S_{\alpha_q} + \sum_{p=1}^n (m_{pq} - m_{qp}) \quad (2.48)$$

where  $m_{pq}$  is the mass transfer rate from phase p to phase q and  $m_{qp}$  is the mass transfer rate from phase q to phase p.

For the primary phase, the volume fraction equation will not be solved and the following equation will be used;

$$\sum_{q=1}^n \alpha_q = 1 \quad (2.49)$$

The volume fraction will be solved through explicit time discretization.

The momentum equation which is solved for the entire domain is presented at Eq. (2.50). The velocity field obtained from this momentum equation is shared between the phases according to their volume fractions.

$$\frac{\partial \rho \mathbf{v}}{\partial t} + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) = -\nabla p + \nabla \cdot (\mu \nabla \mathbf{v} + \nabla \mathbf{v}^T) + \rho \mathbf{g} + \mathbf{F} \quad (2.50)$$

where the volumetric forces between interphases resulting from surface tension are represented by  $\mathbf{F}$ .

Accuracy of the VOF method is determined by the shape and position of the interface. Piecewise-linear approach in geometric construction scheme is used in

order to represent the interface between fluids. Fluent uses a generalized form of the geometric reconstruction scheme for unstructured meshes from the work of Youngs(1982). This approach assumes that a linear slope exists between the interface of the two fluids and uses this linear shape for calculation of the flow of fluid through the cell faces. Calculation of the position of the linear interface relative to the center of each partially-filled cell, depending on the information about the volume fraction and its derivatives in the cell, is the first step of this approach. The second step is the calculation of the amount of fluid transfer to the next cell by making use of computed linear interface representation and information about the normal and tangential velocity distribution on the face. Using the balance of fluxes calculated during the previous step, the volume fraction in each cell is calculated in the third step (ANSYS FLUENT Theory Guide).

#### **2.3.6. $k - \epsilon$ Method**

In the scope of this study, standard  $k - \epsilon$  method is applied in order to be able to investigate the turbulence effects on the problem.

The  $k - \epsilon$  method is considered as the most widely used two-equation transport model, which means that the model has two more transport equations in order to represent the turbulent properties of the flow. Initial development of the model was carried out by Jones and Launder(1972). The “standard”  $k - \epsilon$  method has the form developed by Jones and Launder, and it uses the empirical constants suggested by Launder and Sharma(1974), therefore it is a semi-empirical model. Because the standard  $k - \epsilon$  method is the first two-equation model used in computational fluid dynamics, economical to implement and it gives reasonable results for a wide range of problems, it is the most widely used and the most popular one. (Statistical Theory and Modeling for Turbulent Flows, Second Edition, P.A. Durbin, B.A. Petterson Reif, 2010)

In this method,  $k$  represents the turbulent kinetic energy, whereas  $\varepsilon$  represents the turbulent dissipation. Furthermore,  $\varepsilon$  determines the scale of the turbulence, whereas  $k$  determines energy in the turbulence.

The turbulence kinetic energy,  $k$ , can be obtained as follows;

$$\frac{\partial \rho k}{\partial t} + \frac{\partial \rho k u_i}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} + G_k + G_b - \rho \varepsilon - Y_M + S_k \quad (2.51)$$

Here,  $\frac{\partial \rho k}{\partial t}$  term represents the rate of change of  $k$ ,  $\frac{\partial \rho k u_i}{\partial x_i}$  term represents the transport of  $k$  by convection,  $\frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j}$  term represents the transport of  $k$  by diffusion.  $G_k$  represents the rate of production of turbulence kinetic energy due to the mean velocity gradients and can be obtained as follows;

$$G_k = -\rho u_i' u_i' \frac{\partial u_j}{\partial x_i} \quad (2.52)$$

$G_b$  in Eq. 2.51. represents the generation of  $k$  due to buoyancy where a non-zero gravity field and a temperature gradient are present simultaneously.  $G_b$  can be calculated from Eq. 2.53.

$$G_b = \beta g_i \frac{\mu_t}{Pr_t} \frac{\partial T}{\partial x_i} \quad (2.53)$$

where  $Pr_t$  is the turbulent Prandtl number for energy and  $g_i$  is the component of gravitational acceleration in the  $i^{th}$  direction. For the standard  $k - \varepsilon$  method  $Pr_t$  can be taken as 0.85.  $\beta$ , the coefficient of thermal expansion, can be calculated from the following equation;

$$\beta = -\frac{1}{\rho} \frac{\partial \rho}{\partial T} \quad (2.54)$$

In Eq. 2.51.  $Y_M$  represents the contribution of the fluctuating dilatation in compressible flows.  $Y_M$  can be determined from;

$$Y_M = 2\rho\epsilon\mu_t^2 \quad (2.55)$$

The rate of dissipation of the turbulence,  $\epsilon$ , can be obtained as follows;

$$\frac{\partial \rho \epsilon}{\partial t} + \frac{\partial \rho \epsilon u_i}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_j} \right) + c_{1\epsilon} \frac{\epsilon}{k} G_k + C_{3\epsilon} G_b - C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon \quad (2.56)$$

Here,  $\frac{\partial \rho \epsilon}{\partial t}$  term represents the rate of change of  $\epsilon$ ,  $\frac{\partial \rho \epsilon u_i}{\partial x_i}$  term represents the transport of  $\epsilon$  by convection,  $\frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_j} \right)$  term represents transport of  $\epsilon$  by diffusion,  $c_{1\epsilon} \frac{\epsilon}{k} G_k + C_{3\epsilon} G_b$  term represents the rate of production of  $\epsilon$ ,  $C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon$  term represents the rate of destruction of  $\epsilon$ .  $C_{1\epsilon}$ ,  $C_{2\epsilon}$ ,  $C_{3\epsilon}$  are empirical constants.

In Eq. 2.51 and 2.56,  $\sigma_k$  and  $\sigma_\epsilon$  are the turbulent Prandtl numbers and  $S_k$  and  $S_\epsilon$  are the user defined source terms for  $k$  and  $\epsilon$ , respectively.

$\mu_t$ , which is the turbulent viscosity, can be found as;

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \quad (2.57)$$

where  $C_\mu$  is a constant.

$C_{1\epsilon}$ ,  $C_{2\epsilon}$ ,  $C_\mu$ ,  $\sigma_k$ ,  $\sigma_\epsilon$  in Eq. 2.51, 2.56 and 2.57 are empirical constants which have the following default values;

Table 2.3. Empirical constants of the  $k - \epsilon$  method, Launder and Sharma(1974).

| $C_{1\epsilon}$ | $C_{2\epsilon}$ | $C_\mu$ | $\sigma_k$ | $\sigma_\epsilon$ |
|-----------------|-----------------|---------|------------|-------------------|
| 1.44            | 1.92            | 0.09    | 1.0        | 1.3               |



## **CHAPTER 3**

### **RESULTS AND DISCUSSION**

#### **3.1. General Information**

For the purpose of this study, a previously studied experimental model which was investigated by Çobanoğlu, 2008 is used (this setup was also used by Kemalli O., 2009 and Yıldırım T., 2010).

Çobanoğlu's model was a closed loop, circulating water safe from any external effect. It had contained two reservoirs, one storage and one model reservoir and a measurement tank. The model reservoir constructed to have the following dimensions; 0.47m width, 0.62m length and 0.40m height. The model reservoir was consisted of three different sections, namely the reservoir, the supply tank and the discharge tank. Since our study is modeling the flow only, there was no need to model the supply and discharge tanks hence only the reservoir was modeled and the computations were carried out for this section.

Çobanoğlu's reservoir and also the reservoir used in our study are presented in Figure 3.1. and Figure 3.2.

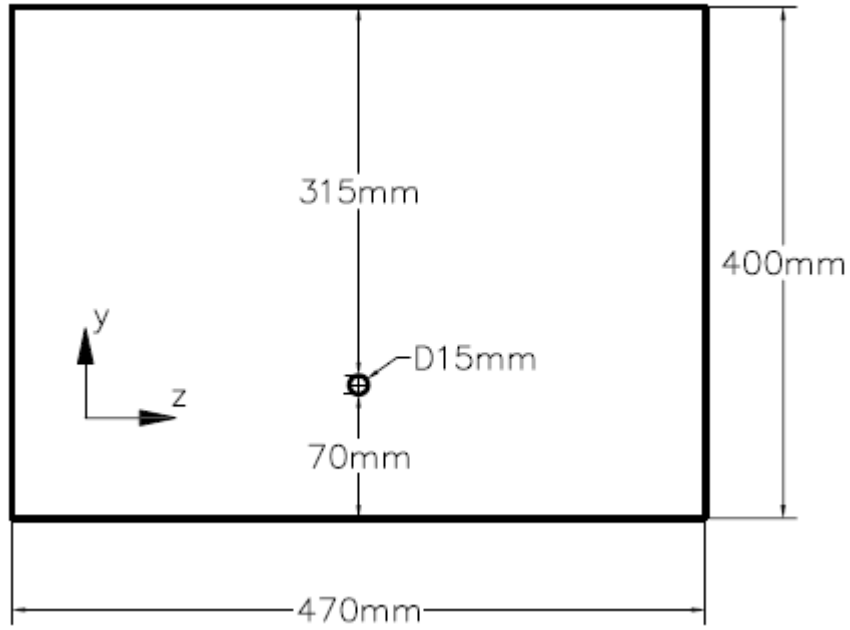


Figure 3.1. Front view of the model setup

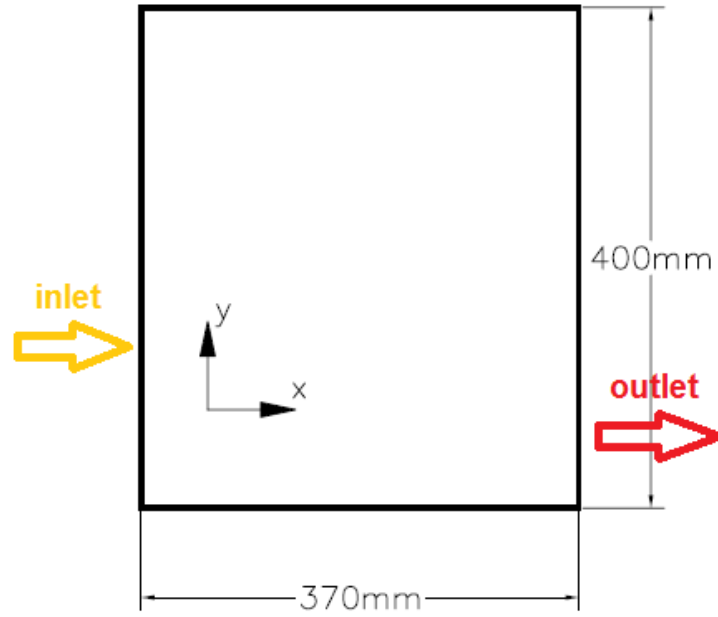


Figure 3.2. Side view of the model setup

In this study, 2D and 3D analytical solutions were carried out then FLUENT procedure was applied for a discharge value of 0.345 lt/s.

An analytical approach using Bernoulli Equations leads to a velocity value of 2.515 m/s as a uniform velocity value through the outlet, resulting from Torricelli's Equation; .

$$V = \sqrt{2gh} \quad (3.1)$$

### 3.2. 2D Potential Flow Solution

There are certain disadvantages of using point sink flow for approaching the problem, as mentioned at Chapter 2. Firstly, the solution predicts the radial velocity,  $V_r$ , as uniform over the orifice surface. Secondly, at  $r = 0$ , this approach fails to provide any solution since it is a singular point in the solution domain.

Graphical representation of velocity potential is presented in Figure 3.3.

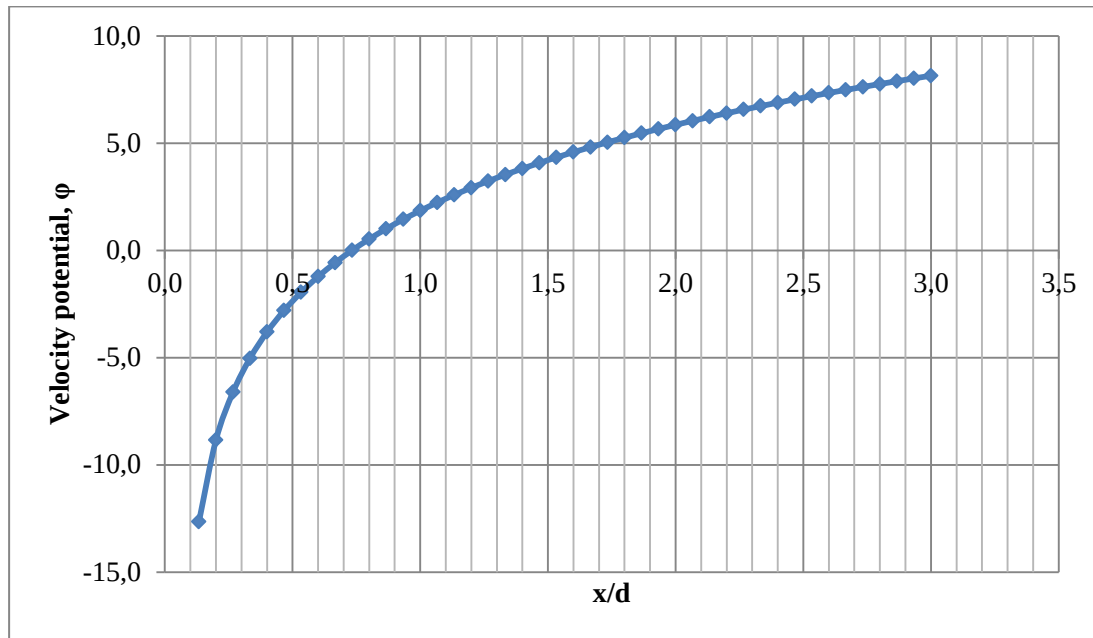


Figure 3.3. Potential flow sink solution velocity potential vs.  $x/d$

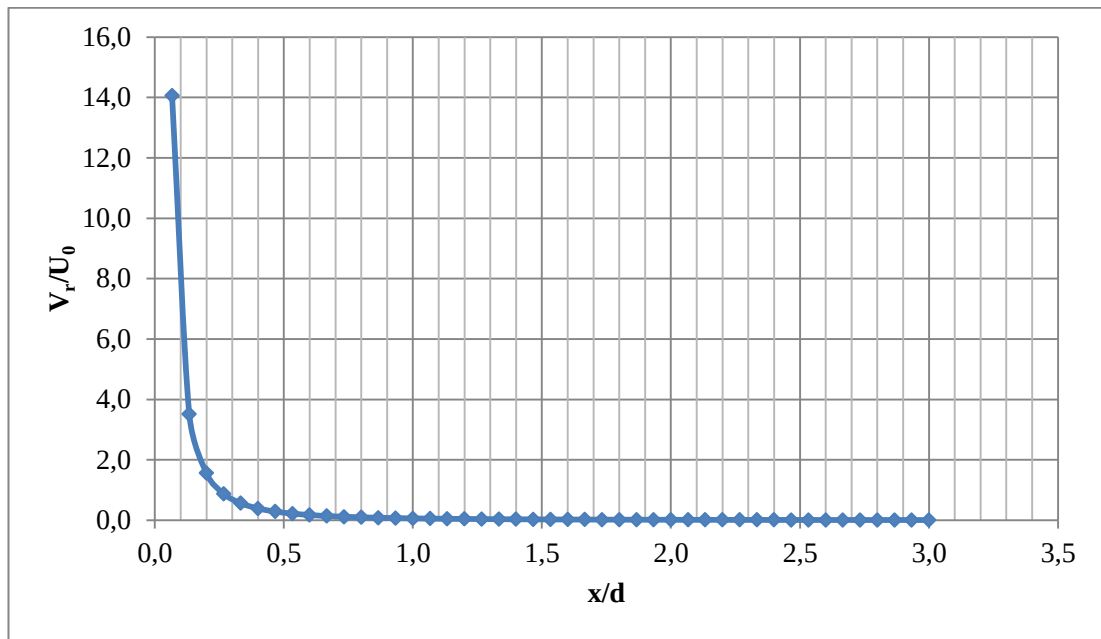


Figure 3.4.  $V_r/U_0$  vs.  $x/d$

As it can be clearly seen from the Figure 3.4. 2D potential flow solution reaches very high velocity values close to the orifice, which is certainly not realistic.

### 3.3. 3D Potential Flow Solution – Shammaa

For the solution of the problems, the procedure which was explained in detail at Chapter 2.3 is used. The complex second degree integrals were calculated by means of a commercially available software, MATLAB. Related MATLAB codes are presented at the Appendices section.

Graphical representations obtained from the calculations are presented below.

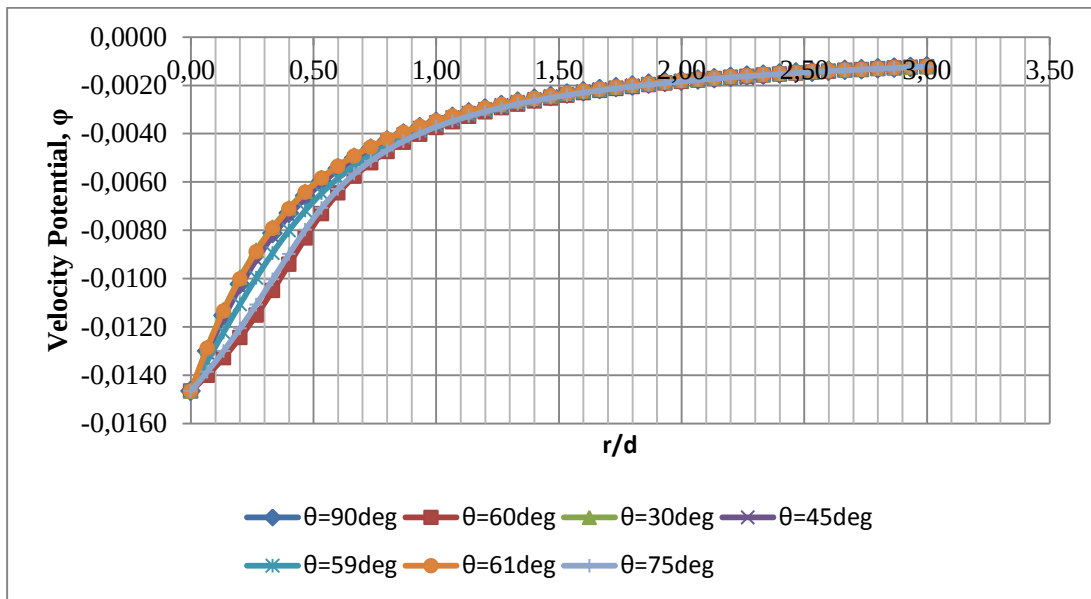


Figure 3.5. Velocity potential  $\phi$  vs.  $r$ , for different  $\theta$  values

It is seen that the velocity potential for several angles show the same tendency however for some angles velocity potential shows an erroneous distribution, considering Figure 3.5.

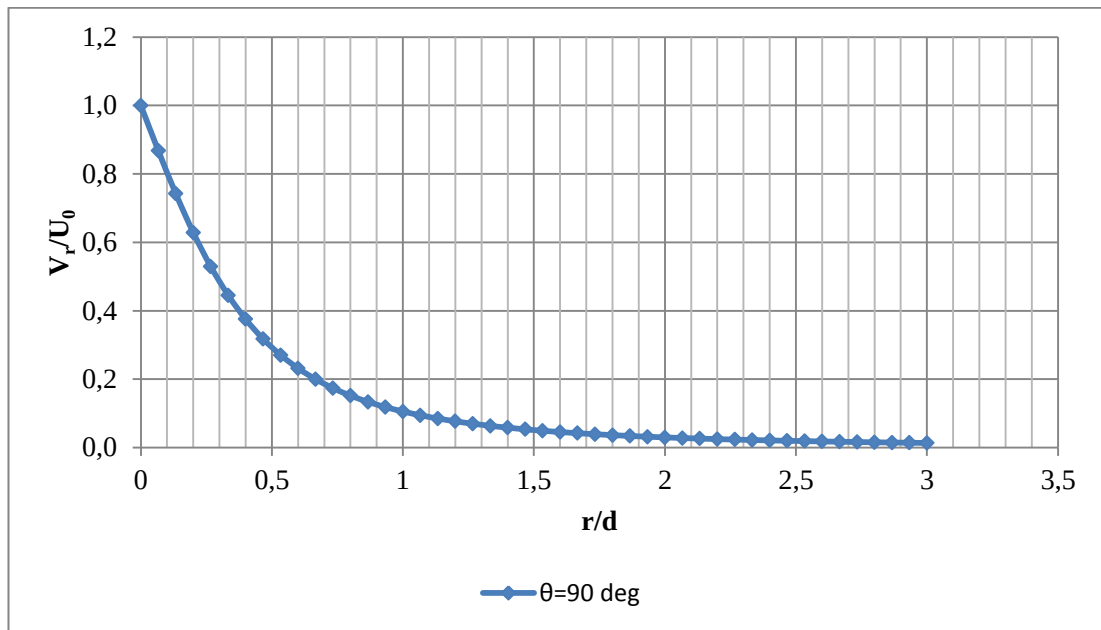


Figure 3.6.  $V_r/U_0$  vs.  $r/d$ , with fixed  $\theta$  value

Using Figure 3.6. it can be concluded that the velocity becomes less than one tenth of the maximum velocity after  $r/d$  becomes less than one.

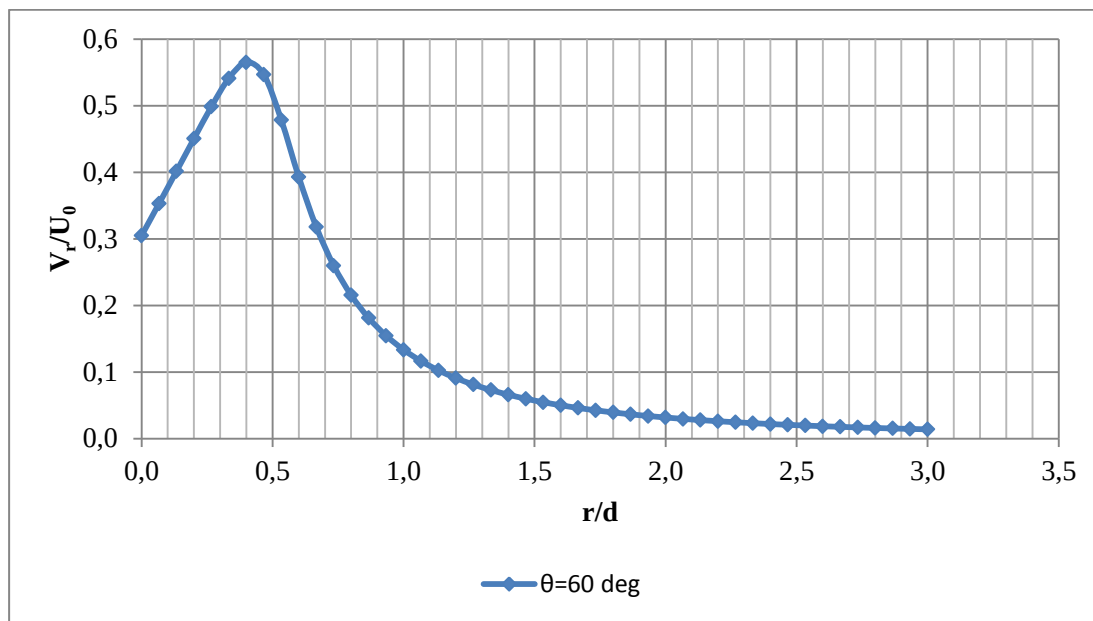


Figure 3.7.  $V_r/U_0$  vs.  $r/d$ , with fixed  $\theta$  value

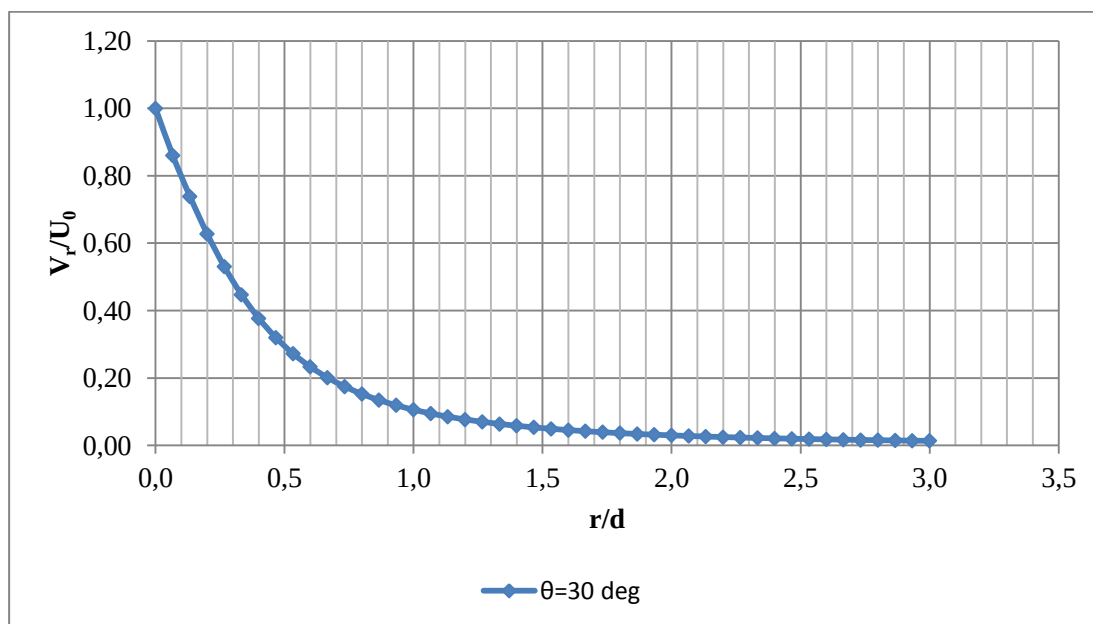


Figure 3.8.  $V_r/U_0$  vs.  $r/d$ , with fixed  $\theta$  value

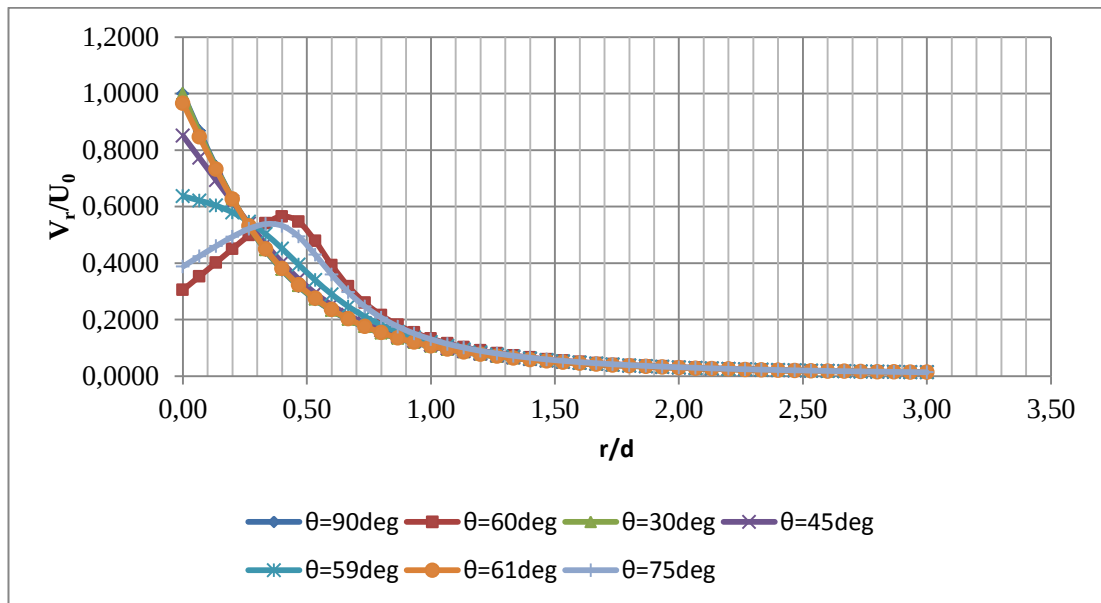


Figure 3.9.  $V_r/U_0$  vs.  $r/d$ , for different  $\theta$  values

Figure 3.9. shows that the solution fails to provide an accurate result for some angles degrees, for example at 60 degrees, as the velocity values are getting lower close to the orifice whereas they should rise.

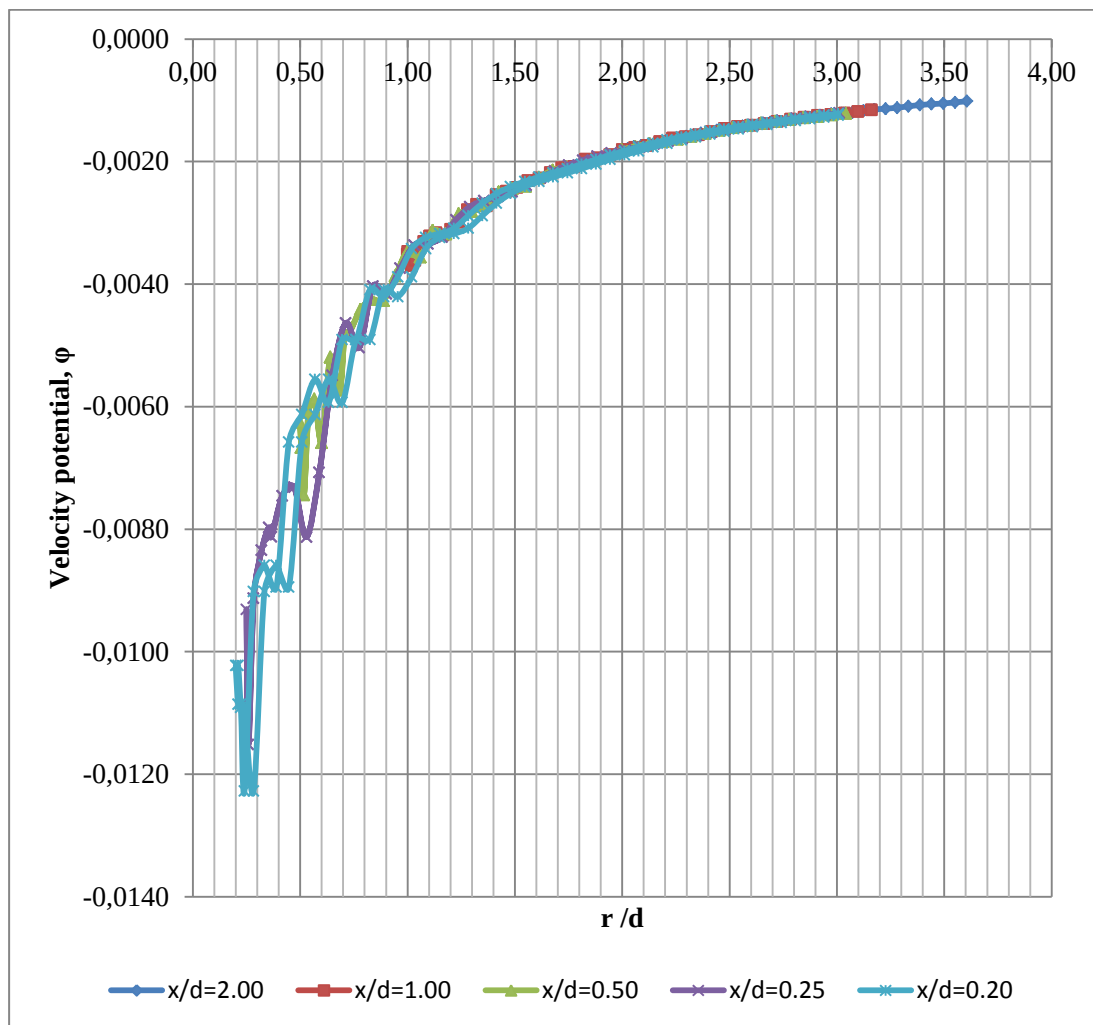


Figure 3.10. Velocity potential  $\phi$  vs.  $r/d$ , with different  $x$  values

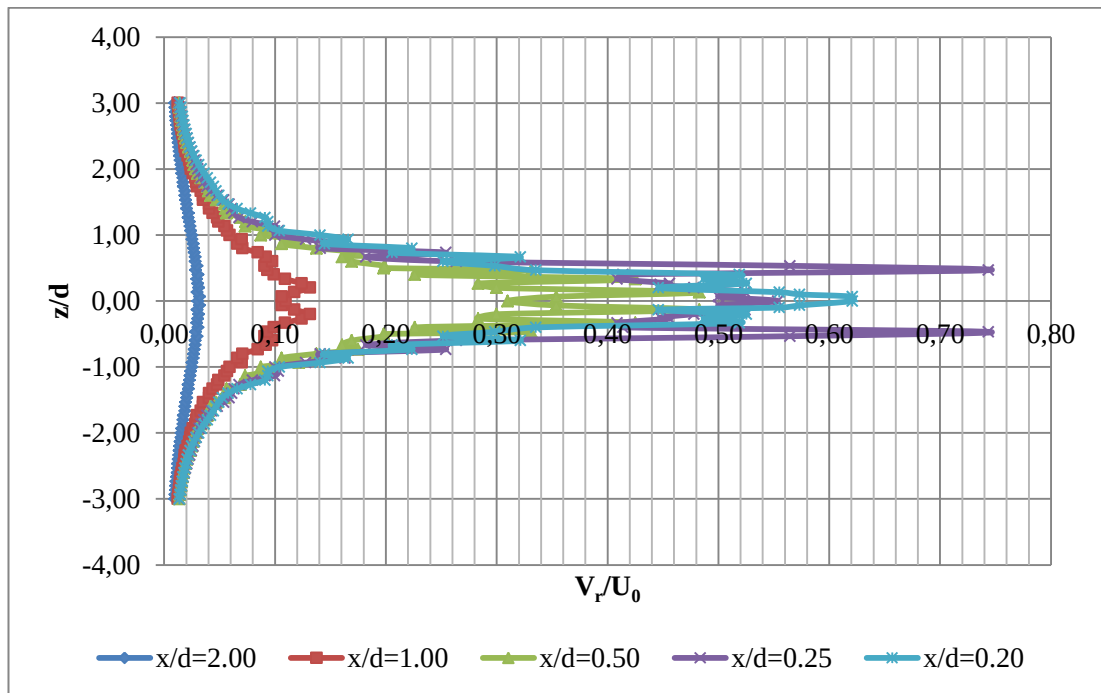


Figure 3.11.  $V_r/U_0$  vs.  $z/d$ , with different  $x$  values

Figure 3.11. shows  $V_r/U_0$  for a symmetrical half of the orifice on  $y$ - $z$  plane. It could be clearly seen from the figure that velocity values are indented and the method fails to provide an accurate solution on  $y$ - $z$  plane.

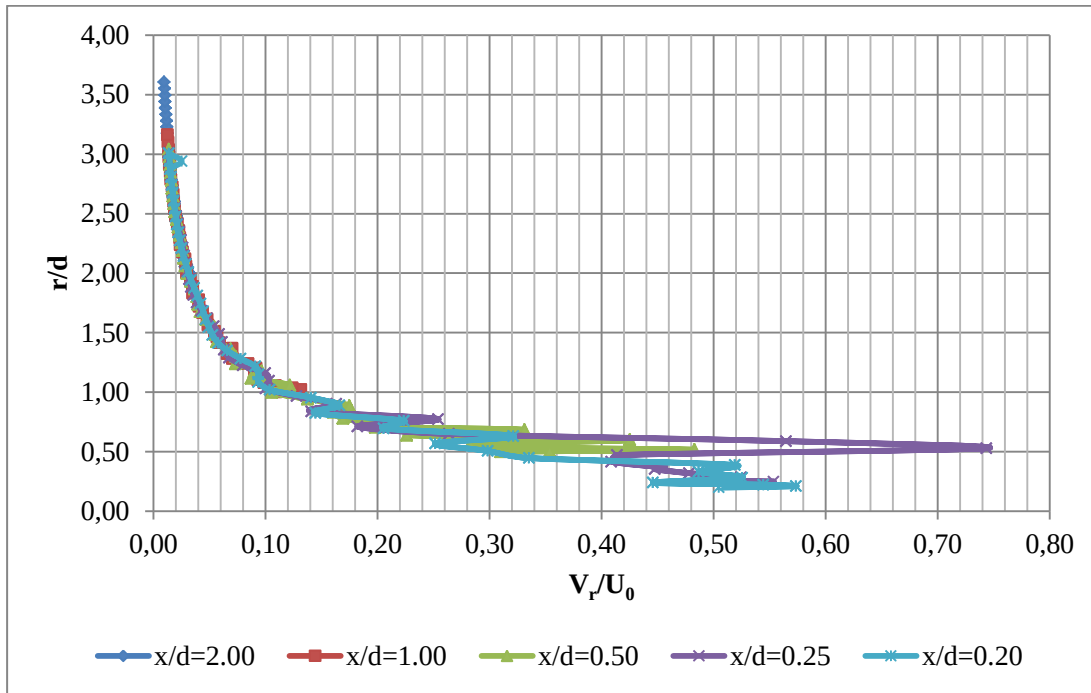


Figure 3.12.  $V_r/U_0$  vs.  $r/d$ , with different  $x$  values

### 3.4. 3D Potential Flow Solution – Bryant

Solutions obtained from the Bryant procedure which was explained in detail at Chapter 2.4 is presented at this section. The complex second degree integrals were calculated by means of a commercially available software, MATLAB. Related MATLAB codes are presented in the Appendices section.

Graphical representations obtained from the calculations are presented below.

It can be concluded that away the orifice the flow can be treated to be inviscid and irrotational, thus potential flow solution can be used. However, in the vicinity of the orifice, viscous effects should be included. This conclusion is also supported by the study of Göbelez (2008) who investigated experimentally threshold of sediment movement due to orifice induced velocity field. The initiations of the particles occurred at maximum  $x/d=2$ .

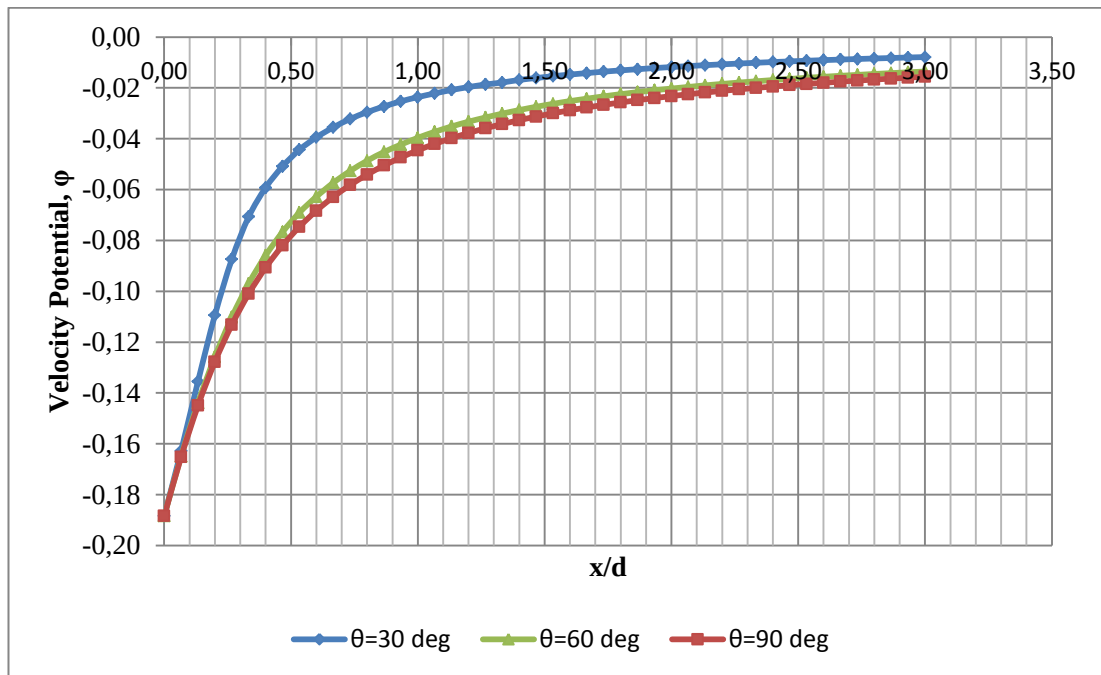


Figure 3.13. Velocity potential  $\phi$  vs.  $x$ , with different  $\theta$  values

Velocity potential shows a gradual decrease at Figure 3.13. as expected and when compared with Figure 3.9. it could be concluded that the Bryant method provides a more accurate velocity potential distribution than the Shammaa method.

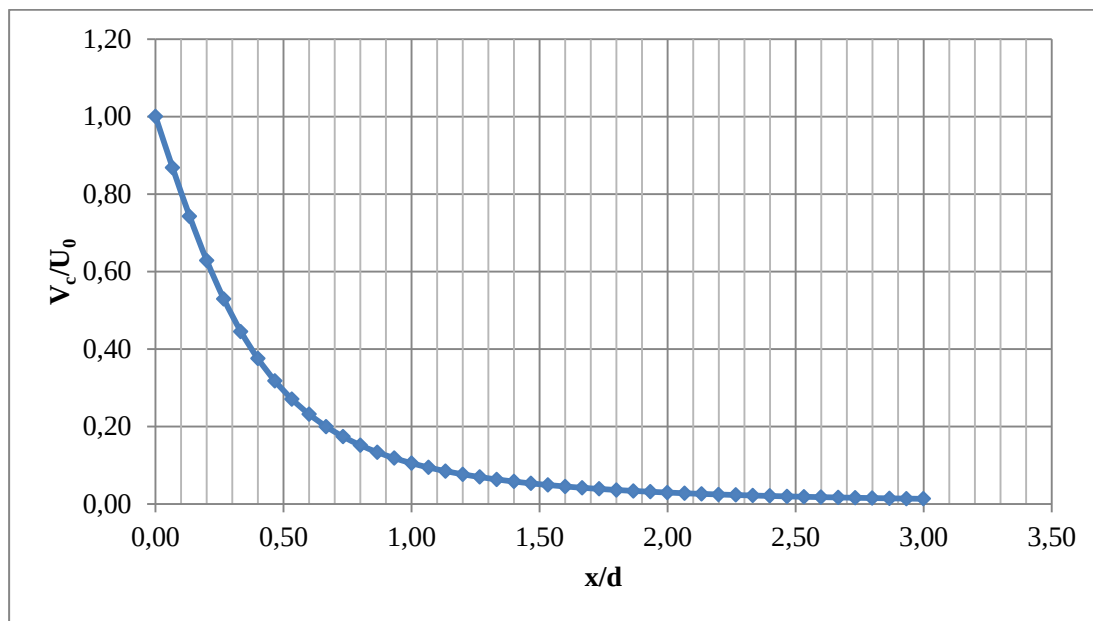


Figure 3.14.  $V_r/U_0$  vs.  $x/d$

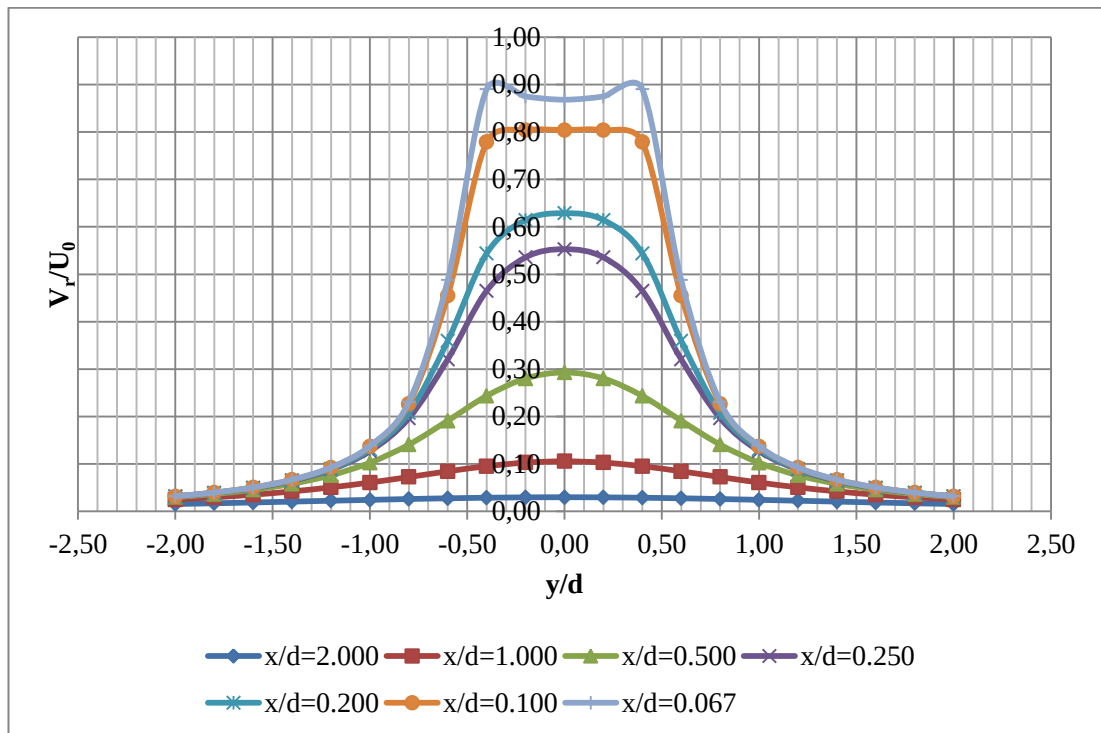


Figure 3.15.  $V_r/U_0$  vs.  $y/d$ , with fixed  $x$  values

In Figure 3.15. after  $x/d=0.1$  a deformation at the  $y$ - $z$  plane velocity function shows as if the centerline velocity is less than the velocity at the outer regions, which can not be considered as real in hydromechanical point of view.

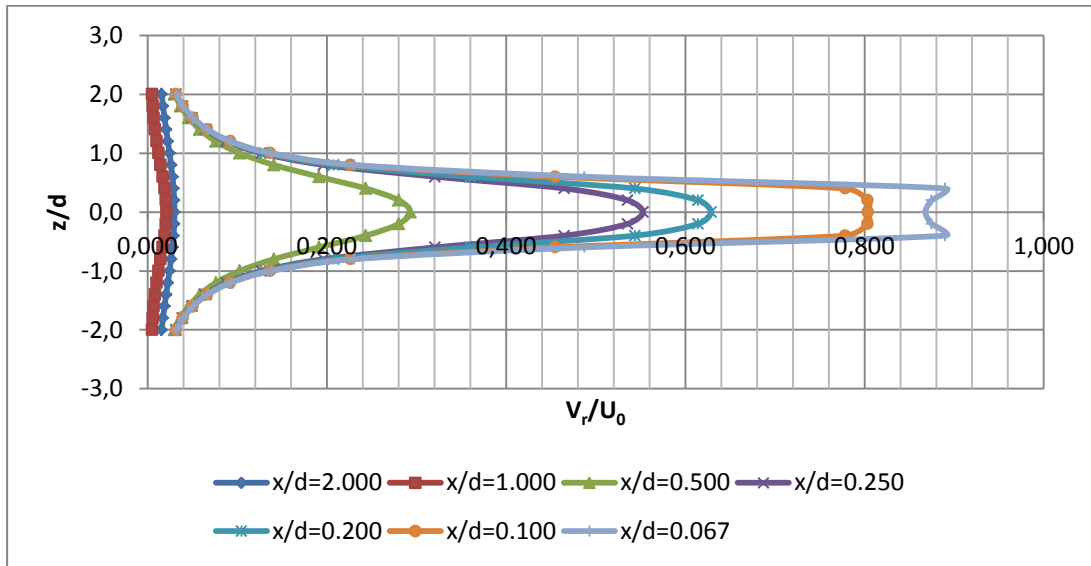


Figure 3.16.  $V_r/U_0$  vs.  $z/d$ , with fixed  $x$  values

In Figure 3.16. after  $x/d=0.1$  a deformation at the  $y$ - $z$  plane velocity function shows as if the centerline velocity is less than the velocity at the outer regions, which can not be considered as real in hydromechanical point of view.

### 3.5. Fluent CFD Solution

The orifice induced velocity field is actually a result of three dimensional, viscous and rotational flow. Therefore, simulation of the flow by using computational flow dynamics techniques was decided. In the present study a commercial software, namely ANSYS FLUENT is used. In order to be able to make a comparison with the potential flow solution, both inviscid and viscous models were constructed. In order to investigate different solutions provided by different models and to take the friction into consideration, inviscid, laminar,  $k - \epsilon$  turbulence method and volume of fluid method with  $k - \epsilon$  turbulence approach were applied.

Firstly, a simulation of the model of Çobanoğlu was investigated by an inviscid approach, then a model which is scaled 100 times bigger than the original model was taken into consideration in order to investigate how FLUENT offers a solution for an imaginative real structure, similar to an existing reservoir. Calculations continued with this 100 times enlarged model since it is important to see real case solutions, and since results are presented as dimensionless parameters, one can be able to compare the results of different scale models.

It is important to note that, the analysis was initiated with the acceptance that the whole tank is full with water. Time-step for the calculations was taken as 0.1 seconds, and an adaptive time-step solution was carried out to see if a time-step of 0.1 seconds was suitable.

Solutions obtained from the analysis carried out by ANSYS FLUENT computational fluid mechanics software are presented in this section.

Graphical representations of the solutions are presented here. Calculation report is presented at the Appendices section.

### **3.5.1. Inviscid Solution of the Çobanoğlu Model**

The inviscid FLUENT solution for the Çobanoğlu model is presented in this section.

Here, the present study displays several flow characteristics by using advantage of a cfd solution, as it is possible to obtain graphical and tabular solutions. Several graphical solutions are presented in this section.

Residuals are presented at the Appendices section.

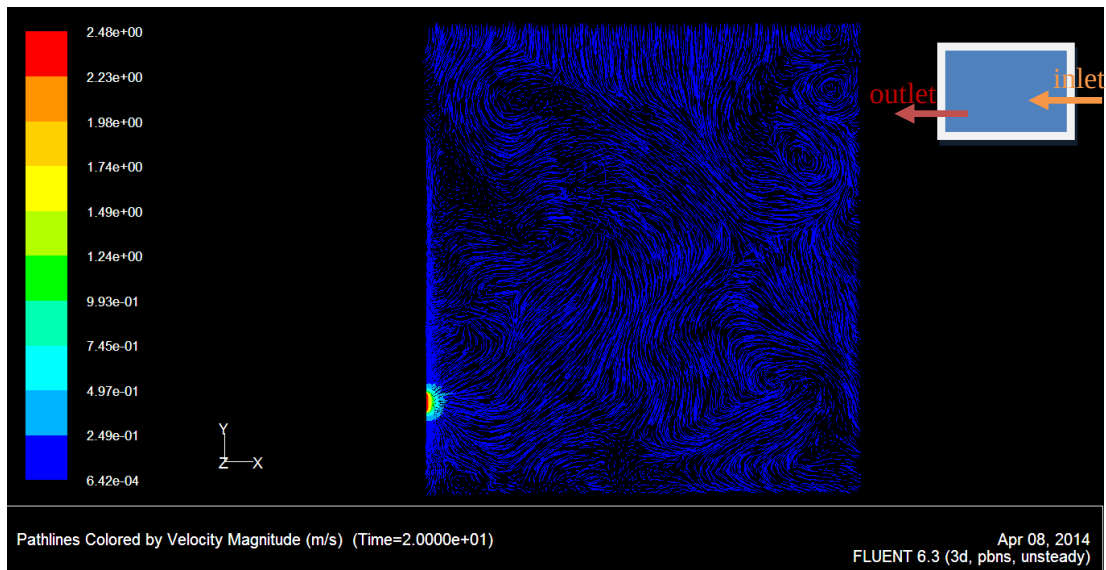


Figure 3.17. Pathlines by velocity magnitude

Considering Figure 3.17. a uniform pathline tendency can not be seen through the whole region, however in Figure 3.18. there is a similar pattern very close to the orifice region.

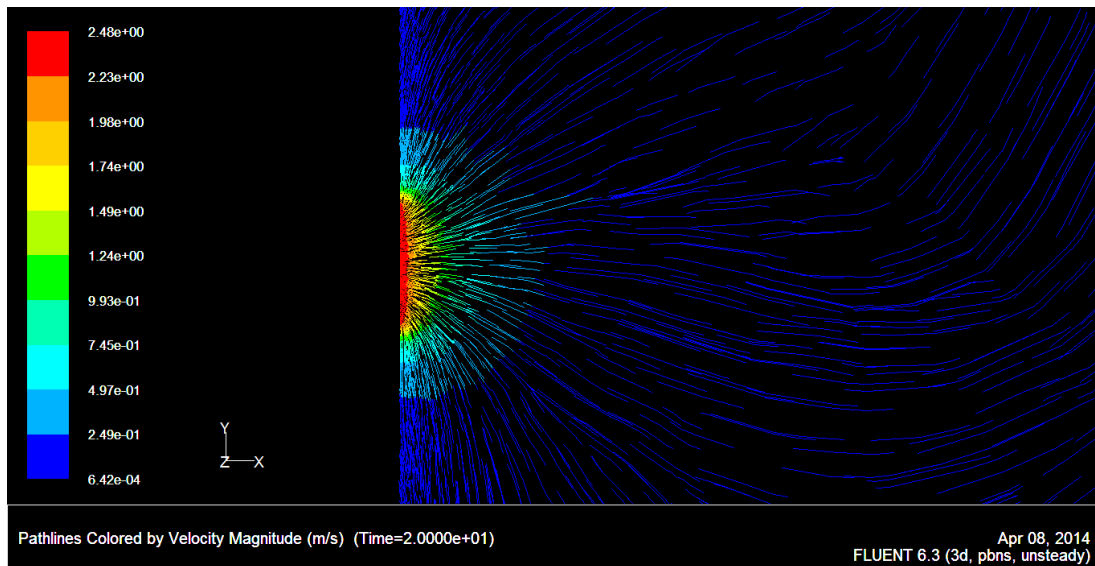


Figure 3.18. Close up of the orifice area for the pathlines by velocity magnitude

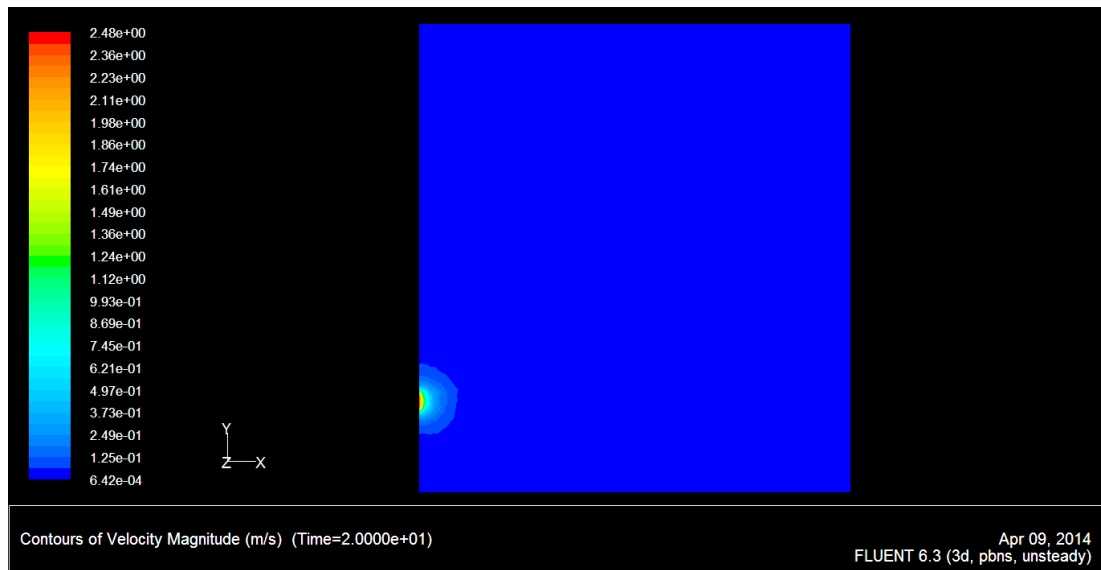


Figure 3.19. Contours of velocity magnitude

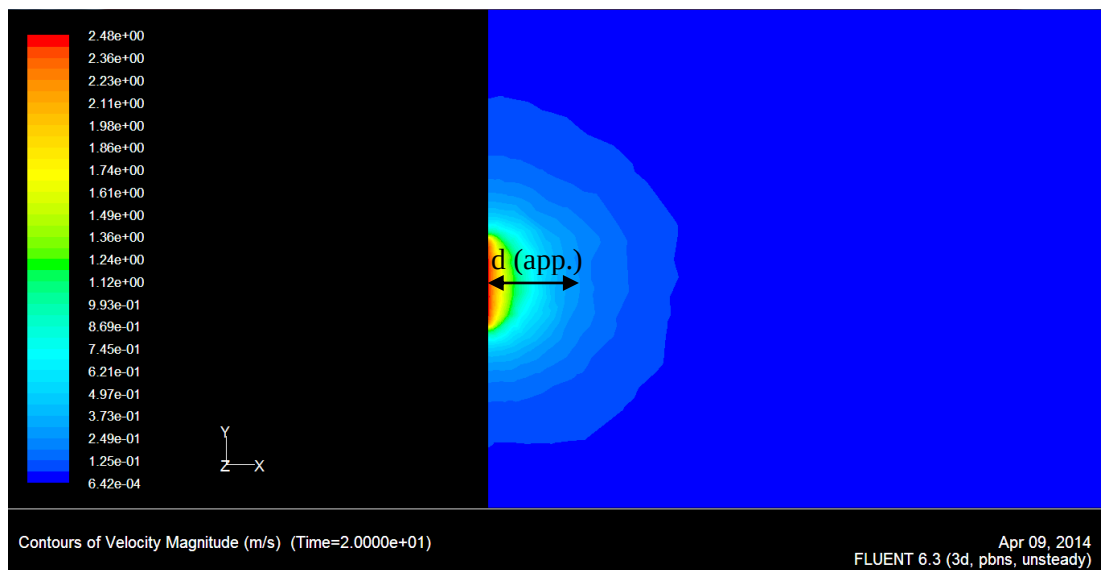


Figure 3.20. Close up of the orifice area for the contours of velocity magnitude

Figures 3.19. and 3.20. show velocity variation behind orifice on an x-y plane cutting the orifice and the box into two equal parts, i.e. the symmetry axis. At a distance approximately  $d$  behind the orifice, velocity magnitude reduces to one tenth of the maximum velocity value.

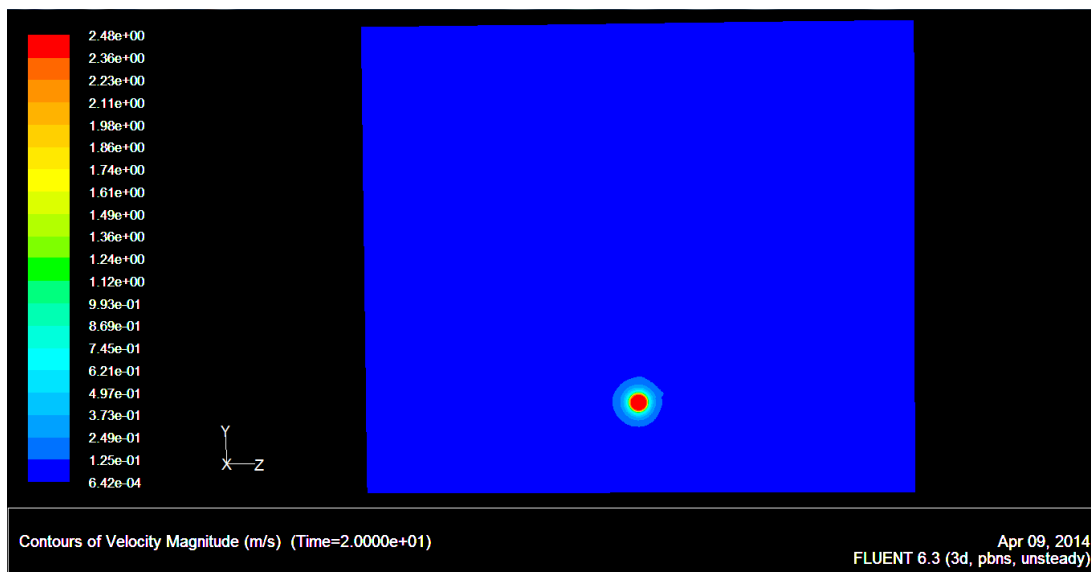


Figure 3.21. Contours of velocity magnitude

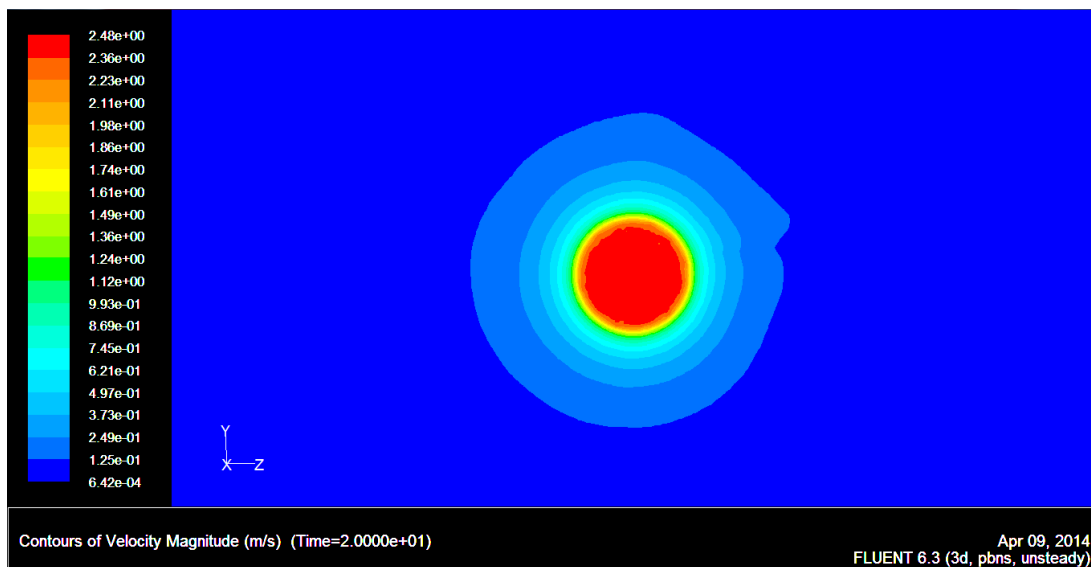


Figure 3.22. Close up of the orifice area for the contours of velocity magnitude

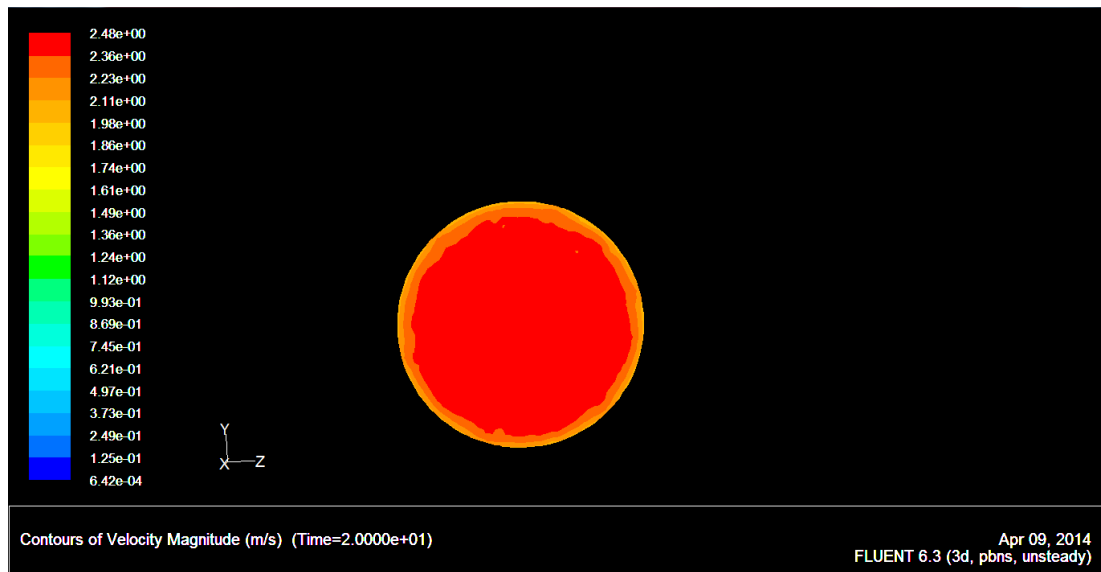


Figure 3.23. Velocity magnitude contours of the orifice area only

Considering Figure 3.20 to Figure 3.23. it can be concluded that the maximum velocity is 2.45 m/s and occurs at the center of the orifice and for a large part of the orifice very close velocity values are seen. Since this solution is inviscid, wall effects are not taken into consideration therefore velocity is not zero on the walls and decreases gradually.

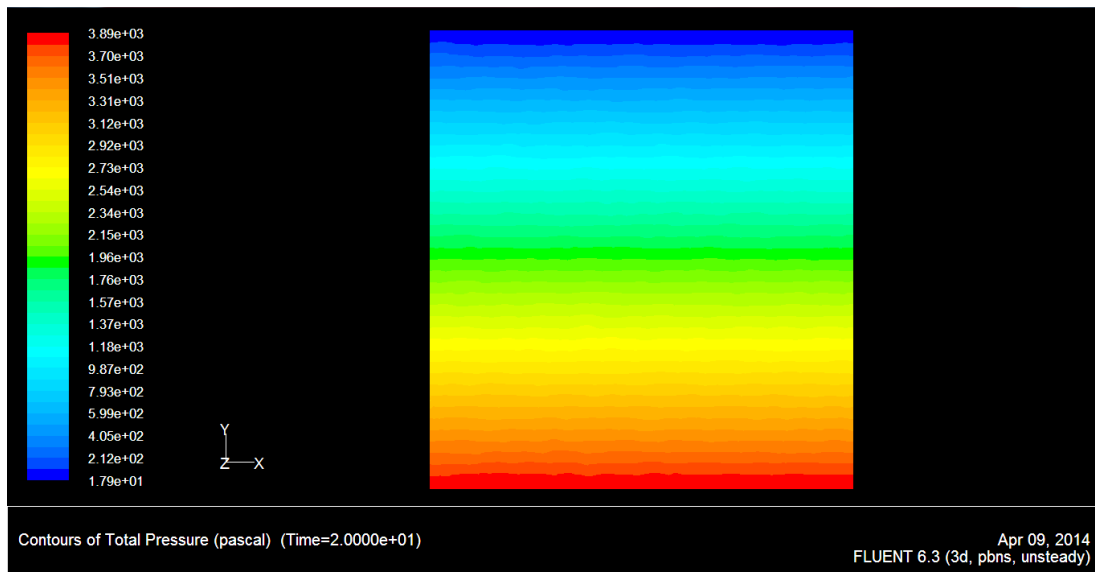


Figure 3.24. Pressure contours



Figure 3.25. Pressure contours of the orifice area

Pressure distribution is almost uniform along the orifice, with a little difference between the upper and lower part of the orifice as can be seen in Figure 3.25.

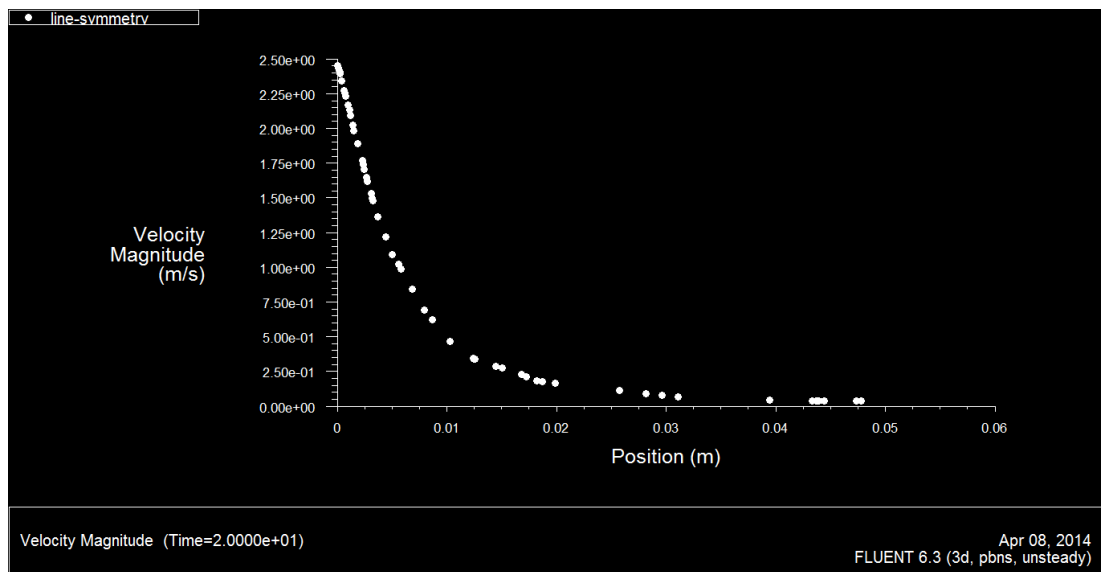


Figure 3.26. Velocity magnitude readings of the centerline velocity

Considering Figure 3.26. it is seen that after approximately  $x/d=1$ , velocity magnitude becomes less than one tenth of the maximum velocity.

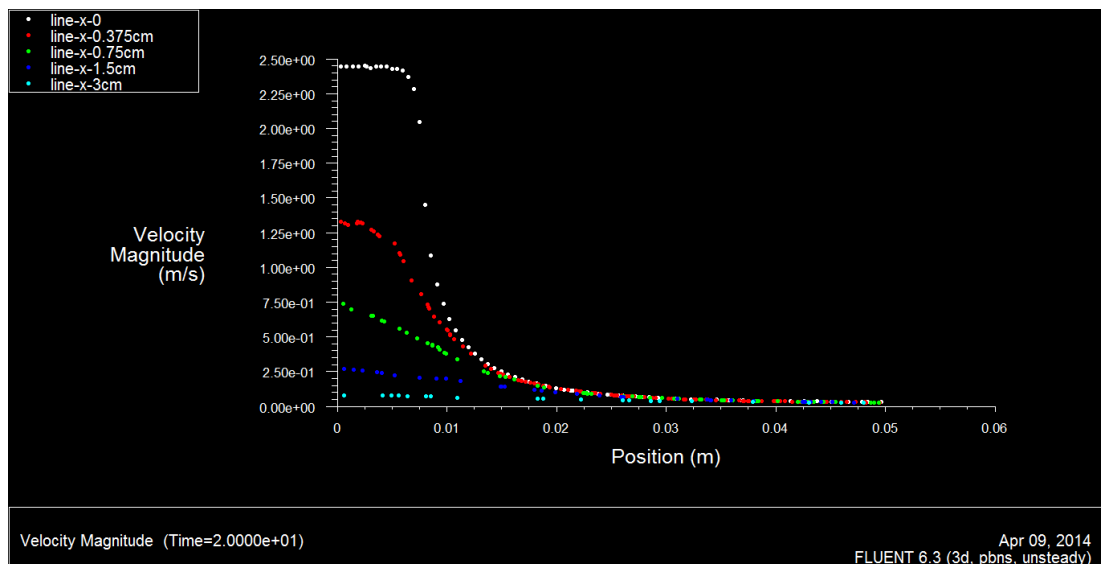


Figure 3.27. y-z plane velocity magnitude on z-axis for different x values

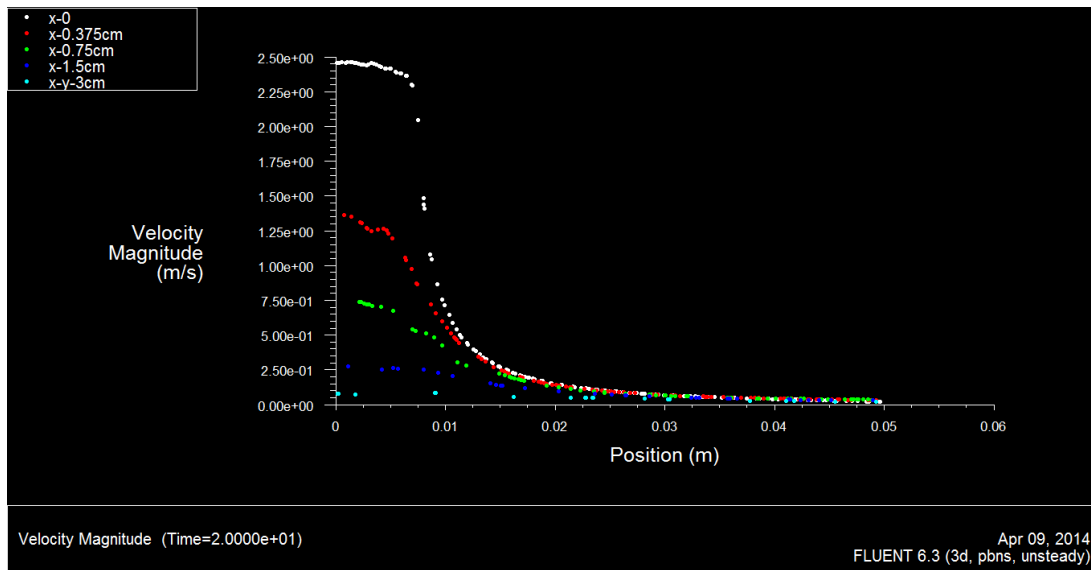


Figure 3.28. y-z plane velocity magnitude on y-axis for different x values

In Figure 3.27. and Figure 3.28. graphs are presented along the planes cutting the orifice and the box into two equal parts, i.e. the symmetry plane. It is seen that rate of decrease in velocity magnitude is higher close to the orifice region. As it was seen before, after the distance approximately equals to the orifice diameter, velocity at the orifice section( $x=0$ ) becomes less than one tenth of the maximum velocity value. In addition, since the wall effects are not taken into consideration for this solution, velocity continues to decrease on the wall segments.

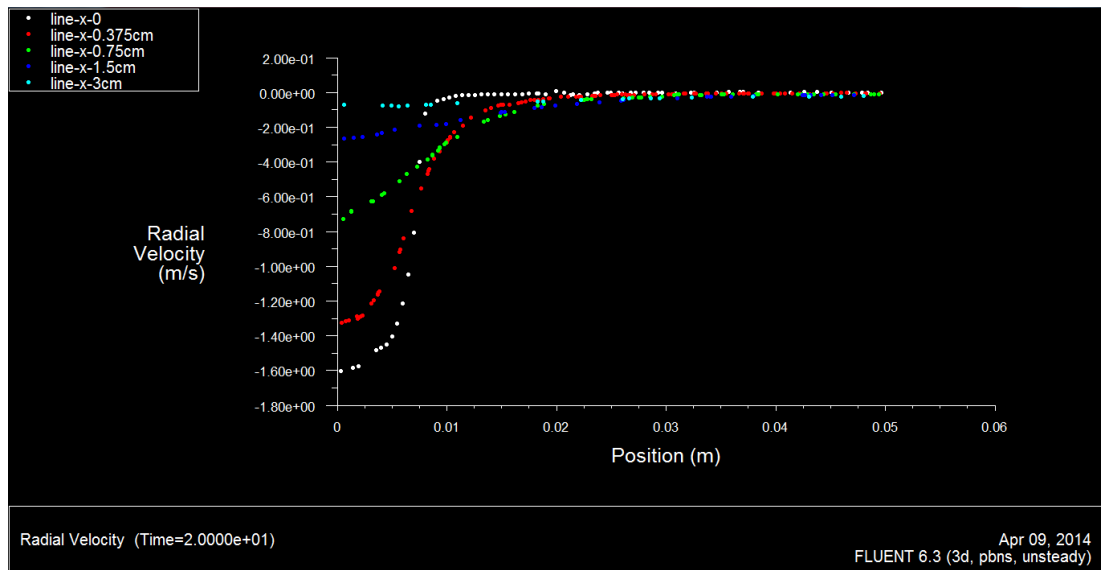


Figure 3.29.  $V_r$  vs.  $z$ , for different  $x$  values

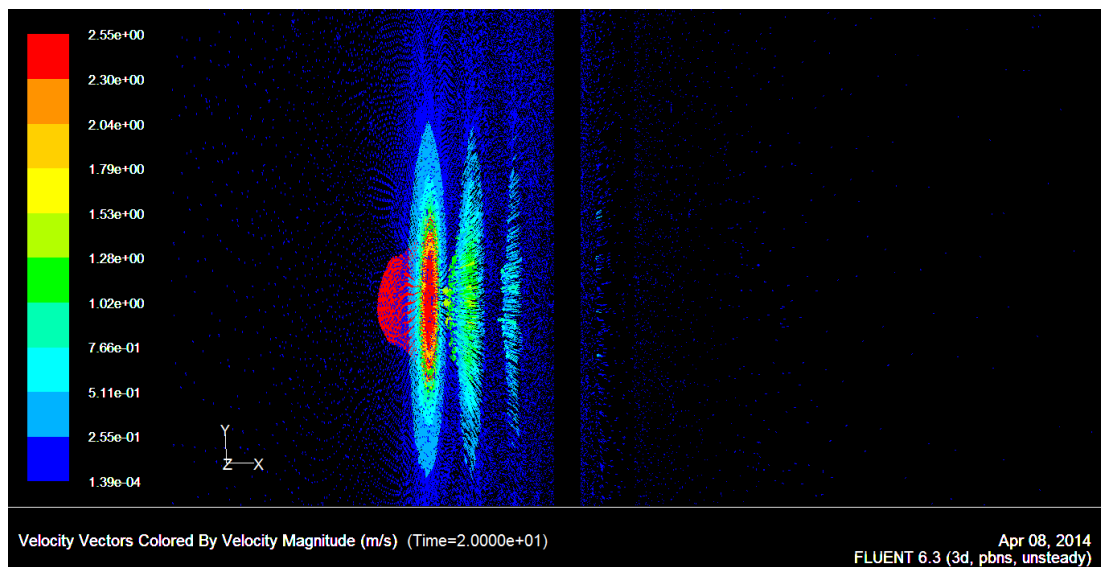


Figure 3.30. Velocity vectors for  $x=0, 0.375\text{cm}, 0.75\text{cm}, 1.5\text{cm}, 3\text{cm}$

Figure 3.30. shows velocity vectors on  $y$ - $z$  plane for different  $x$  values. It can be seen that after  $x=2d$  there is almost no motion towards the orifice.

### 3.5.2. Inviscid Solution of the Enlarged Model

The inviscid FLUENT solution for the enlarged model is presented in this section. Apart from the numerical values, the several flow properties such as centerline velocity, orifice pressure and y-z plane velocity show similar trends with the small model, as a natural consequence of inviscid flow.

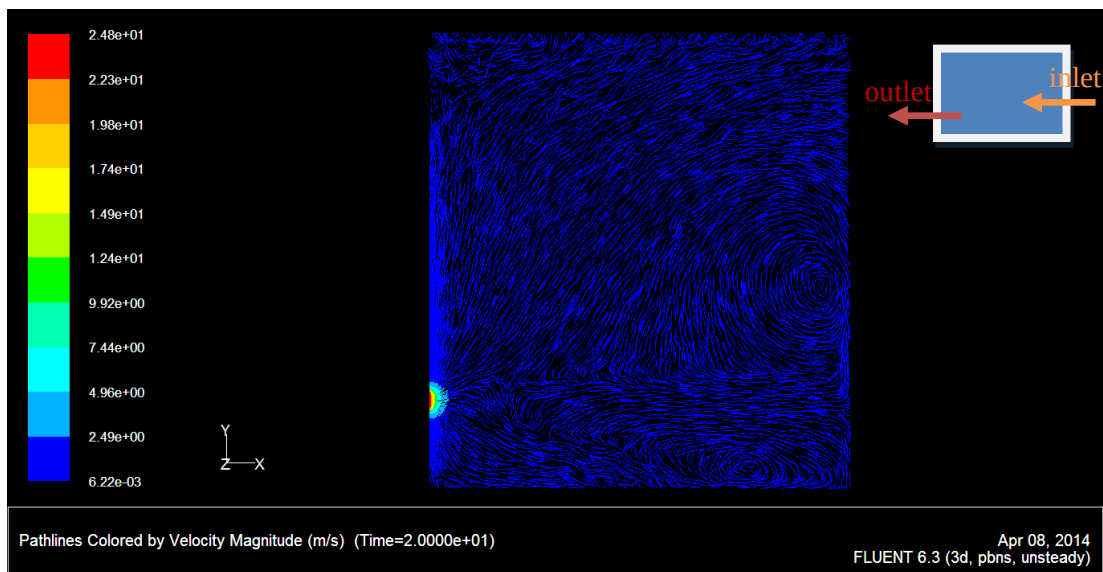


Figure 3.31. Pathlines by velocity magnitude

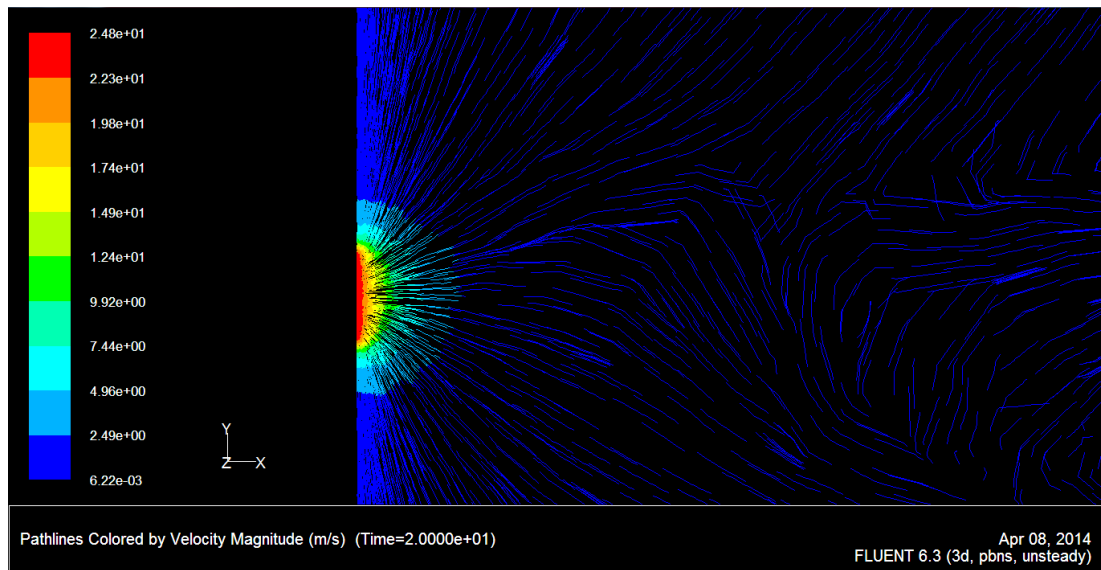


Figure 3.32. Close up of the orifice area for the pathlines by velocity magnitude

Considering Figure 3.31. and Figure 3.32. a uniform pathline pattern can not be seen.

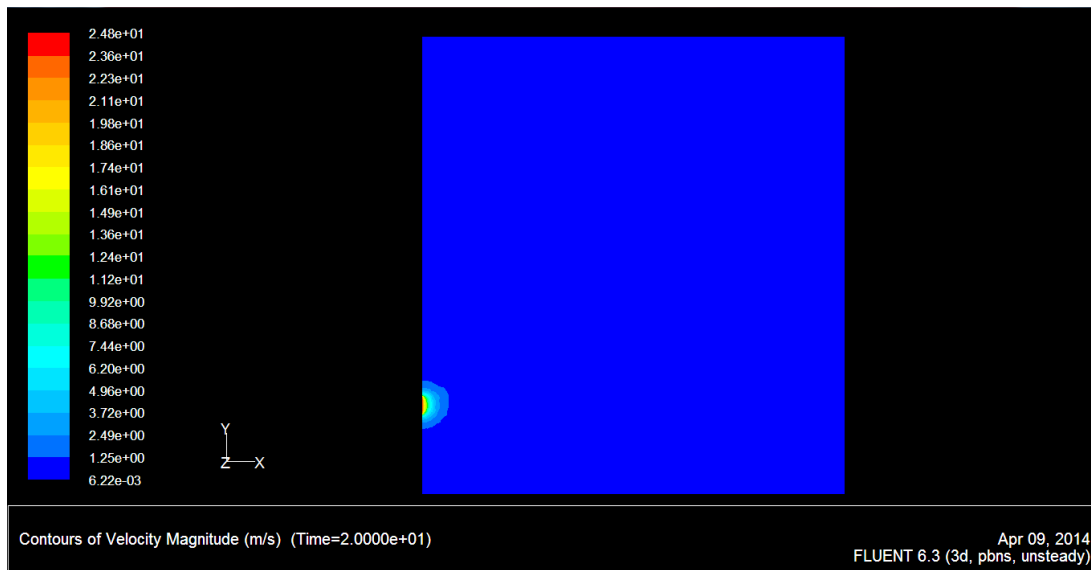


Figure 3.33. Contours of velocity magnitude

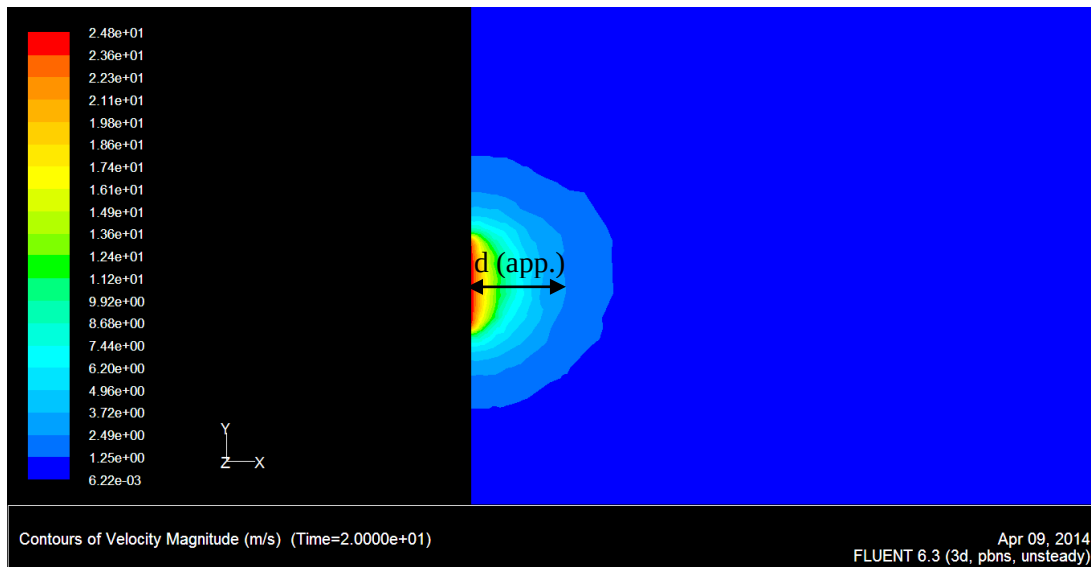


Figure 3.34. Close up of the orifice area for the contours of velocity magnitude

Figures 3.33. and 3.34. show velocity variation behind orifice on an x-y plane cutting the orifice and the box into two equal parts, i.e. the symmetry axis. As it was the case for the inviscid Çobanoğlu model, it is again seen that the velocity magnitude

decreases to one tenth of the maximum velocity value at a distance approximately  $d$  behind the orifice.

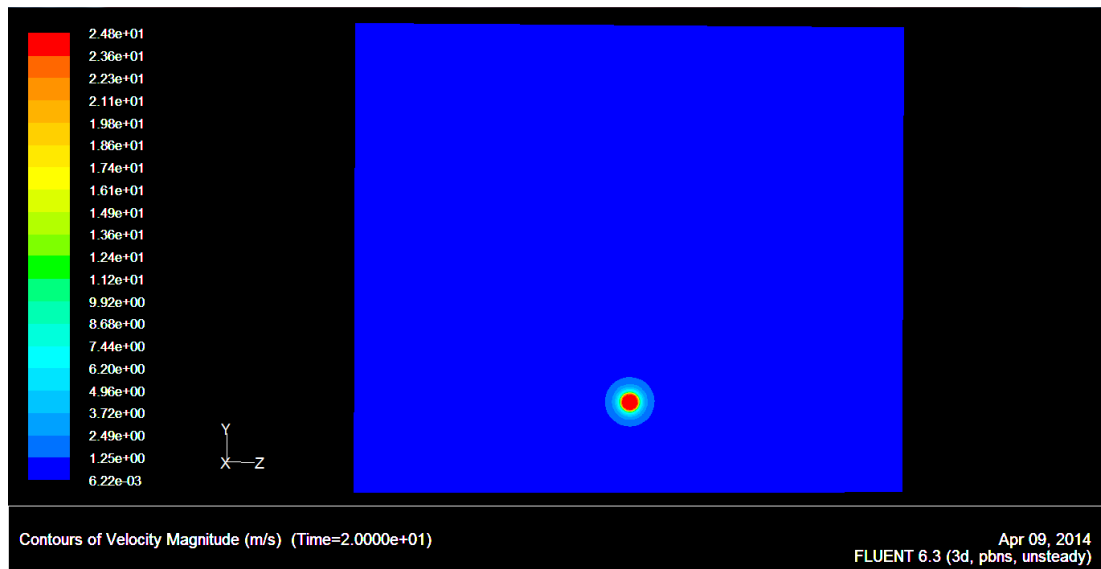


Figure 3.35. Contours of velocity magnitude

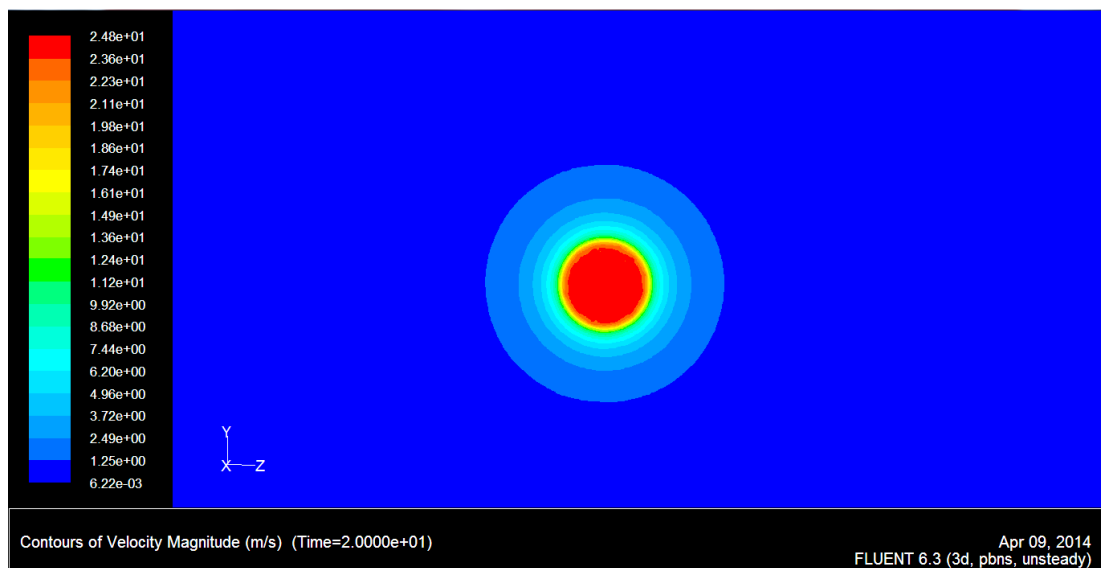


Figure 3.36. Close up of the orifice area for the contours of velocity magnitude

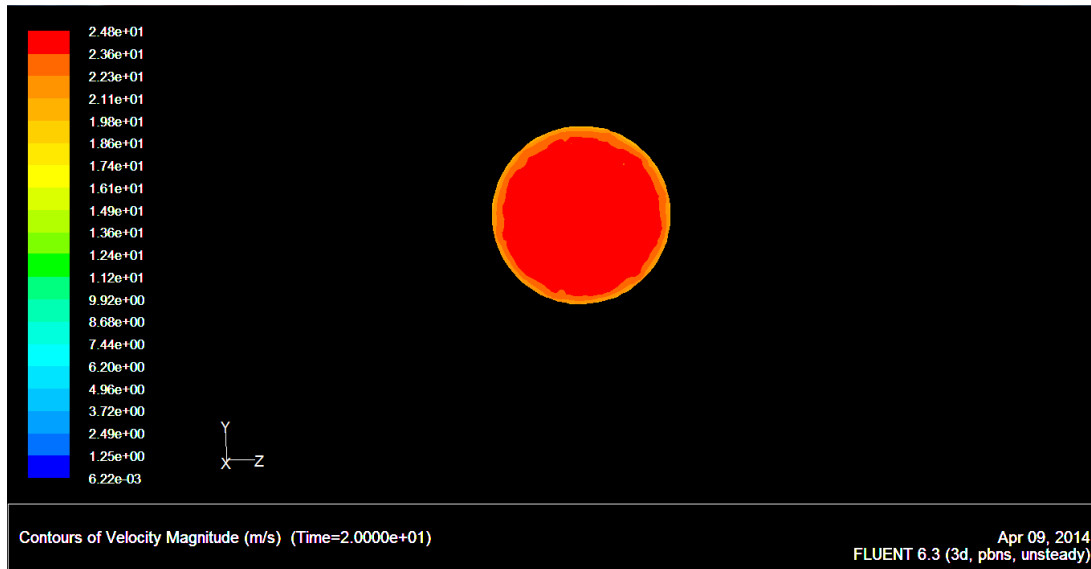


Figure 3.37. Velocity magnitude contours of the orifice area only

Considering Figure 3.35 to Figure 3.37. it can be concluded that the maximum velocity is 24.47 m/s and occurs at the center of the orifice and for a large part of the orifice very close velocity values are seen. Velocity magnitude is decreasing towards the perimeter. Since this solution is inviscid, wall effects are not taken into consideration therefore velocity is not zero on the walls and decreases gradually.

Comparing with the Çobanoğlu model, it can be seen that the velocity magnitude rises to approximately ten times the velocity magnitude of the original model, and the trends are very similar.

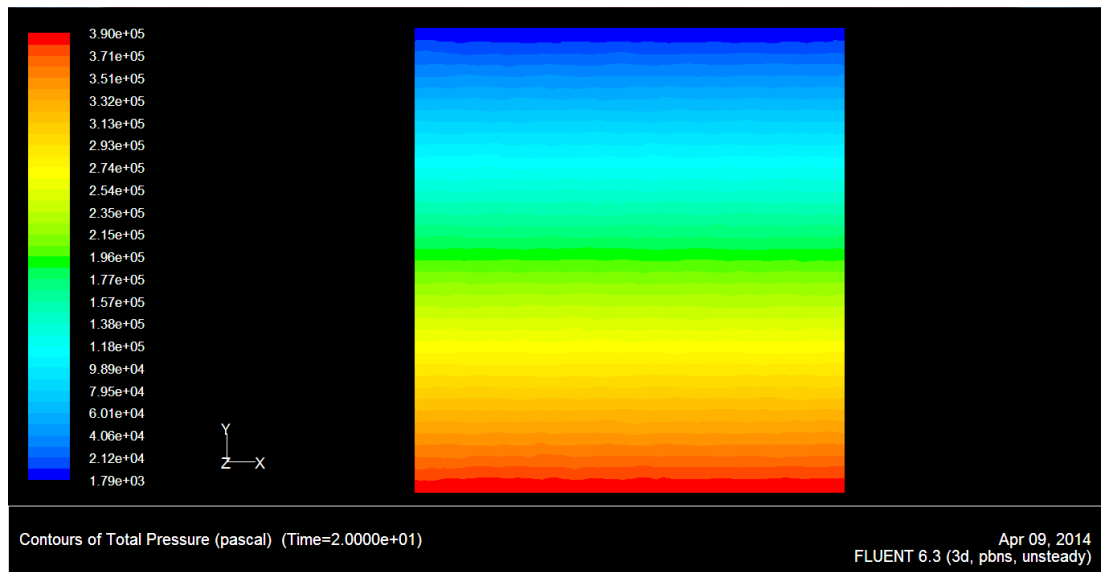


Figure 3.38. Pressure contours



Figure 3.39. Pressure contours of the orifice area

Pressure distribution is changing between  $2.83 \times 10^5$  Pa to  $3.32 \times 10^5$  Pa with the upper part of the orifice being in the lower pressure section, as can be seen in Figure 3.39.

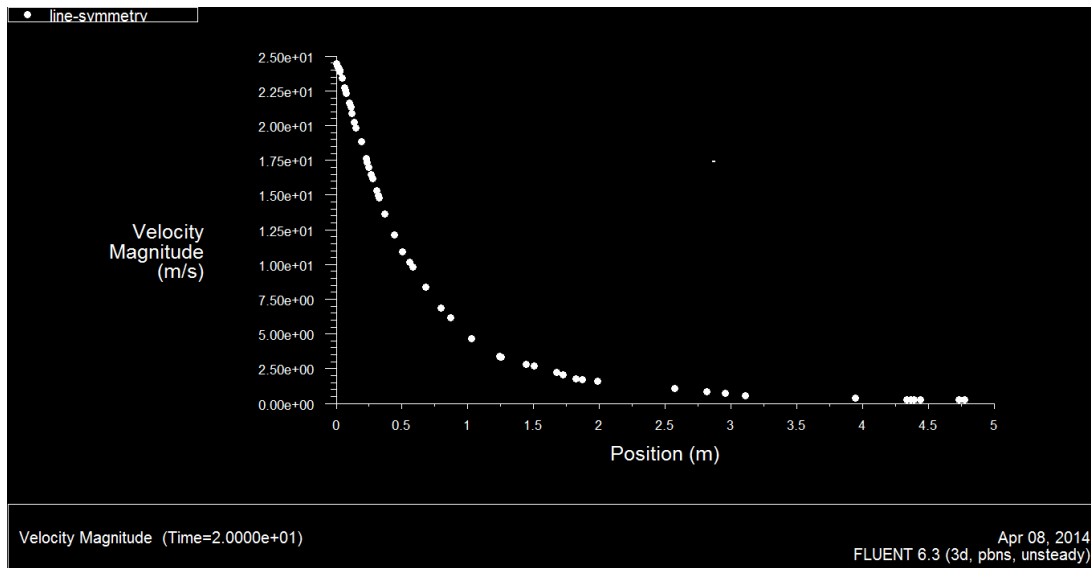


Figure 3.40. Velocity magnitude readings of the centerline velocity

Considering Figure 3.40, it is seen that after approximately  $x/d=1$ , velocity magnitude becomes less than one tenth of the maximum velocity, as it was also the case in the Çobanoğlu model.

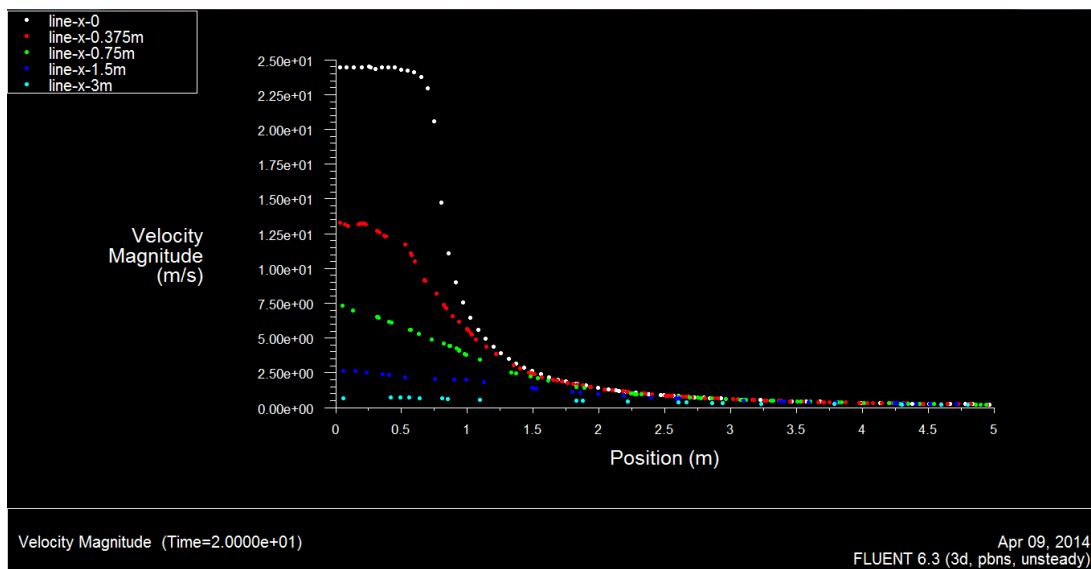


Figure 3.41. y-z plane velocity magnitude on z-axis for different x values

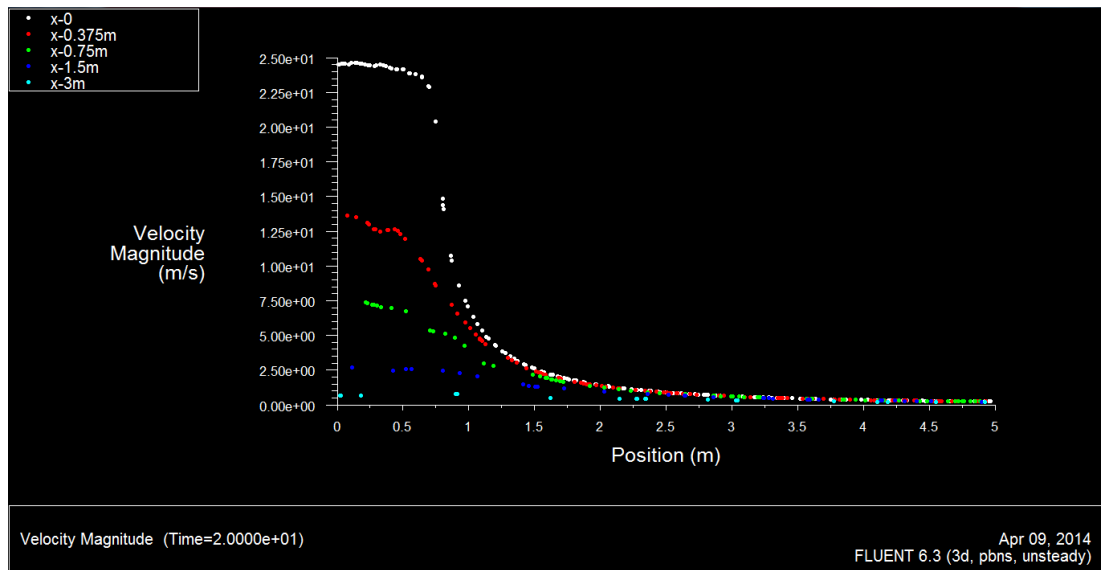


Figure 3.42. y-z plane velocity magnitude on y-axis for different x values

In Figure 3.41. and Figure 3.42. graphs are presented along the planes cutting the orifice and the box into two equal parts, i.e. the symmetry plane. It is seen that rate of decrease in velocity magnitude is higher close to the orifice region. As it was seen before, as well as the Çobanoğlu model, after the distance approximately equals to the orifice diameter, velocity at the orifice section( $x=0$ ) becomes less than one tenth of the maximum velocity value. In addition, since the wall effects are not taken into consideration for this solution, velocity continues to decrease on the wall segments.

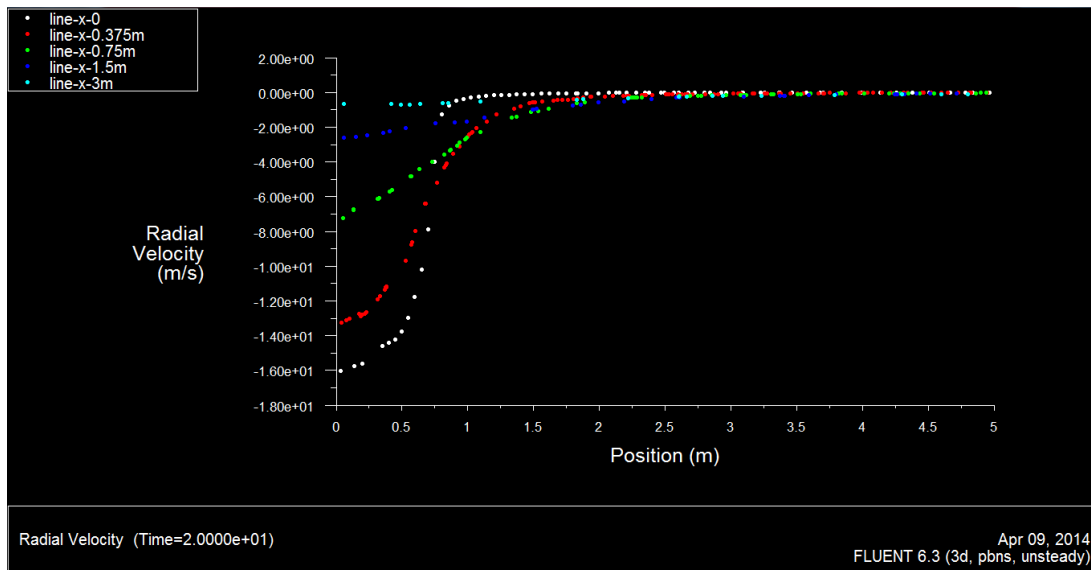


Figure 3.43.  $V_r$  vs.  $z$ , for different  $x$  values

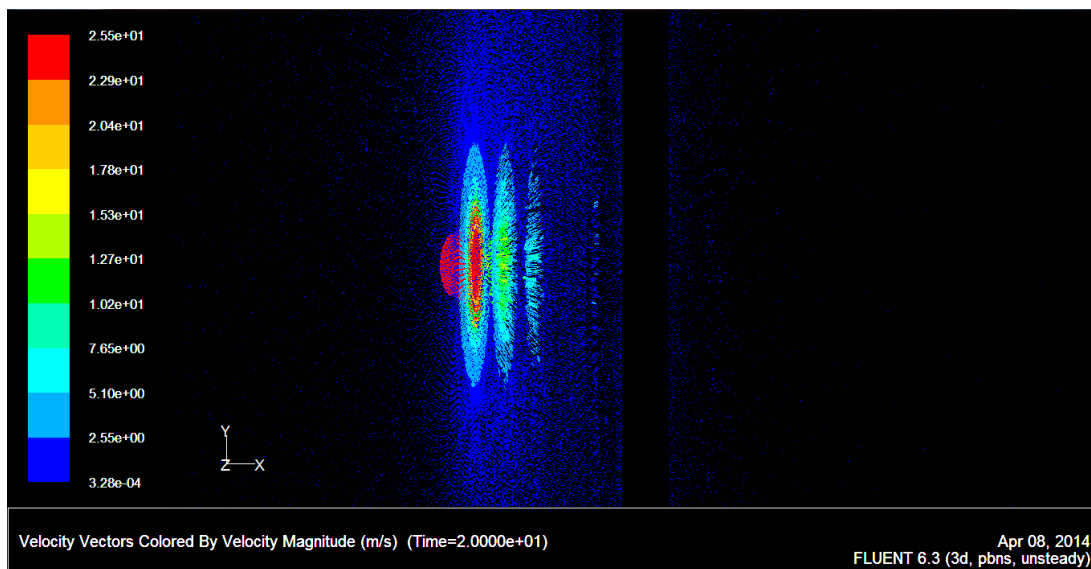


Figure 3.44. Velocity vectors for  $x=0, 0.375\text{m}, 0.75\text{m}, 1.5\text{m}, 3\text{m}$ .

Figure 3.44. shows velocity vectors on  $y$ - $z$  plane for different  $x$  values. It can be seen that after  $x=2\text{d}$  there is almost no motion towards the orifice.

### 3.5.3. Laminar Solution of the Enlarged Model

The laminar FLUENT solution for the enlarged model is presented in this section.

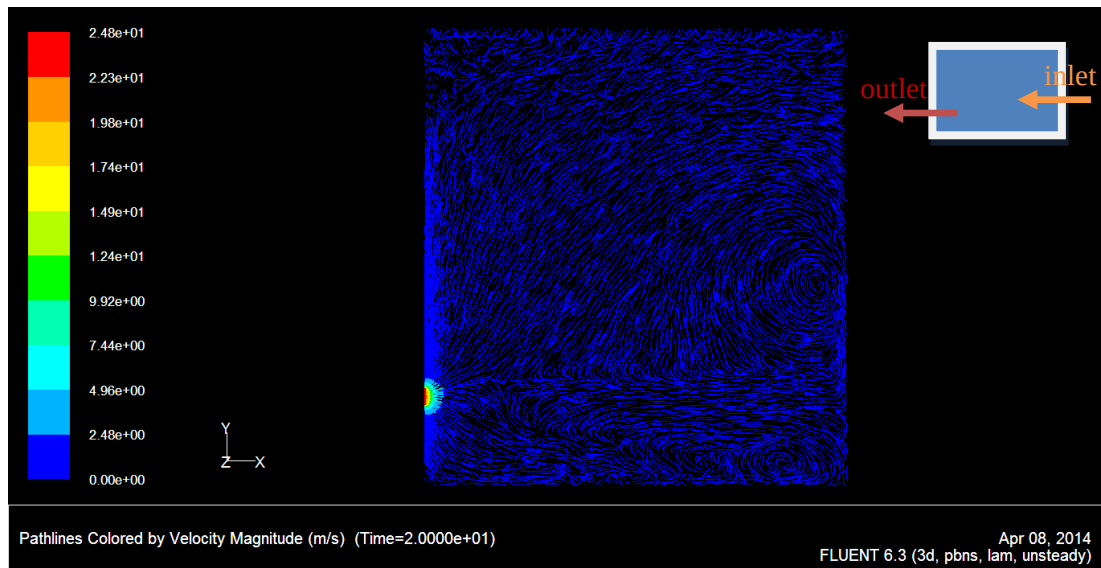


Figure 3.45. Pathlines by velocity magnitude

As it was the case with the inviscid solutions, a uniform pathline pattern can not be seen considering Figure 3.45. and Figure 3.46.

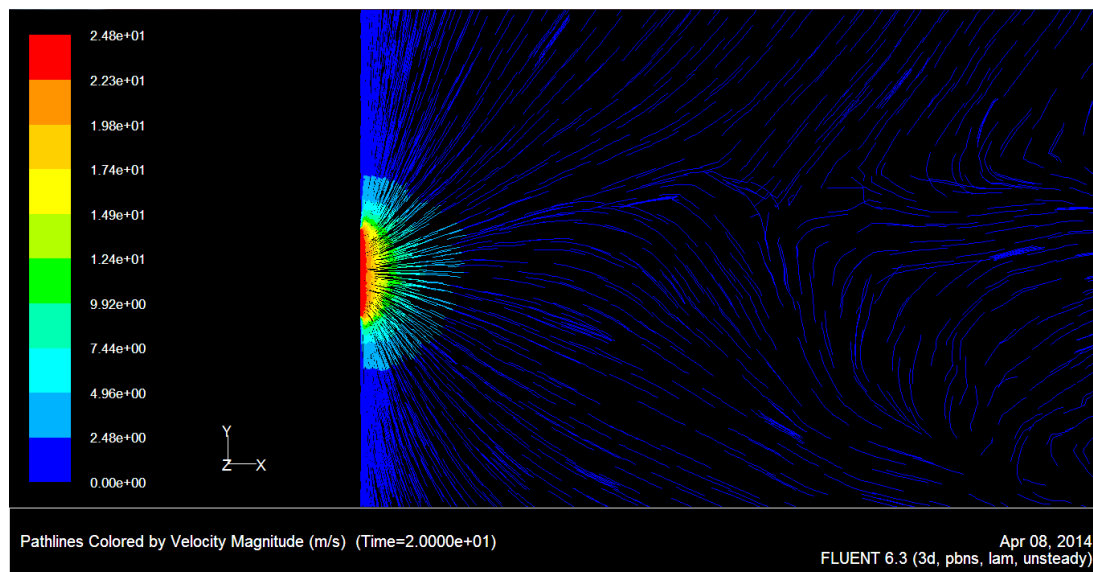


Figure 3.46. Close up of the orifice area for the pathlines by velocity magnitude

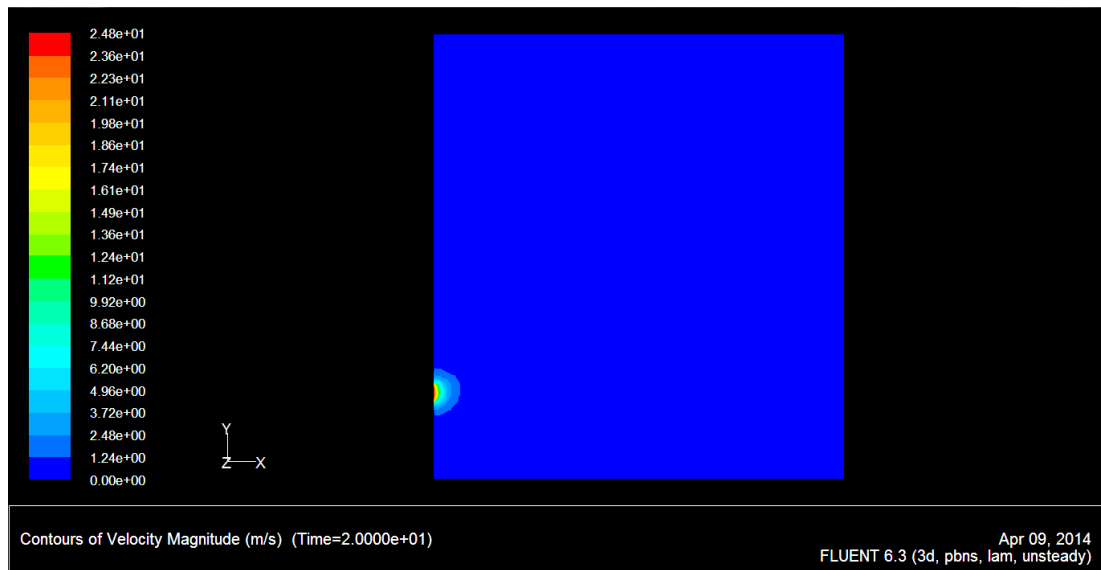


Figure 3.47. Contours of velocity magnitude

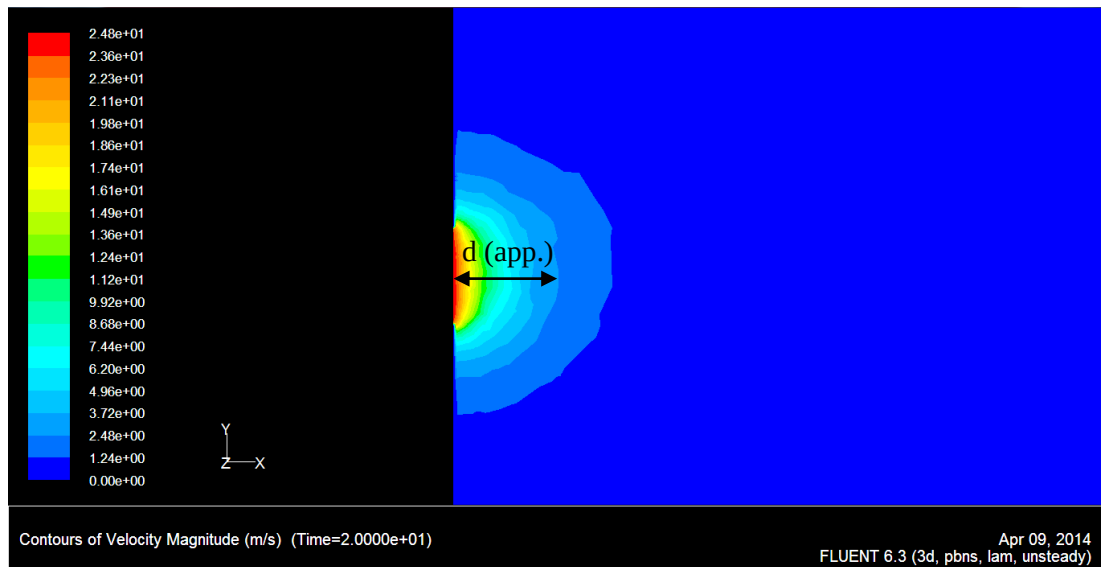


Figure 3.48. Close up of the orifice area for the contours of velocity magnitude

Figures 3.47. and 3.48. show velocity variation behind orifice on an x-y plane cutting the orifice and the box into two equal parts, i.e. the symmetry axis. As it was the case for the inviscid solutions, it is again seen that the velocity magnitude decreases to

one tenth of the maximum velocity value at a distance approximately  $d$  behind the orifice.



Figure 3.49. Contours of velocity magnitude



Figure 3.50. Close up of the orifice area for the contours of velocity magnitude



Figure 3.51. Velocity magnitude contours of the orifice area only

Considering Figure 3.49 to Figure 3.51. it can be concluded that the maximum velocity is 24.47 m/s and occurs at the center of the orifice and for a large part of the orifice very close velocity values are seen. Velocity magnitude is decreasing towards the perimeter and apart from the inviscid solutions becomes zero at the wall because the laminar solution takes the wall effect into account.

Again comparing with the Çobanoğlu model, it can be seen that the velocity magnitude rises to approximately ten times the velocity magnitude of the original model, and the trends are very similar except the effect of the walls.

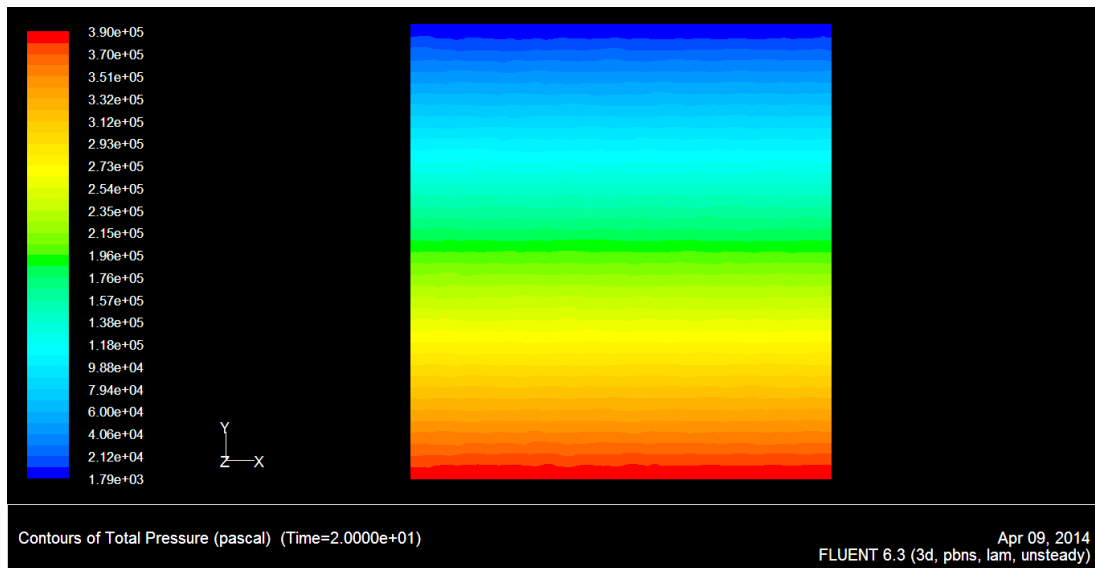


Figure 3.52. Pressure contours



Figure 3.53. Pressure contours of the orifice area

Pressure distribution is almost uniform along the orifice, with a little difference between the upper and lower part of the orifice as can be seen in Figure 3.53.

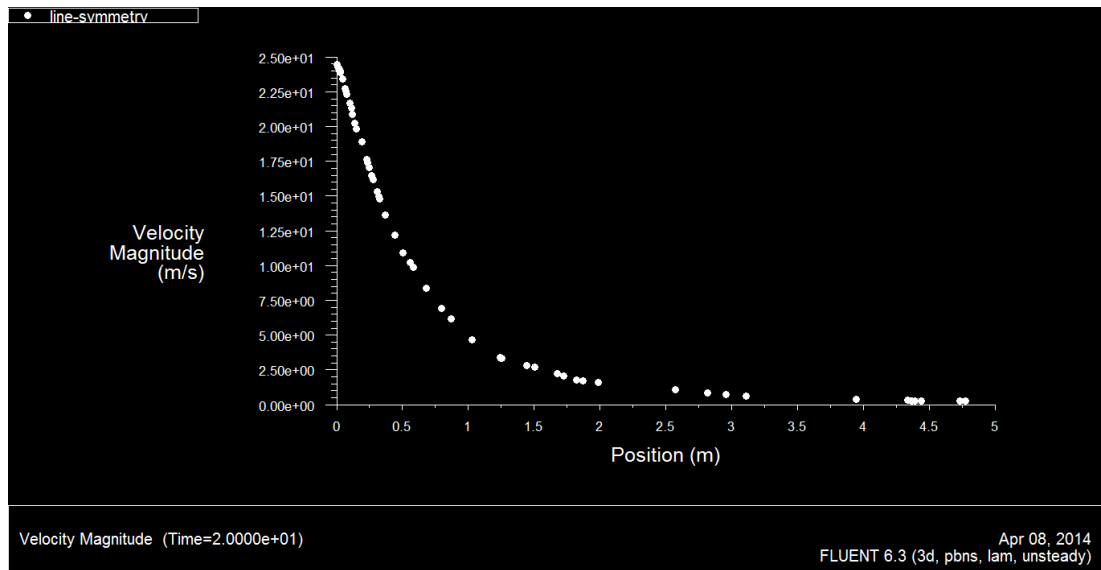


Figure 3.54. Velocity magnitude readings of the centerline velocity

Considering Figure 3.54. it is seen that after approximately  $x/d=1$ , velocity magnitude becomes less than one tenth of the maximum velocity, as it was also the case at the inviscid solutions.

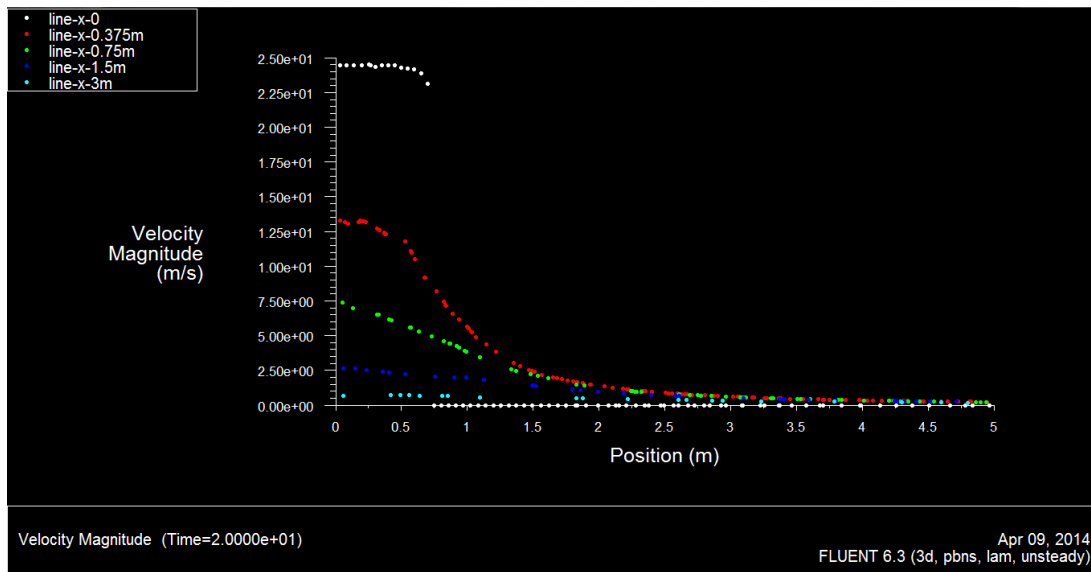


Figure 3.55. y-z plane velocity magnitude on z-axis for different x values

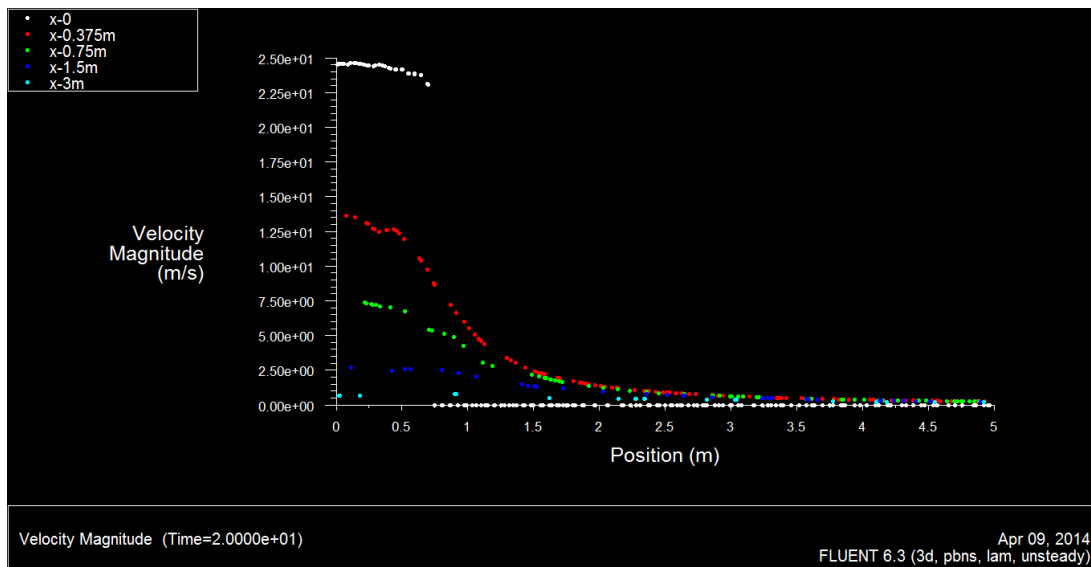


Figure 3.56. y-z plane velocity magnitude on y-axis for different x values

In Figure 3.55. and Figure 3.56. graphs are presented along the planes cutting the orifice and the box into two equal parts, i.e. the symmetry plane. It is seen that rate of decrease in velocity magnitude is higher close to the orifice region. The most

important difference of this solution is that at  $x=0$ , velocity in both planes becomes zero at the walls(0.75m from both sides), as it can be clearly seen in both figures.

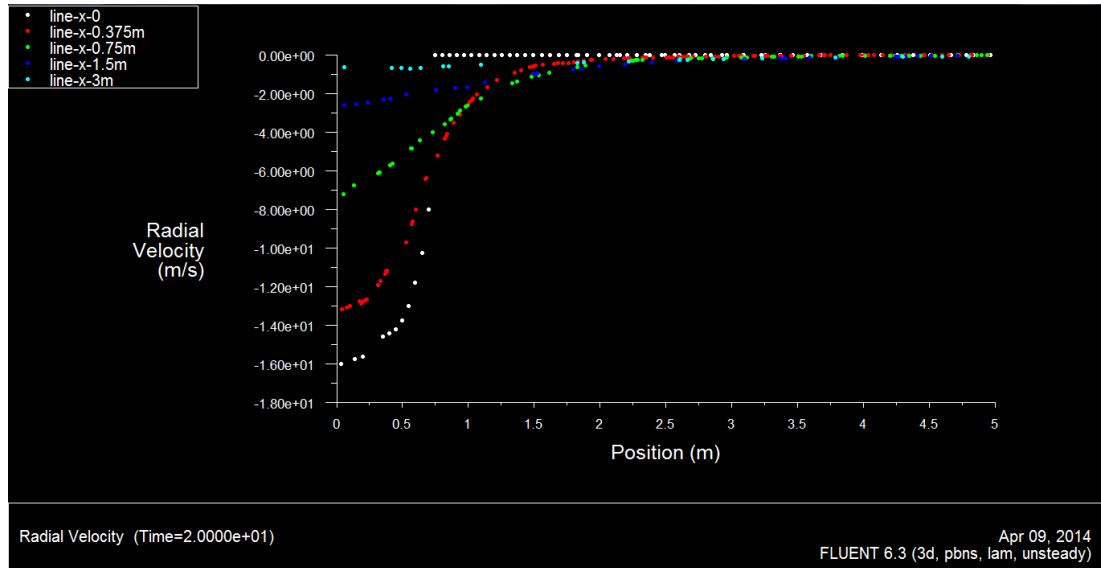


Figure 3.57.  $V_r$  vs.  $z$ , for different  $x$  values

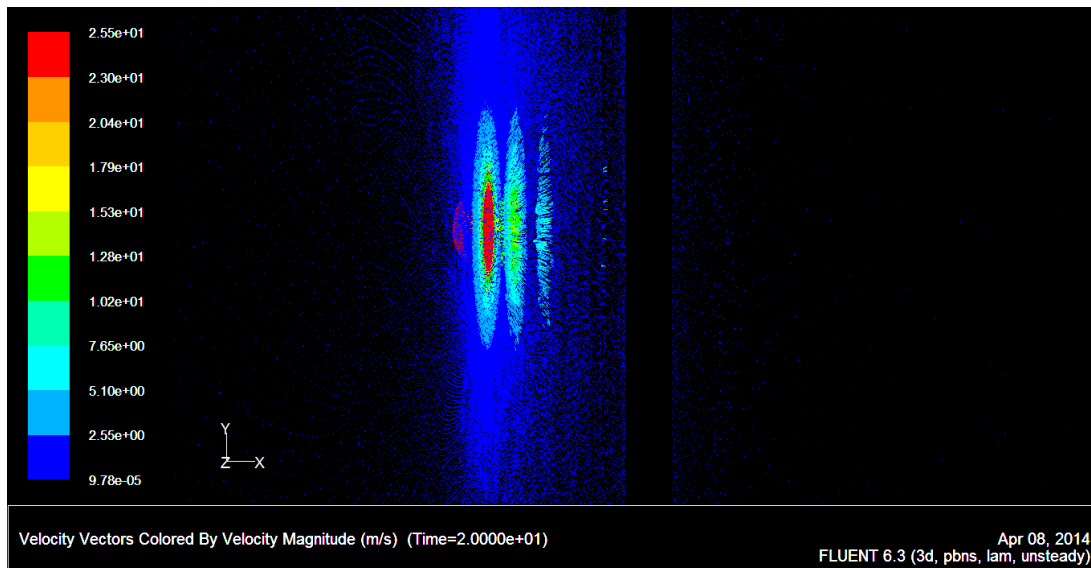


Figure 3.58. Velocity vectors for  $x=0, 0.375\text{m}, 0.75\text{m}, 1.5\text{m}, 3\text{m}$

Figure 3.58. shows velocity vectors on  $y$ - $z$  plane for different  $x$  values. It can be seen that after  $x=2d$  there is almost no motion towards the orifice, as the previous inviscid solutions.

### 3.5.4. $k - \epsilon$ Solution of the Enlarged Model

The  $k - \epsilon$  method of the FLUENT solution for the enlarged model is presented in this section.  $k - \epsilon$  method is applied with enhanced wall function option.

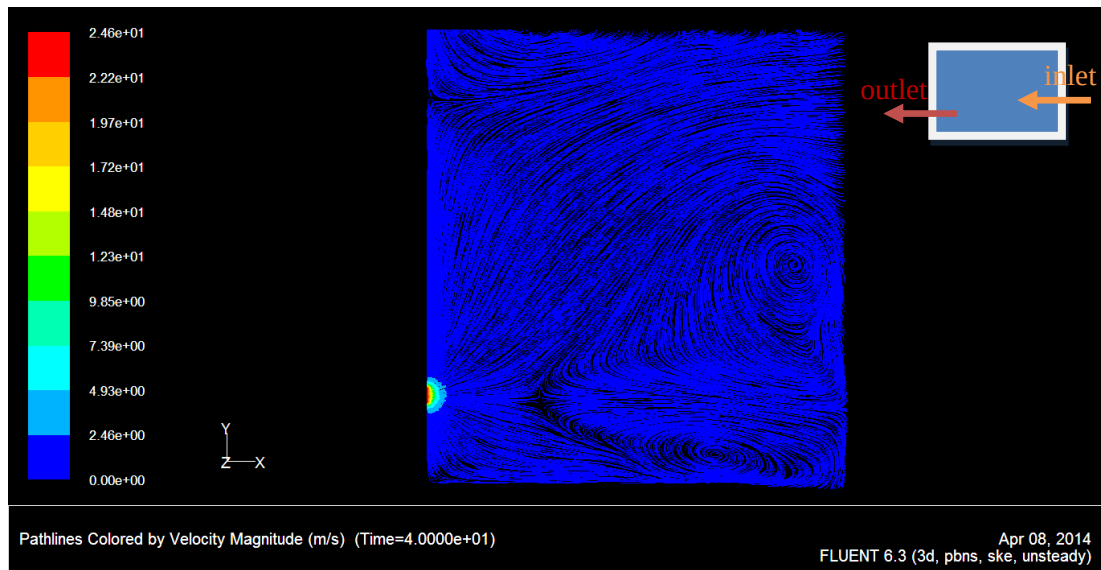


Figure 3.59. Pathlines by velocity magnitude

Considering Figure 3.59, a uniform pathline tendency can not be seen through the whole region and two vortices can be located, however in Figure 3.60, there is a similar pattern very close to the orifice region.

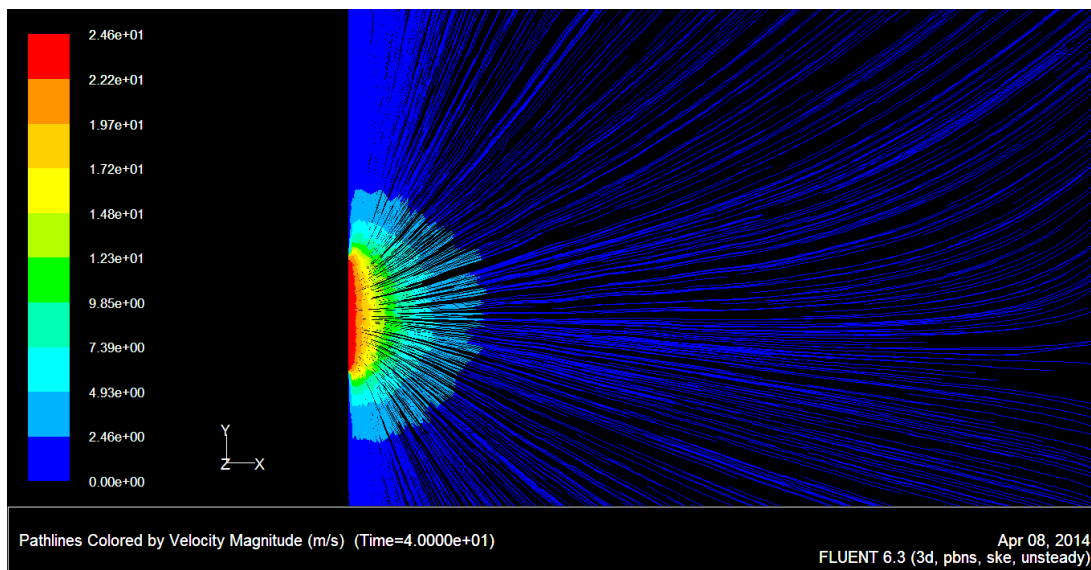


Figure 3.60. Close up of the orifice area for the pathlines by velocity magnitude

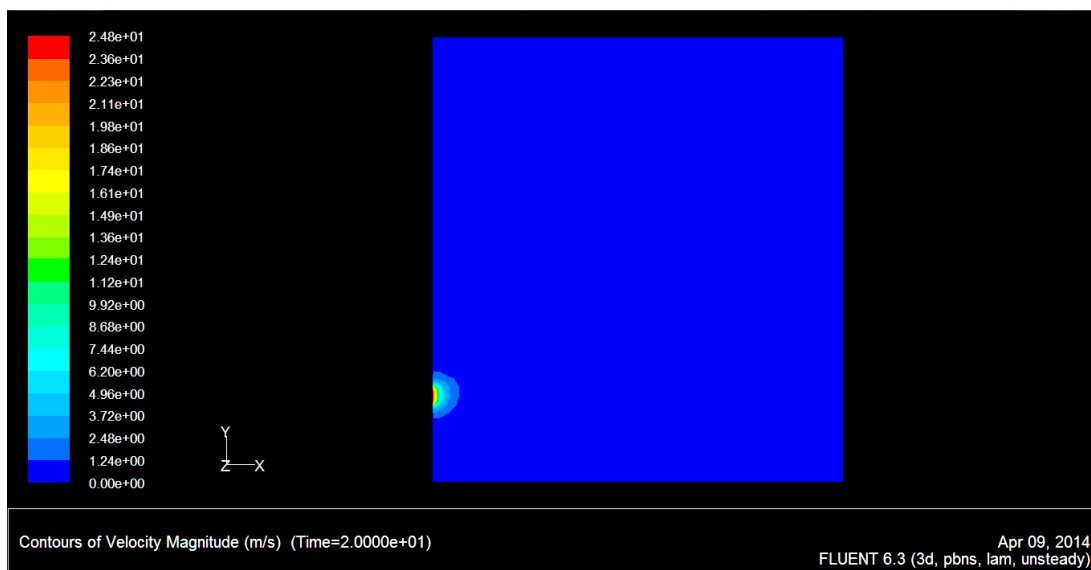


Figure 3.61. Contours of velocity magnitude

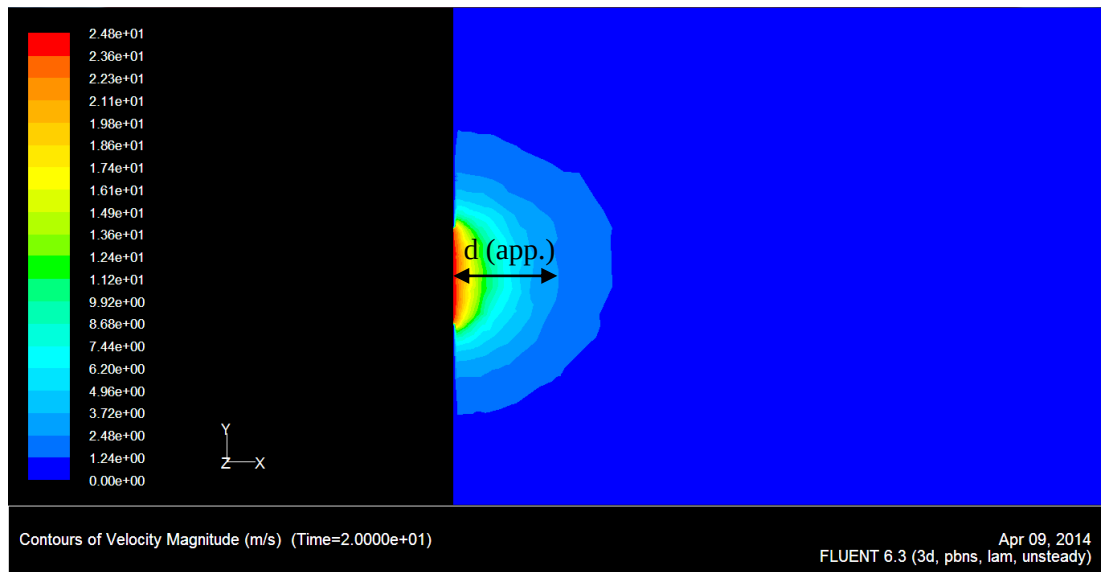


Figure 3.62. Close up of the orifice area for the contours of velocity magnitude

Figures 3.61. and 3.62. show velocity variation behind orifice on an x-y plane cutting the orifice and the box into two equal parts, i.e. the symmetry axis. As it was the case for the previous solutions, it is again seen that the velocity magnitude decreases to one tenth of the maximum velocity value at a distance approximately  $d$  behind the orifice.



Figure 3.63. Contours of velocity magnitude



Figure 3.64. Close up of the orifice area for the contours of velocity magnitude



Figure 3.65. Velocity magnitude contours of the orifice area only

Considering Figure 3.63 to Figure 3.65. it can be concluded that the maximum velocity is 24.34 m/s and occurs at the center of the orifice and for a large part of the orifice very close velocity values are seen. Velocity magnitude is decreasing towards the perimeter and apart from the inviscid solutions it becomes zero at the wall because the k- $\epsilon$  solution takes the wall effect into account.

Again comparing with the Çobanoğlu model, it can be seen that the velocity magnitude rises to approximately ten times the velocity magnitude of the original model, and the trends are very similar except the effect of the walls.

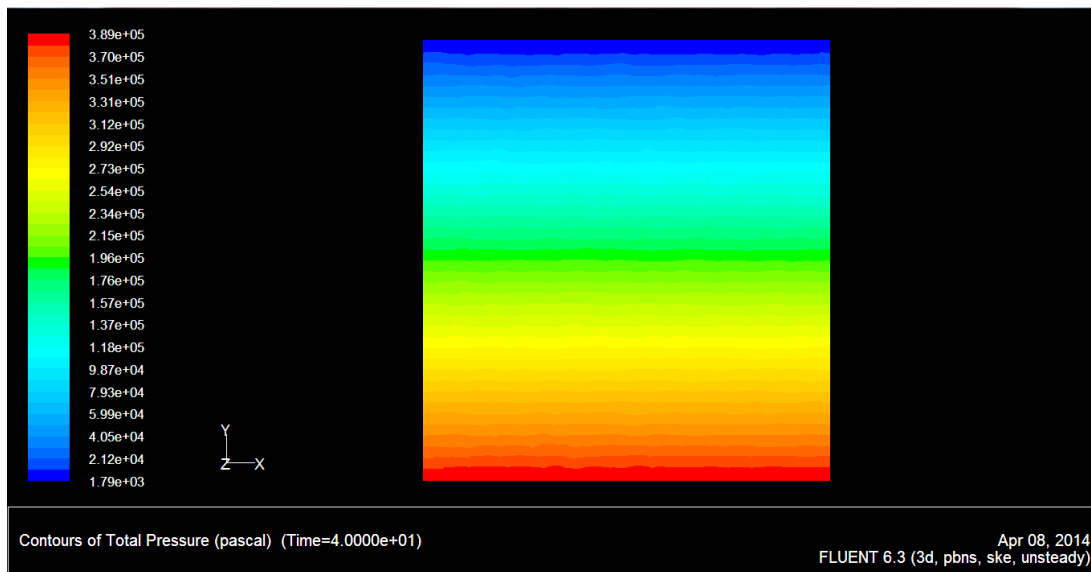


Figure 3.66. Pressure contours



Figure 3.67. Pressure contours of the orifice area

Pressure distribution is almost uniform along the orifice, with a little difference between the upper and lower part of the orifice as can be seen in Figure 3.67.

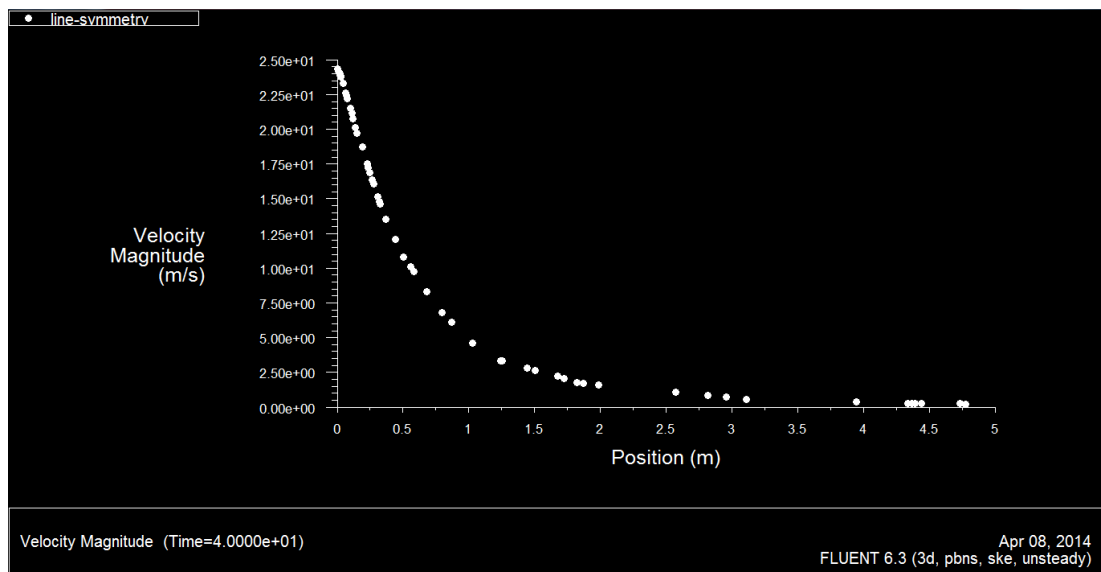


Figure 3.68. Velocity magnitude readings of the centerline velocity

Considering Figure 3.68. it is seen that after approximately  $x/d=1$ , velocity magnitude becomes less than one tenth of the maximum velocity, as it was also the case with the previous solutions.

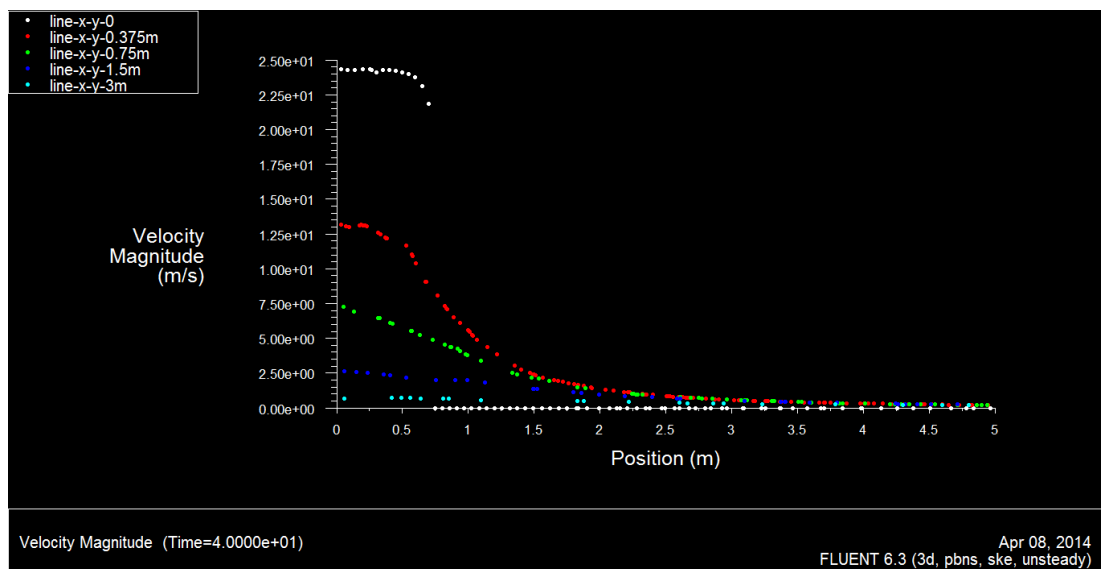


Figure 3.69. z plane velocity magnitude on z-axis for different x values

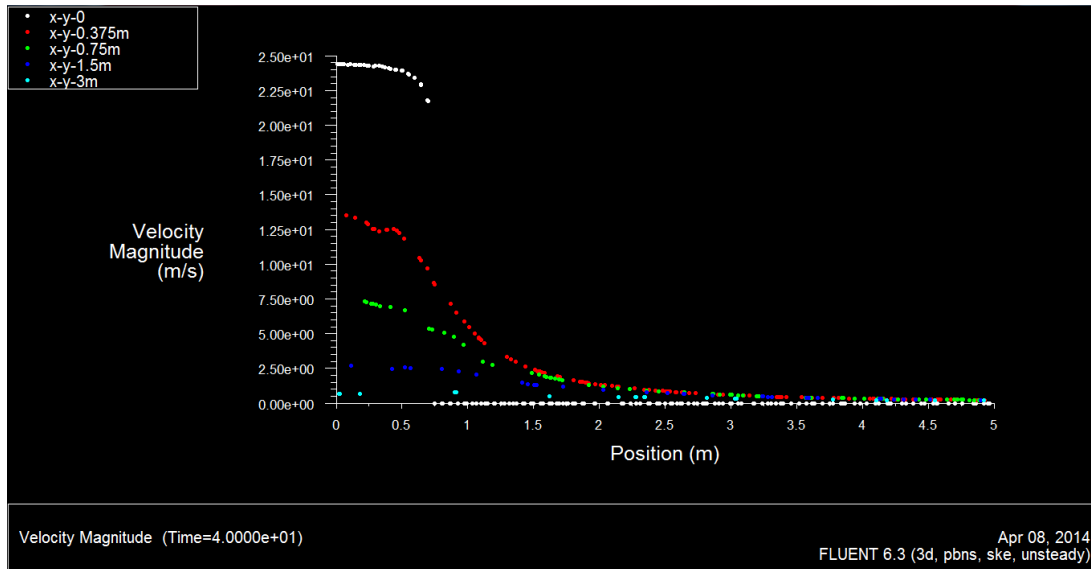


Figure 3.70. y-z plane velocity magnitude on y-axis for different x values

In Figure 3.69. and Figure 3.70. graphs are presented along the planes cutting the orifice and the box into two equal parts, i.e. the symmetry plane. It is seen that rate of decrease in velocity magnitude is higher close to the orifice region. Since k- $\epsilon$  solution takes into account the wall effects like the laminar solution, at  $x=0$ , velocity in both planes becomes zero at the walls (0.75m from both sides), as it can be clearly seen in both figures.

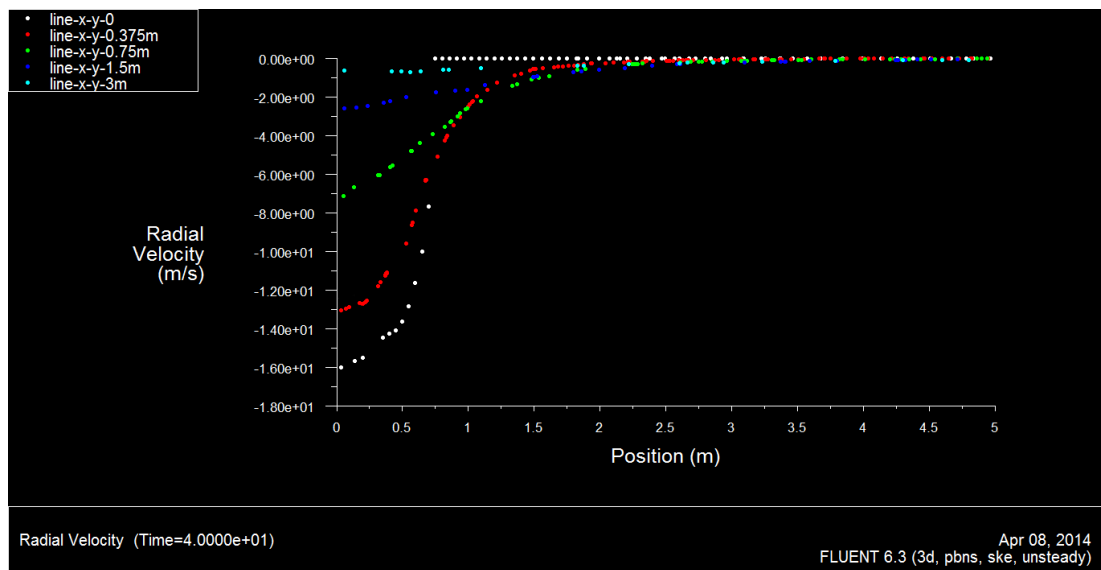


Figure 3.71.  $V_r$  vs.  $z$ , for different  $x$  values

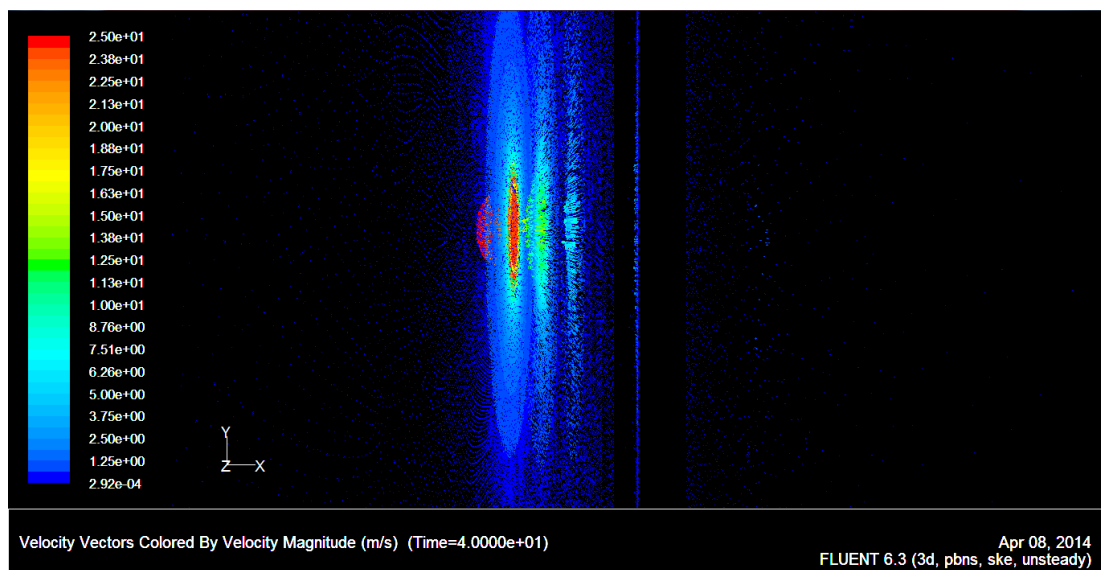


Figure 3.72. Velocity vectors for  $x=0, 0.375\text{m}, 0.75\text{m}, 1.5\text{m}, 3\text{m}$

Figure 3.72. shows velocity vectors on  $y$ - $z$  plane for different  $x$  values. It can be seen that after  $x=2d$  there is almost no motion towards the orifice, similar to the previous solutions.

### 3.5.5. VOF Solution of the Enlarged Model

The Volume of Fluid method of the FLUENT solution for the enlarged model is presented in this section.

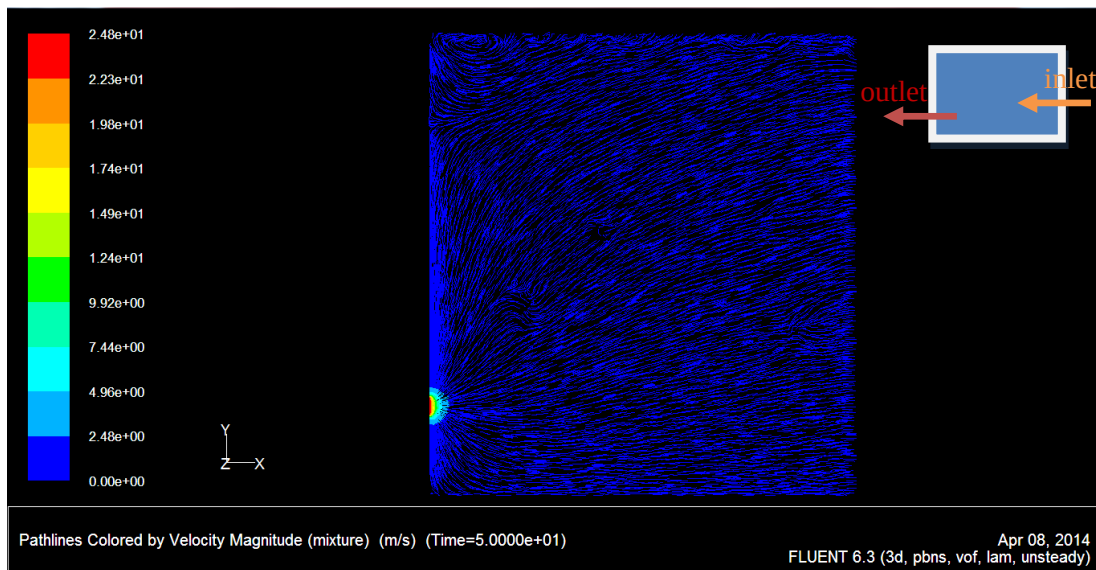


Figure 3.73. Pathlines by velocity magnitude

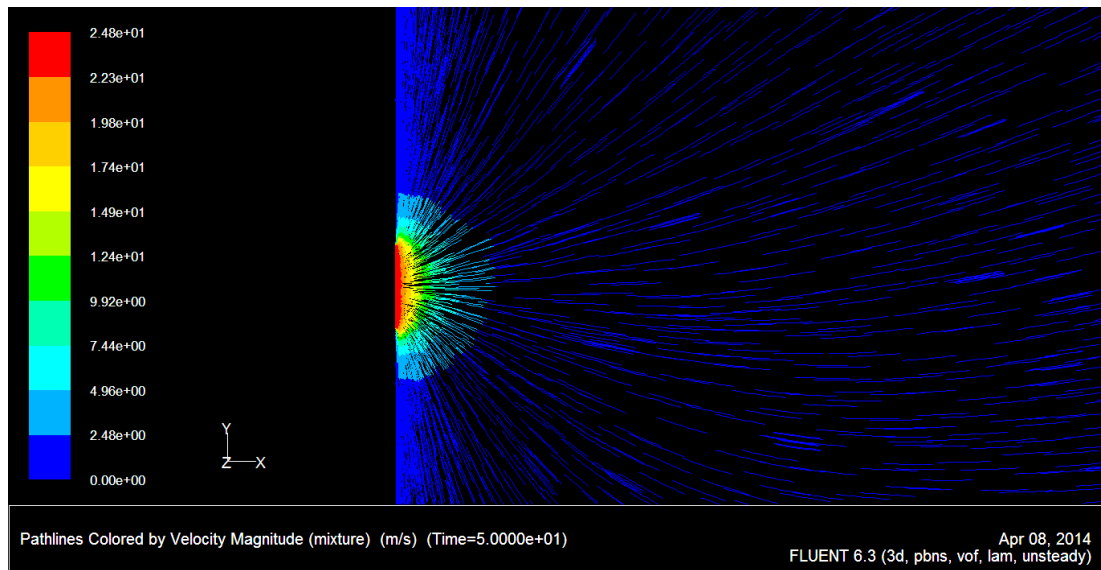


Figure 3.74. Close up of the orifice area for the pathlines by velocity magnitude

Considering Figure 3.73 and Figure 3.74. a uniform pathline pattern can be seen apart from the other solutions.

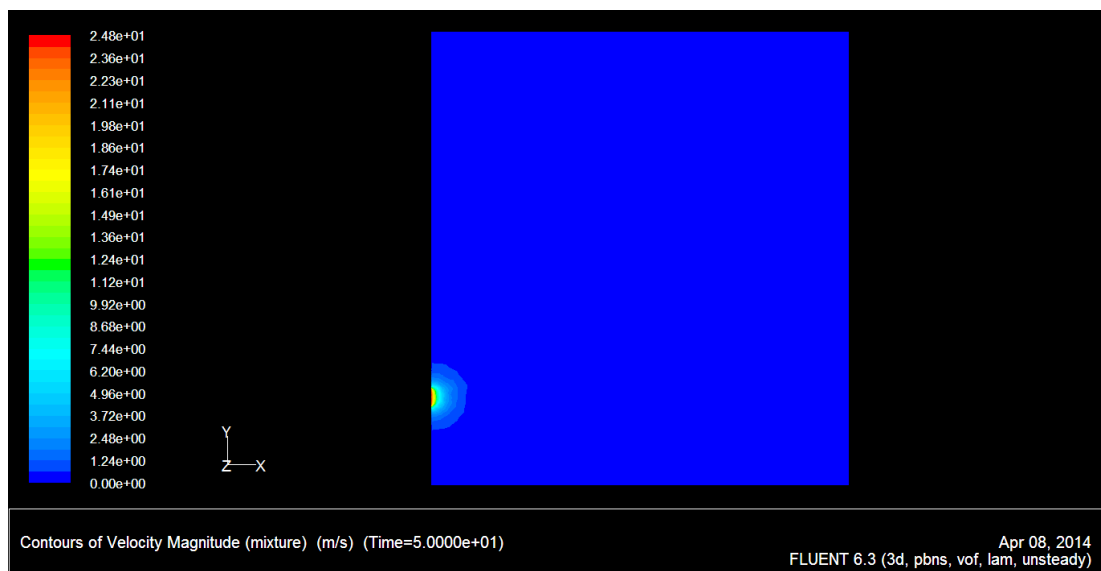


Figure 3.75. Contours of velocity magnitude

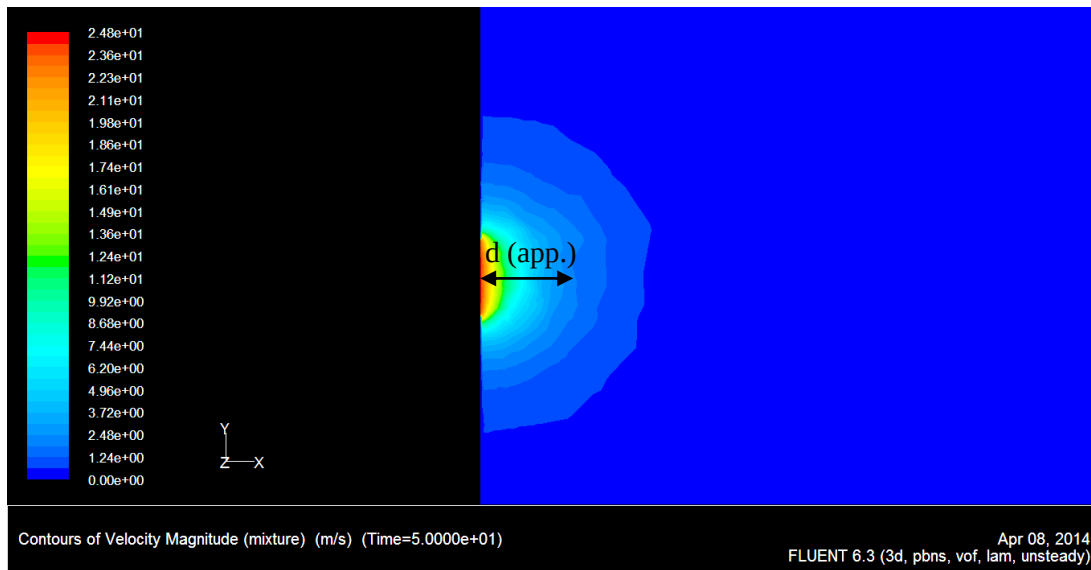


Figure 3.76. Close up of the orifice area for the contours of velocity magnitude

Figures 3.75. and 3.76. show velocity variation behind orifice on an x-y plane cutting the orifice and the box into two equal parts, i.e. the symmetry axis. As it was the case for the previous solutions, it is again seen that the velocity magnitude decreases to one tenth of the maximum velocity value at a distance approximately  $d$  behind the orifice.



Figure 3.77. Contours of velocity magnitude



Figure 3.78. Close up of the orifice area for the contours of velocity magnitude

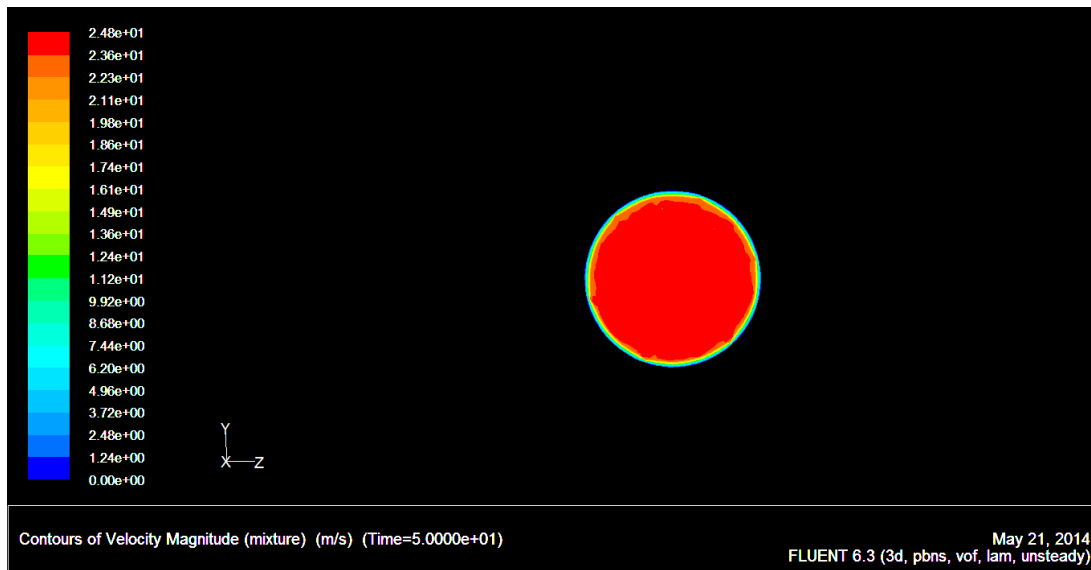


Figure 3.79. Velocity magnitude contours of the orifice area only

Considering Figure 3.77 to Figure 3.79. it can be concluded that the maximum velocity is 24.46 m/s and occurs at the center of the orifice and for a large part of the orifice very close velocity values are seen. Velocity magnitude is decreasing towards the perimeter and apart from the inviscid solutions it becomes zero at the wall because the vof solution takes the wall effect into account.

Again comparing with the Çobanoğlu model, it can be seen that the velocity magnitude rises to approximately ten times the velocity magnitude of the original model, and the trends are very similar except the effect of the walls.

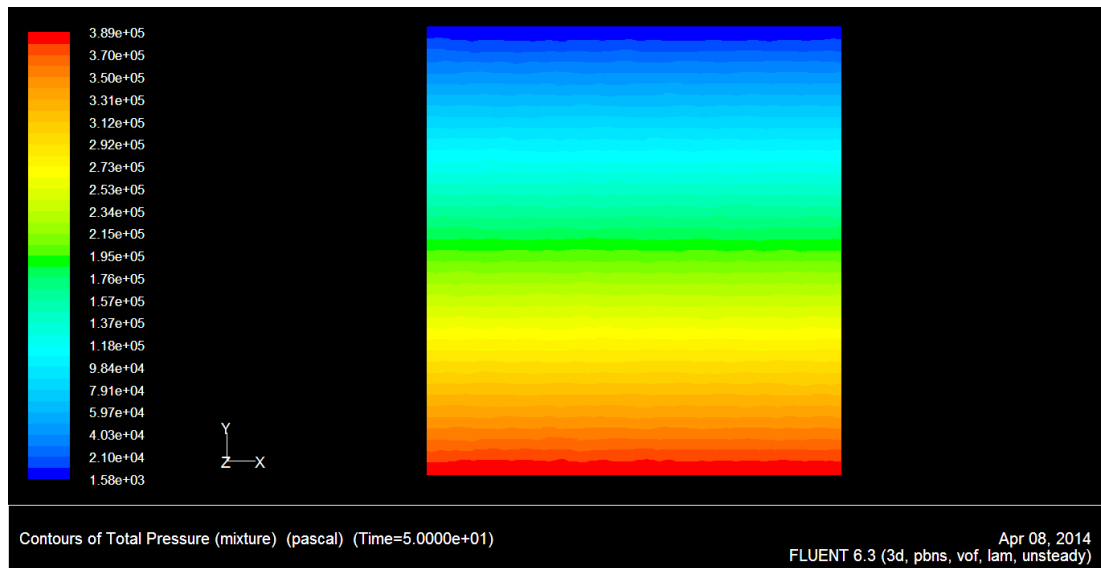


Figure 3.80. Pressure contours



Figure 3.81. Pressure contours of the orifice area only

Pressure distribution is almost uniform along the orifice, with a little difference between the upper and lower part of the orifice as can be seen in Figure 3.81.

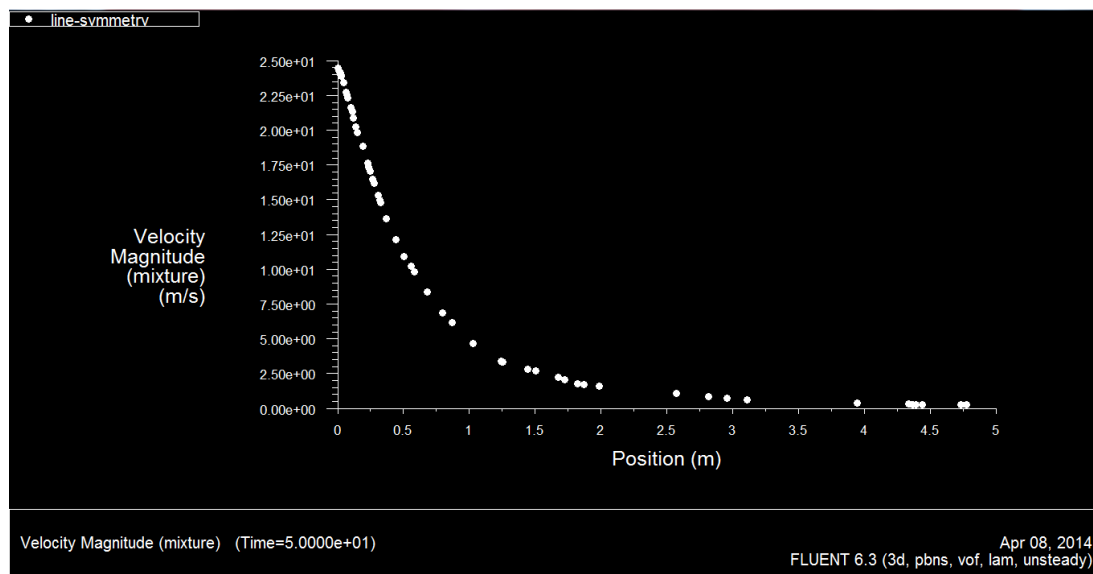


Figure 3.82. Velocity magnitude readings of the centerline velocity

Considering Figure 3.82. it is seen that after approximately  $x/d=1$ , velocity magnitude becomes less than one tenth of the maximum velocity, as it was also the case with the previous solutions.

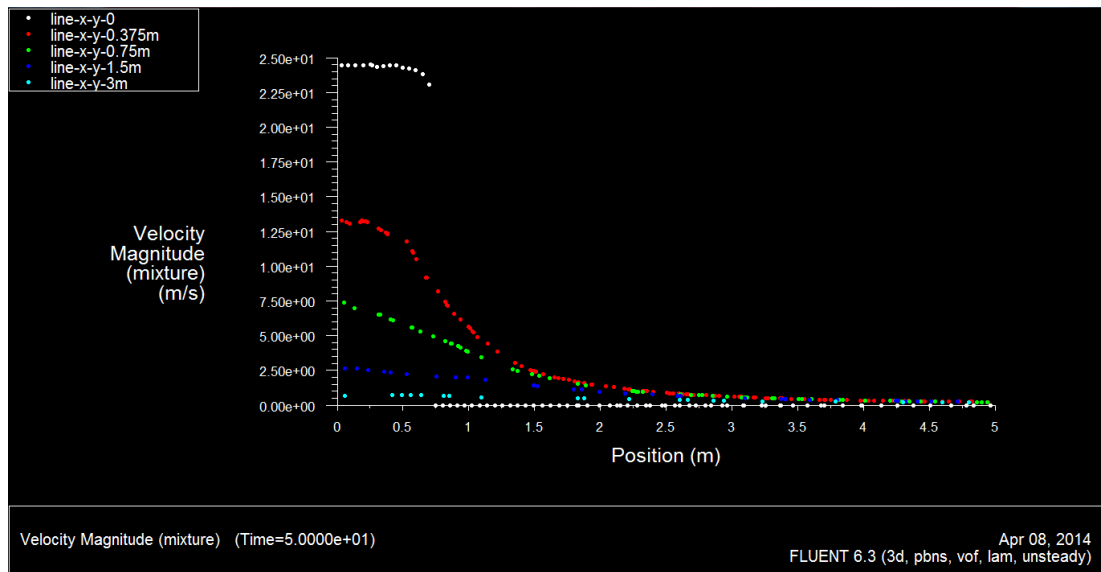


Figure 3.83. y-z plane velocity magnitude on z-axis for different x values

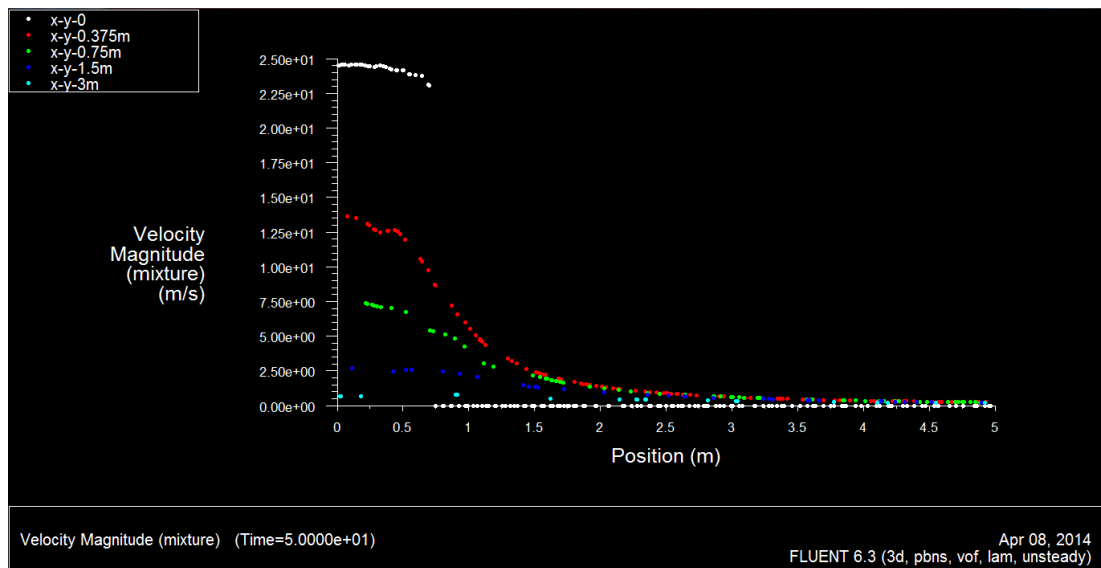


Figure 3.84. y-z plane velocity magnitude on y-axis for different x values

In Figure 3.83. and Figure 3.84. graphs are presented along the planes cutting the orifice and the box into two equal parts, i.e. the symmetry plane. It is seen that rate of decrease in velocity magnitude is higher close to the orifice region. Since vof

solution takes into account the wall effects like the laminar and k- $\epsilon$  solutions, at  $x=0$ , velocity in both planes becomes zero at the walls (0.75m from both sides), as it can be clearly seen in both figures.

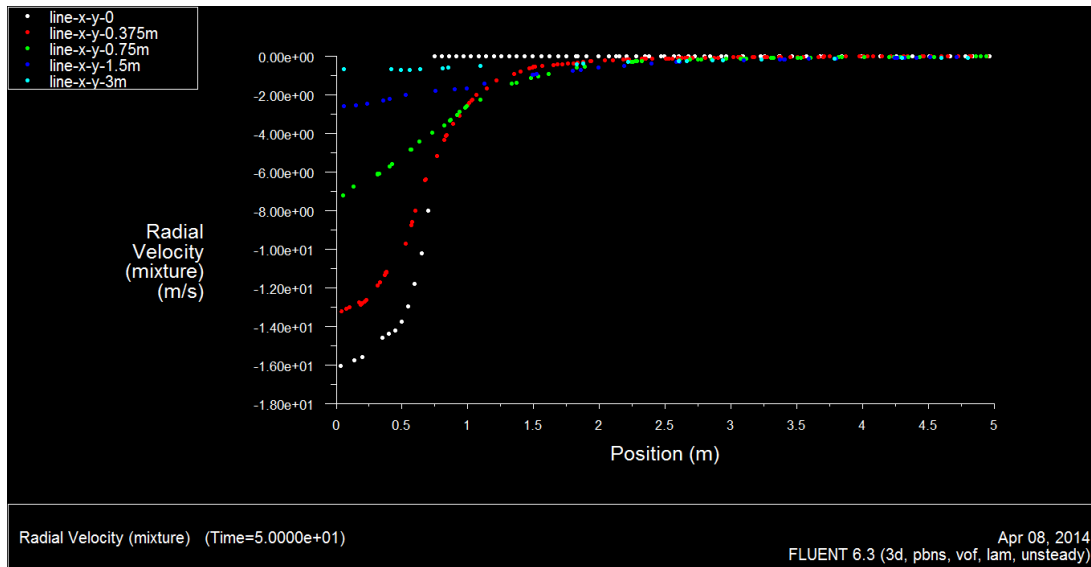


Figure 3.85.  $V_r$  vs.  $z$ , for different  $x$  values

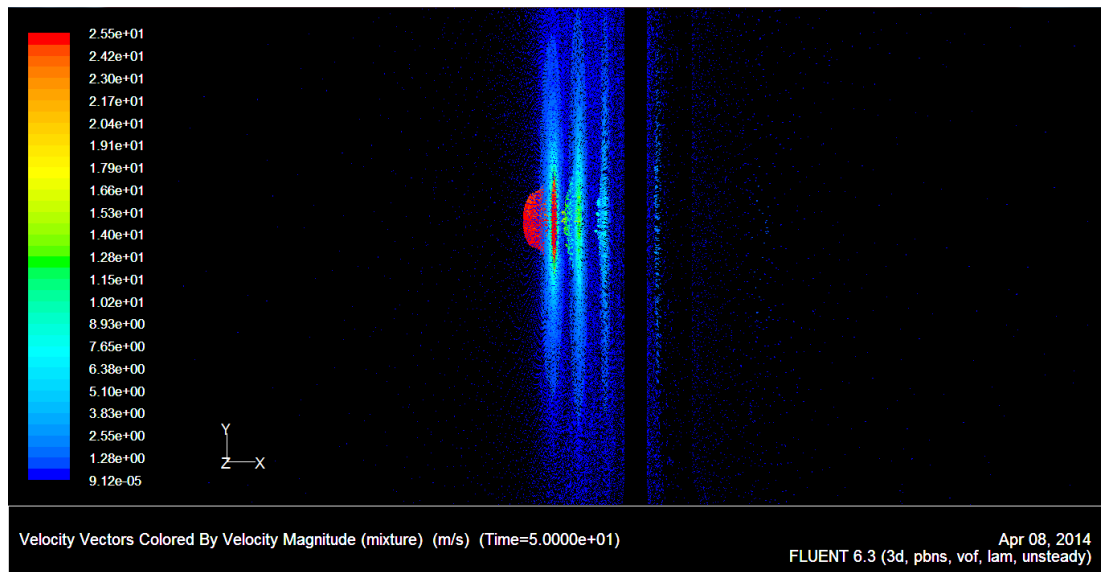


Figure 3.86. Velocity vectors for  $x=0, 0.375\text{m}, 0.75\text{m}, 1.5\text{m}, 3\text{m}$

Figure 3.86. shows velocity vectors on y-z plane for different x values. It can be seen that after  $x=2d$  there is almost no motion towards the orifice, similar to the previous solutions.



## CHAPTER 4

### COMPARISON OF OBTAINED RESULTS

#### 4.1. Comparison of Obtained Results

Results obtained from the solution procedures reported in Chapters 2 and 3 will be compared in this section.

Using Torricelli Equation as described in Chapter 3, a discharge value of 0.4445 lt/s is obtained.

Furthermore, when the discharge value obtained from the FLUENT calculation and the discharge value obtained from Torricelli Equation are compared, it is found out that the discharge coefficient,  $C_d$ , equals 0.78. When compared, it is seen that the obtained  $C_d$  value agrees with the values presented in Yıldırım's study(2010), which is shown at Figure 4.1.

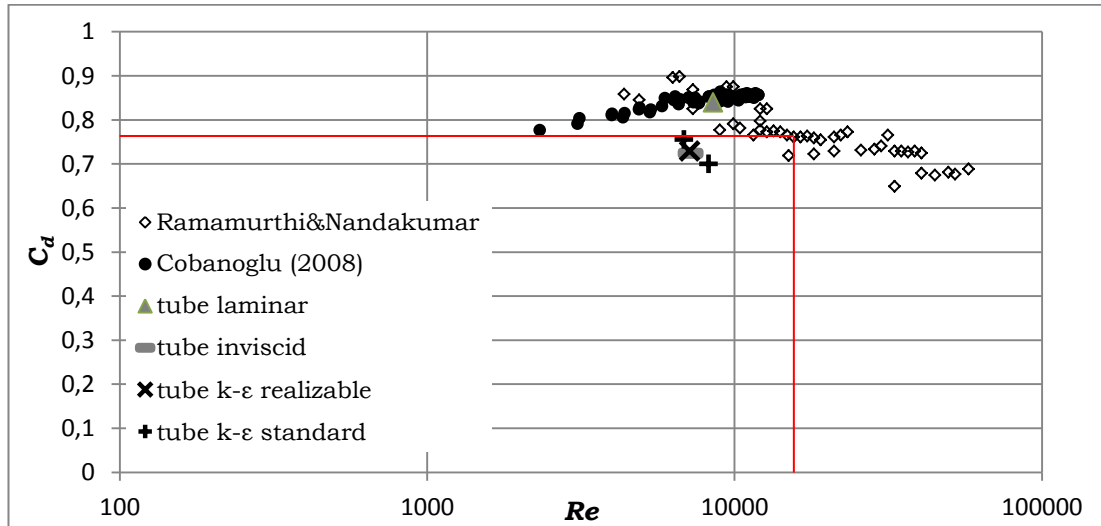


Figure 4.1.  $C_d$  vs  $Re$  from Yıldırım's study, Scrutinization of Flow Characteristics Through Orifices, 2010

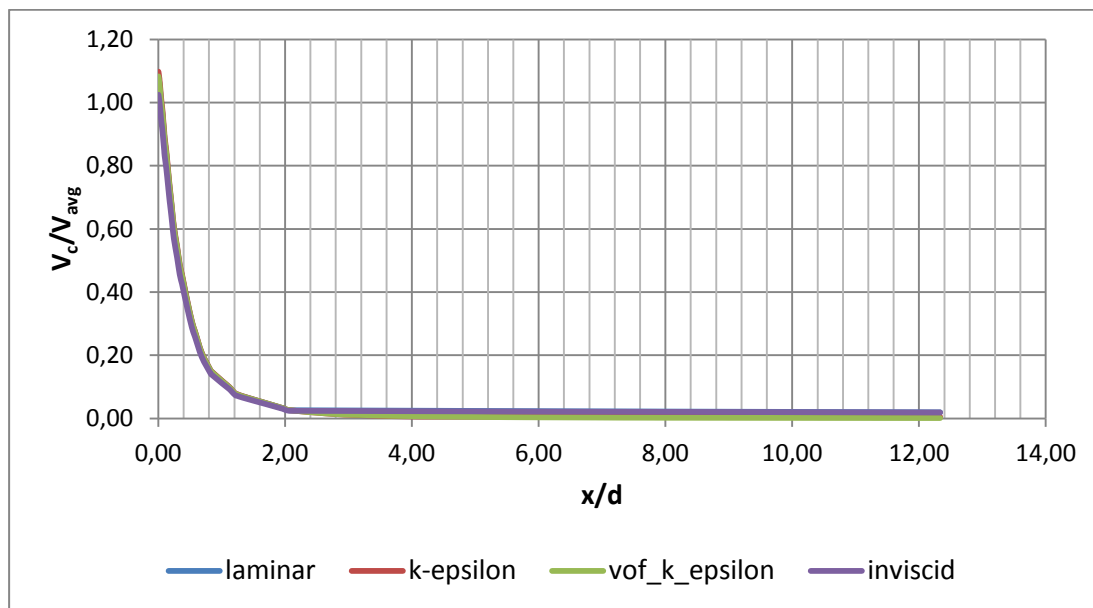


Figure 4.2. Centerline velocity values for different FLUENT solutions

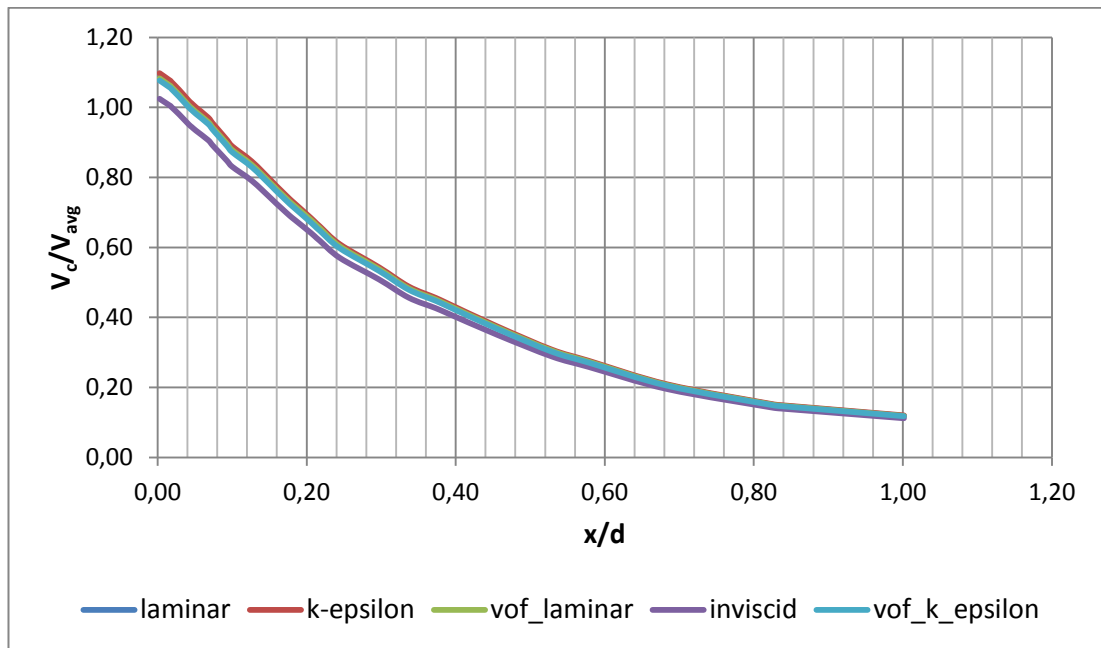


Figure 4.3. Closer view of the orifice region for centerline velocity values for different FLUENT solutions

In Figure 4.3. it can be seen that the velocity values for inviscid, laminar, k- $\epsilon$  and vof FLUENT solutions are very similar until an  $x/d$  value of 0.8. Between  $x/d$  values 0.8 and 0.4, viscous effects start to develop and after 0.4 the difference between potential solution and other solutions becomes larger. It is important to note that there is very small difference between laminar, k- $\epsilon$  and vof solutions.

Although, there are several products of solution presented in Chapters 2 and 3, the only result compared will be the velocity along the centerline. Since, in Chapter 3, the performance of the velocity potential was found unsatisfactory in the vicinity of the orifice, it is important to demonstrate the performance of CFD near the outlet. Fluent solution is deviating from 3D potential solutions within the distance of  $x/d$  smaller than 0.2.

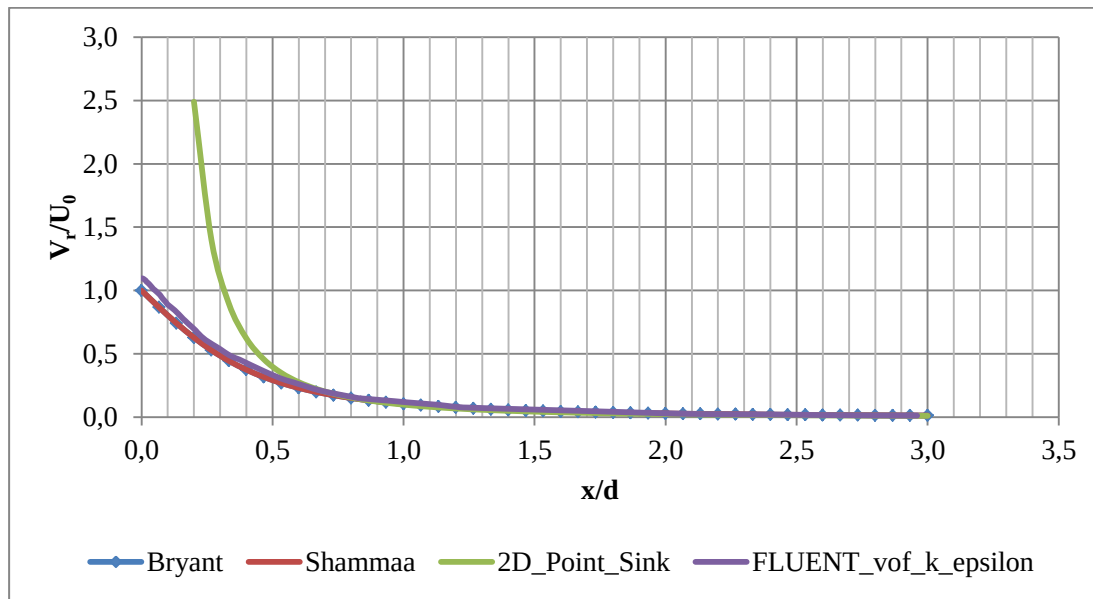


Figure 4.4.  $V_r/U_0$  vs.  $x/d$  for different solutions

Figure 4.4 presents the centerline velocity for different solutions. It could be clearly seen from Figure 4.4. and that the 2D point sink solution agrees well with the other solutions up to nearly  $x \approx 0.8d$ . However after this point, the velocity obtained by the point sink solution rises to very high amounts, which makes no sense in hydromechanical point of view. This is due to the solution takes into account only the magnitude of total discharge and the distance related to the obtained velocity. However, up to a distance  $x = 0.8d$  upstream of the orifice, 2D point sink solution may be used because it is a very practical and simple method.

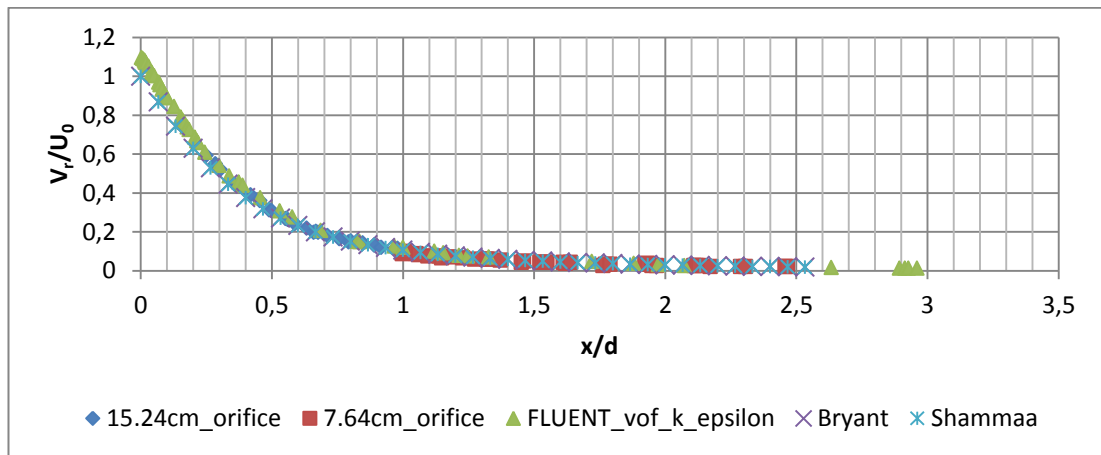


Figure 4.5.  $V_r/U_0$  vs.  $x/d$  for different solutions

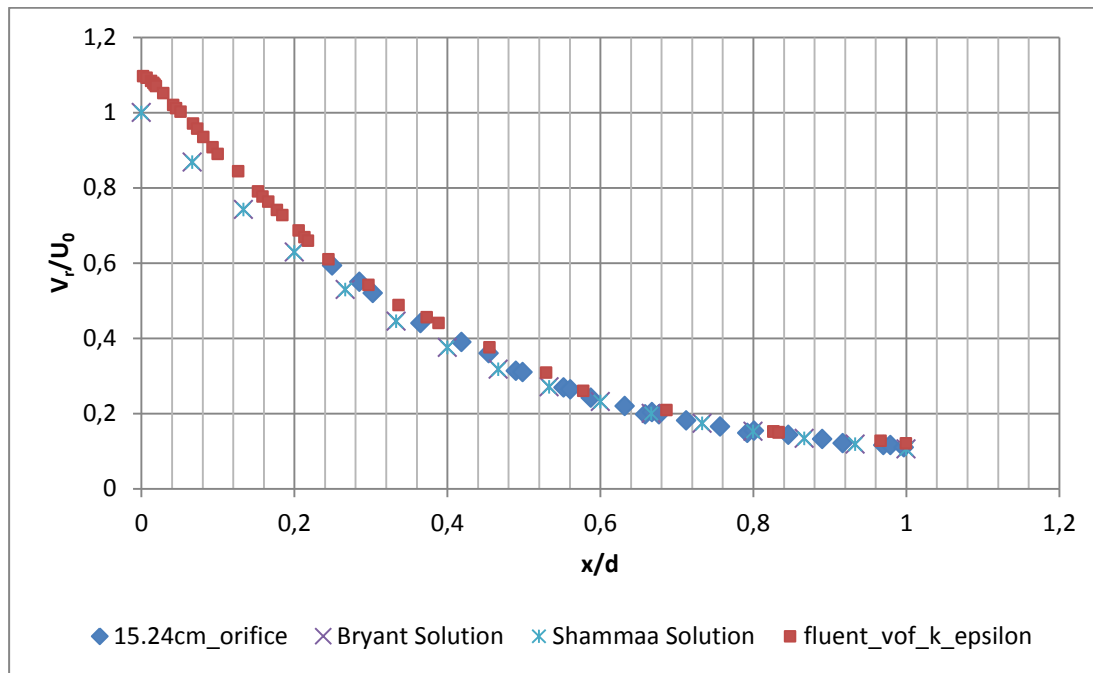


Figure 4.6. Closer view of the orifice area for  $V_r/U_0$  vs.  $x/d$  for different solutions

Figure 4.5. and Figure 4.6. present the comparison of FLUENT, Bryant and Shammaa solutions, as well as the two experiments which were presented at the study of Bryant et.al. Shammaa and Bryant solutions give almost the same results. From inlet up to a distance of  $x=0.8d$ , all the solutions provided similar results, however after this point FLUENT solution deviates from Shammaa and Bryant solutions. It is seen that experiment data agrees with all the solutions up to  $x=0.8d$ .

## CHAPTER 5

### CONCLUSION

#### 5.1. Conclusion

2D potential flow solution can be used up to some extent. However, for cases which need investigation close to the orifice, the 2D potential flow solution fails to provide accurate results.

Bryant and Shammaa solutions can be used far from the orifice, however it should be noted that although these solutions provide accurate centerline velocity results, at some point they fail to represent correct potential values.

Although it also fails very close the orifice, Bryant solution represents the solution closer to the real case. However, since the analytical methods ignore gravitational acceleration, they fail to provide real velocities at the vertical direction.

Solutions obtained by the analytical methods do not represent the real case if the discharge from the orifice is not known. There is only one way to predict the discharge from the orifice, which is making use of the Torricelli's law;  $v = \sqrt{2gh}$ . However, discharge obtained by this law cannot be considered as real and it differs from the FLUENT solution.

FLUENT solution provides the best solution since every factor can be taken into consideration. However, it is a time consuming method to apply. Therefore, type of the problem is very important to decide which method is to be used. In case the problem requires investigation of vertical velocities, the most appropriate solution will be the solution provided by FLUENT.

## REFERENCES

- Shammaa Y., Zhu D.Z., Rajaratnam N., (2005). Flow Upstream of Orifices and Sluice Gates, *Journal of Hydraulic Engineering*, 2005.
- Bryant D. B., Khan A. A., Aziz N. M., (2008). Investigation of Flow Upstream of Orifices, *Journal of Hydraulic Engineering*, 2008.
- Yıldırım, T., (2010). Scrutinization of Flow Characteristics Through Orifices, M.S. Thesis, Department of Civil Engineering, Middle East Technical University, Ankara.
- Kemalli, O., (2009). Measurements of Velocity Profiles by Using Particle Image Velocimeter, M.S. Thesis, Department of Civil Engineering, Middle East Technical University, Ankara.
- Göbelez, Ö., (2008). Experimental Analysis of the Flow Through a Bottom Outlet on the Threshold of Motion of Particles, M.S. Thesis, Thesis, Department of Civil Engineering, Middle East Technical University, Ankara.
- Çobanoğlu, İ., (2008). Experimental Study of Single and Multiple Outlets Behavior Under Constant Head, M.S. Thesis, Thesis, Department of Civil Engineering, Middle East Technical University, Ankara.
- Perry G., Anayiotos A. S., Green D., (1996). Accuracy of Color Doppler Velocity in the Flow Field Proximal to a Regurgitant Orifice: Implications for Color Doppler Quantitation of Valvular Incompetence, *Ultrasound in Medicine and Biology*, Volume 22, Number 5.
- Rodriguez L., Anconina J., Flachskampf F. A., Weyman A. E., Levine R. A., Thomas J. D., (1992). Impact of Finite Orifice Size on Proximal Flow Convergence. Implications for Doppler Quantification of Valvular Regurgitation, *Circular Research, Journal of the American Heart Association*.
- Anayiotos A. S., Perry G., Myers J. G., Green D. W., Fan P. H., Nanda N. C., (1995). A Numerical and Experimental Investigation of the Flow Acceleration Region Proximal to an Orifice, *Ultrasound in Medicine and Biology*, Volume 21, Number 4.

- Introduction to Fluid Mechanics, Sixth Edition, Fox R.W., McDonald A.T., Pritchard P.J., Wiley & Sons, 2004.
- Introduction to Fluid Mechanics, Fay J.A., MIT Press, 1994.
- Fluid Mechanics, An Introduction to the Theory of Fluid Flows, Durst F., Springer, 2008.
- Fluid Mechanics, Sixth Edition, Douglas J.F., Gasiorek J. M., Swaffield J.A., Jack L.B., Pearson, 2011.
- Statistical Theory and Modeling for Turbulent Flows, Second Edition, P.A. Durbin, B.A. Petterson Reif, 2010.
- CFD-Online, <http://www.cfd-online.com>, last visited on May 2014)
- ANSYS FLUENT 6.3 User's Guide
- ANSYS FLUENT 6.3 Theory Guide
- Modelling of Flow in Rectangular Sedimentation Tanks by an Explicit Third-Order Upwinding Technique, Gerges and McCorquodale, 1997.
- Jones W.P., Launder B.E., (1971). The Prediction of Laminarization With a Two-Equation Model of Turbulence, Int. J. of Heat Mass Transfer, Vol. 15, pp. 301-314.
- Launder, B. E. and Sharma, B. I., (1974). Application of the Energy-Dissipation Model of Turbulence to the Calculation of Flow Near a Spinning Disc, Letters in Heat and Mass Transfer, Vol. 1, No. 2, pp. 131-138.

## APPENDIX A

### MATLAB Codes

#### A.1. 2D Potential Flow Sink Solution – Radial Velocity

```
clear;
clc;

r=input('input r (distance from the sink) in meters\n');
Q=input('input Q (volumetric flowrate) in m3/s\n');
sample=input('input sampling number\n');
for i=1:sample
    x(i)=i*(r/sample);
    Ur(i)=Q/(2*pi*x(i));
    x2(i)=x(i)/(2*r);
end

plot(x2,Ur,'b')
xlabel('r/d')
ylabel('Ur')
```

## A.2. Potential Flow – Shammaa Solution

```
clear;
clc;

%r=0.1;
%theta=pi/4;
%d=0.2;
r=input('input r (distance of the point from the sink) in meters\n');
theta=input('input theta (polar angle) in degrees\n');
d=input('input d (orifice diameter) in meters\n');
Q=input('input Q (volumetric flowrate) in m3/s\n');
fun=@(r0,theta0) (r0./sqrt(r0.*r0+r.*r-2*r.*r0*cos(theta).*cos(theta0)));
q=integral2(fun,0,d/2,0,2*pi)
f=-2.*Q./(pi^2.*d^2).*q

fun2=@(r0,theta0) ((r.*r0-r0.*r0*cos(theta).*cos(theta0))./(sqrt(r0.*r0+r.*r-
2*r.*r0*cos(theta).*cos(theta0)).*sqrt(r0.*r0+r.*r-
2*r.*r0*cos(theta).*cos(theta0)).*sqrt(r0.*r0+r.*r-2*r.*r0*cos(theta).*cos(theta0))));
n=integral2(fun2,0,d/2,0,2*pi)
Vr=2.*Q./(pi^2.*d^2).*n
```

### A.3. Potential Flow – Bryant Solution

```
clear;
clc;
x=input('input x distance in meters\n');
y=input('input y distance in meters\n');
z=input('input z distance in meters\n');
d=input('input d (orifice diameter) in meters\n');
Q=input('input Q (flowrate) in cubic meters/second\n');
U0=input('input U0 (average velocity) in meters/second\n');
potential_integral=@(r0,theta0) (r0./sqrt(x*x+(y-r0.*sin(theta0)).*(y-
r0.*sin(theta0))+(z-r0.*cos(theta0)).*(z-r0.*cos(theta0))));
p=integral2(potential_integral,0,d/2,0,2*pi);
velocity_potential=-2*Q*p/(pi*pi*d*d);

Vx_integral=@(r0,theta0) ((x*r0)./sqrt((x*x+(y-r0.*sin(theta0)).*(y-
r0.*sin(theta0))+(z-r0.*cos(theta0)).*(z-r0.*cos(theta0))).*(x*x+(y-
r0.*sin(theta0)).*(y-r0.*sin(theta0))+(z-r0.*cos(theta0)).*(z-
r0.*cos(theta0))).*(x*x+(y-r0.*sin(theta0)).*(y-r0.*sin(theta0))+(z-
r0.*cos(theta0)).*(z-r0.*cos(theta0))));
Vx=(integral2(Vx_integral,0,d/2,0,2*pi))*U0/(2*pi);

Vy_integral=@(r0,theta0) (((y-r0.*sin(theta0)).*r0)./sqrt((x*x+(y-
r0.*sin(theta0)).*(y-r0.*sin(theta0))+(z-r0.*cos(theta0)).*(z-
r0.*cos(theta0))).*(x*x+(y-r0.*sin(theta0)).*(y-r0.*sin(theta0))+(z-
r0.*cos(theta0)).*(z-r0.*cos(theta0))).*(x*x+(y-r0.*sin(theta0)).*(y-
r0.*sin(theta0))+(z-r0.*cos(theta0)).*(z-r0.*cos(theta0))));
Vy=(integral2(Vy_integral,0,d/2,0,2*pi))*U0/(2*pi);

Vz_integral=@(r0,theta0) (((z-r0.*cos(theta0)).*r0)./sqrt((x*x+(y-
r0.*sin(theta0)).*(y-r0.*sin(theta0))+(z-r0.*cos(theta0)).*(z-
r0.*cos(theta0))).*(x*x+(y-r0.*sin(theta0)).*(y-r0.*sin(theta0))+(z-
r0.*cos(theta0)).*(z-r0.*cos(theta0))).*(x*x+(y-r0.*sin(theta0)).*(y-
r0.*sin(theta0))+(z-r0.*cos(theta0)).*(z-r0.*cos(theta0))));
Vz=(integral2(Vz_integral,0,d/2,0,2*pi))*U0/(2*pi);

velocity_potential
Vx
Vy
Vz
```

## APPENDIX B

### FLUENT Residuals

#### B.1. Çobanoğlu Model Inviscid Solution

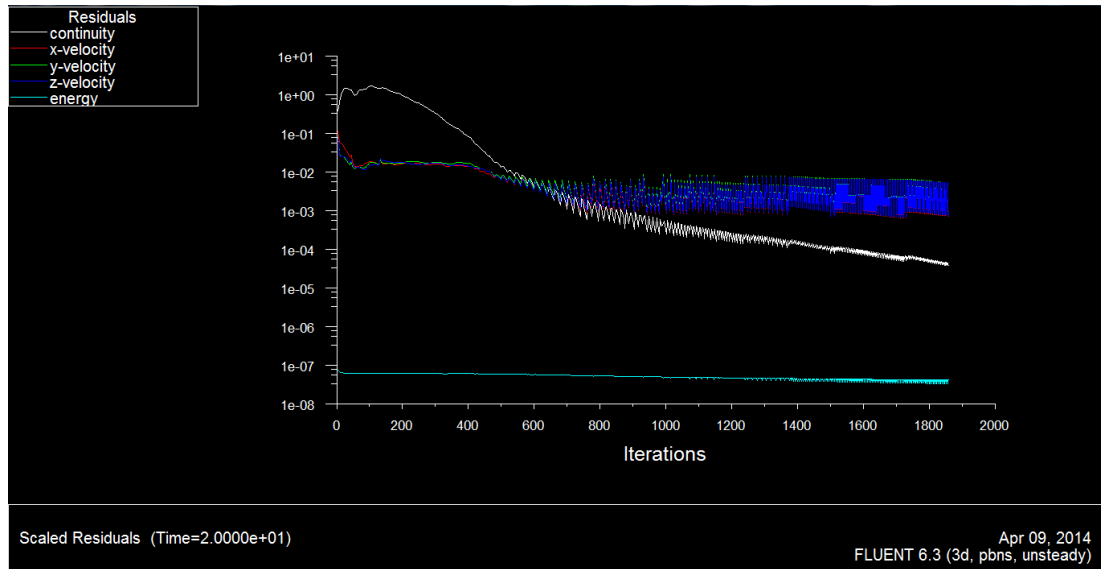


Figure B.1. Residuals of the inviscid flow solution

#### B.2. Enlarged Model Inviscid Solution

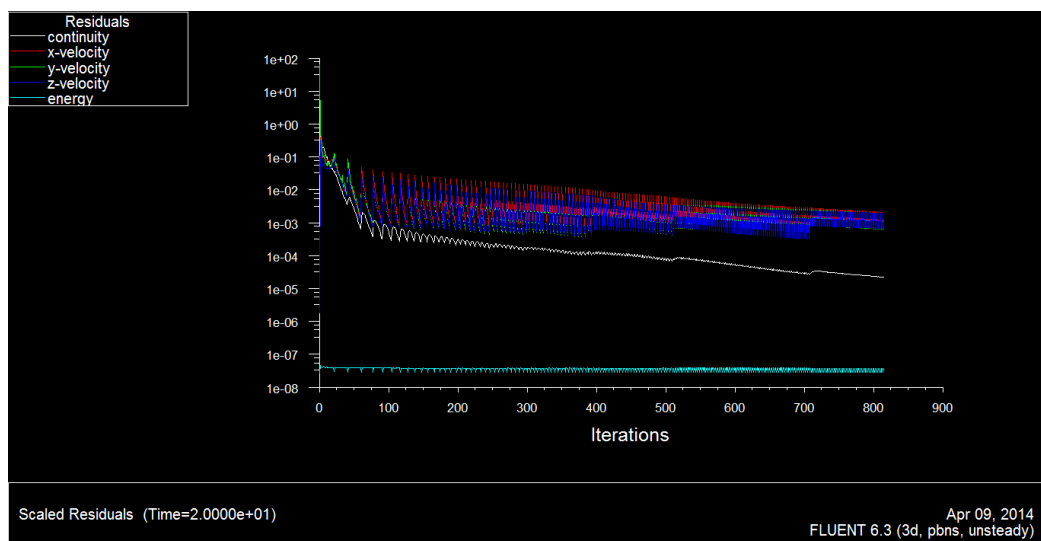


Figure B.2. Residuals of the inviscid flow solution

### B.3. Enlarged Model Laminar Flow Solution

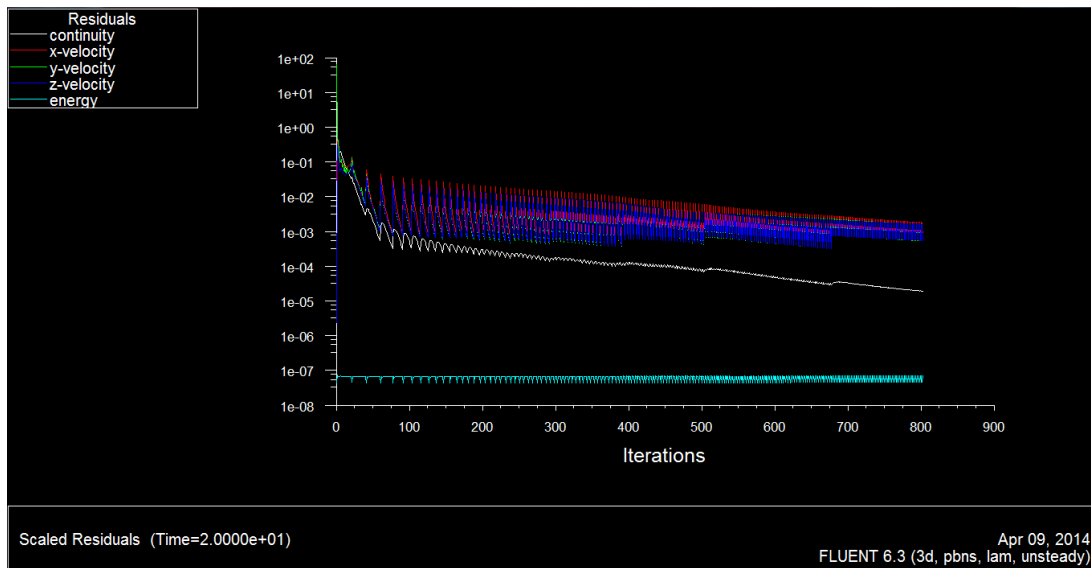


Figure B.3. Residuals of the laminar flow solution

### B.4. Enlarged Model k- $\epsilon$ Solution

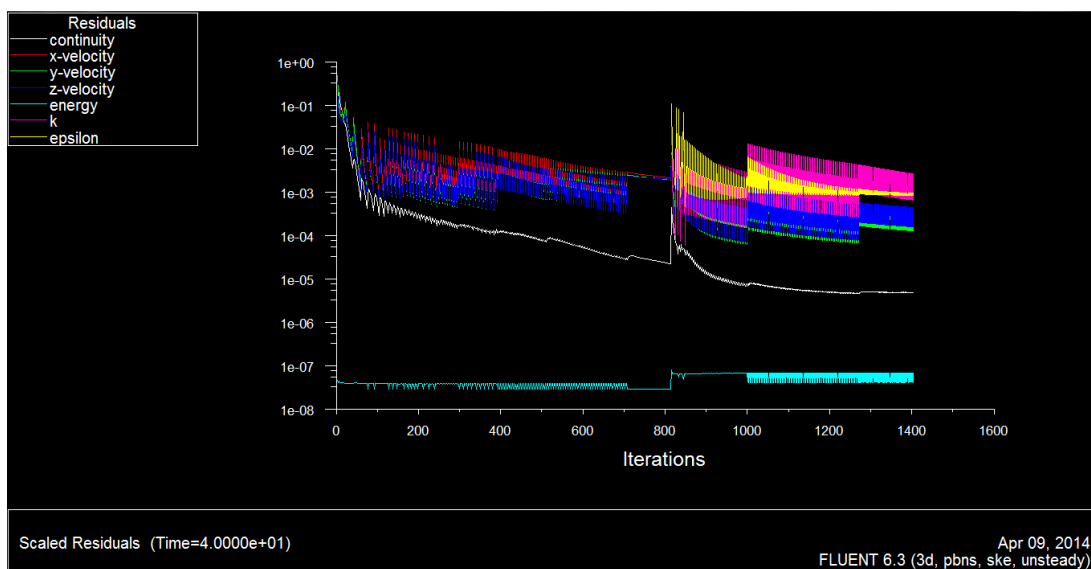


Figure B.4. Residuals of the k- $\epsilon$  solution

## B.5. Enlarged Model VOF Solution

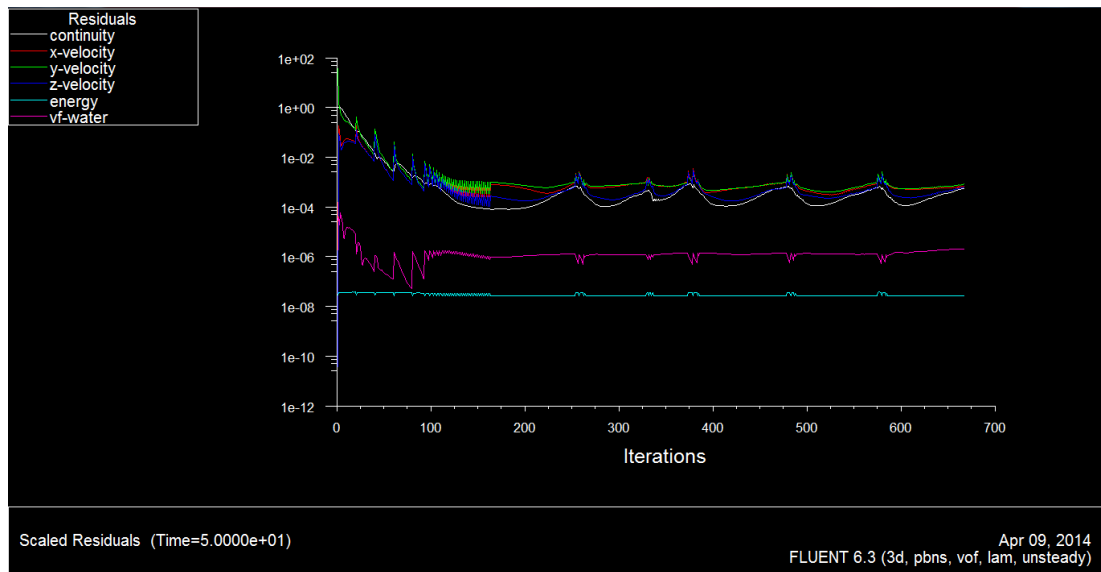


Figure B.5. Residuals of the vof solution

## APPENDIX C

### Volume of Fluid $k - \varepsilon$ Turbulence Solution for Çobanoğlu Model

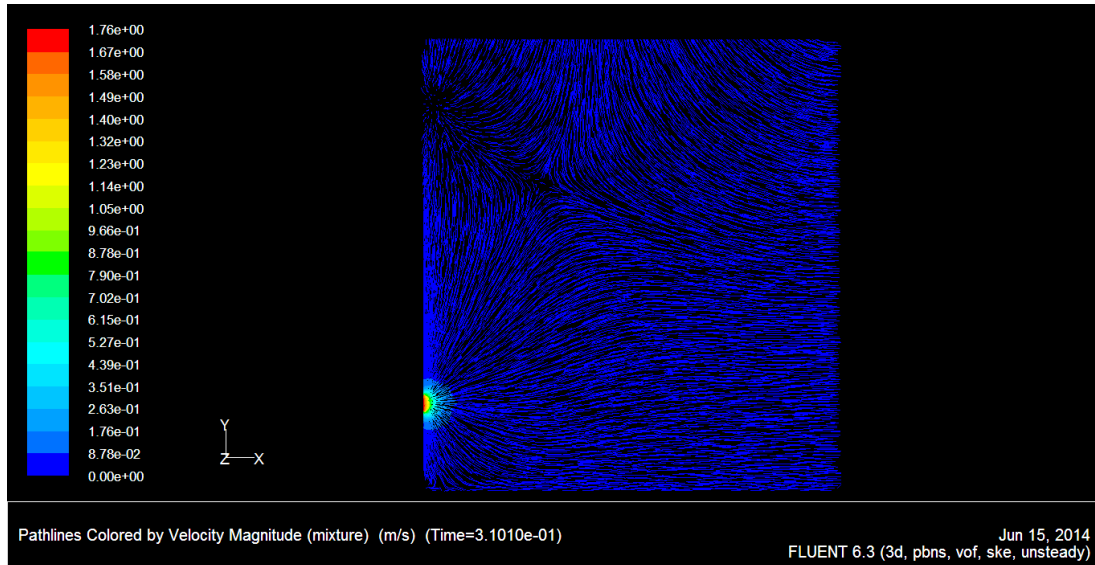


Figure C.6. Pathlines by velocity magnitude

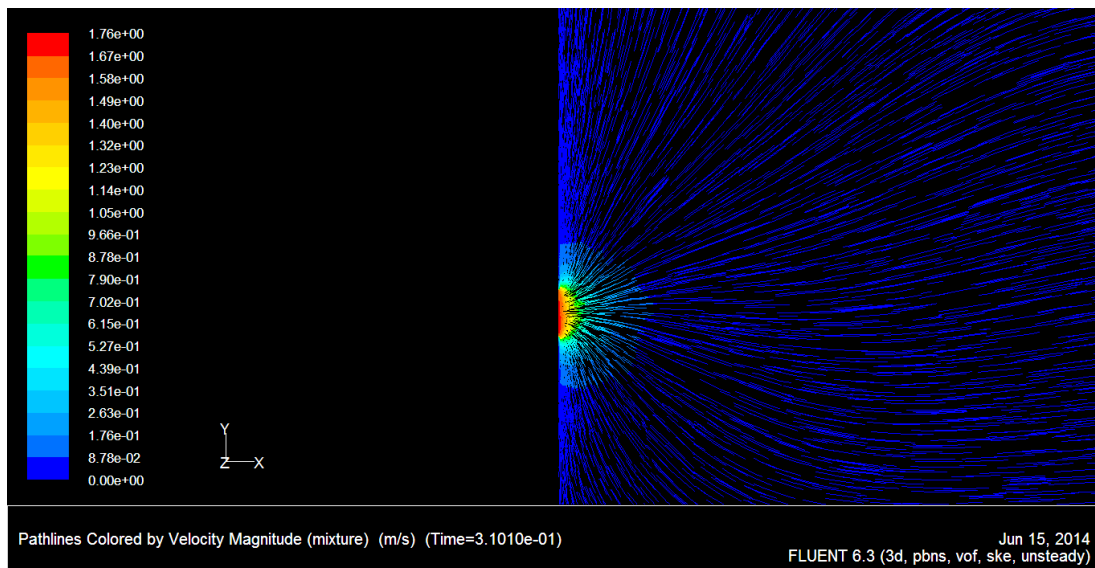


Figure C.7. Close up of the orifice area for the pathlines by velocity magnitude

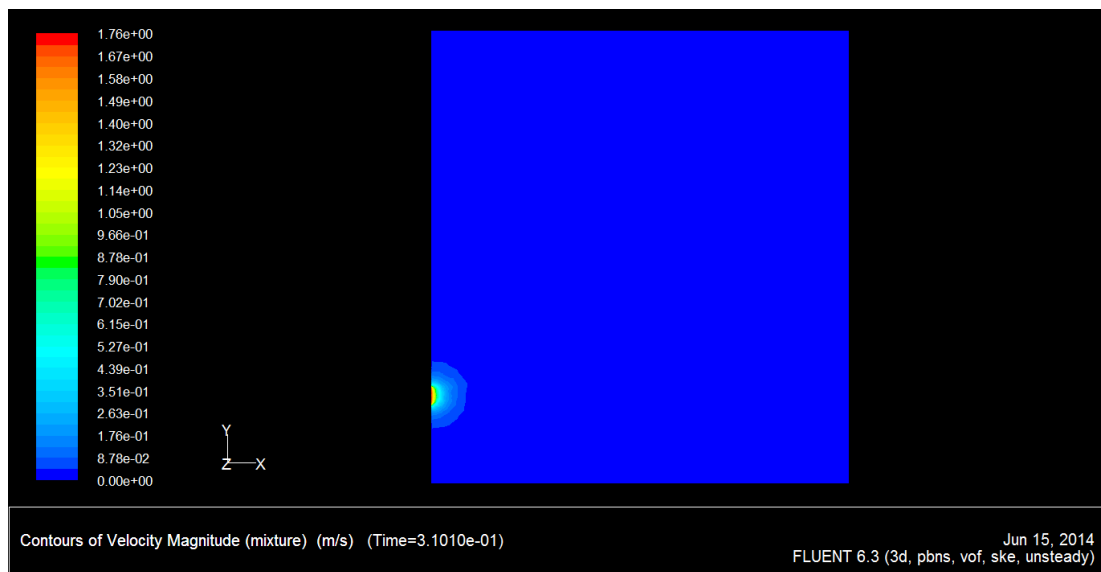


Figure C.8. Contours of velocity magnitude

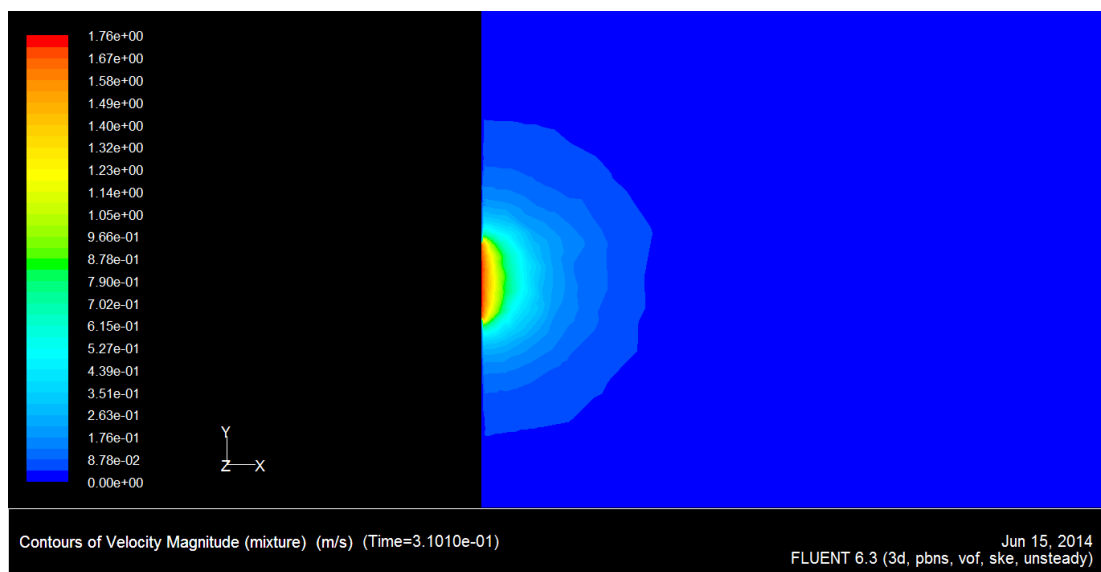


Figure C.9. Close up of the orifice area for the contours of velocity magnitude



Figure C.10. Contours of velocity magnitude

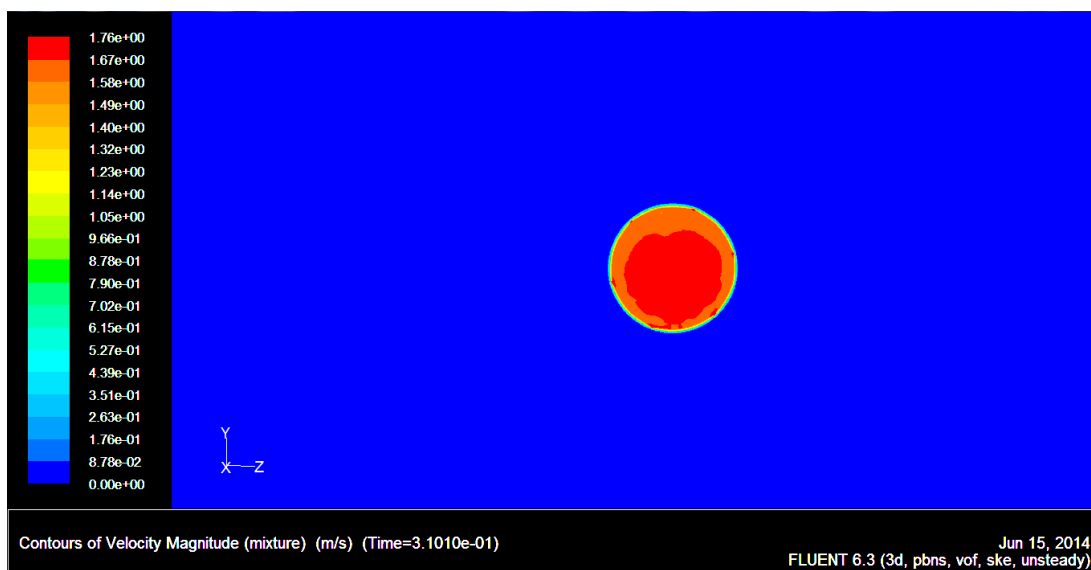


Figure C.11. Close up of the orifice area for the contours of velocity magnitude

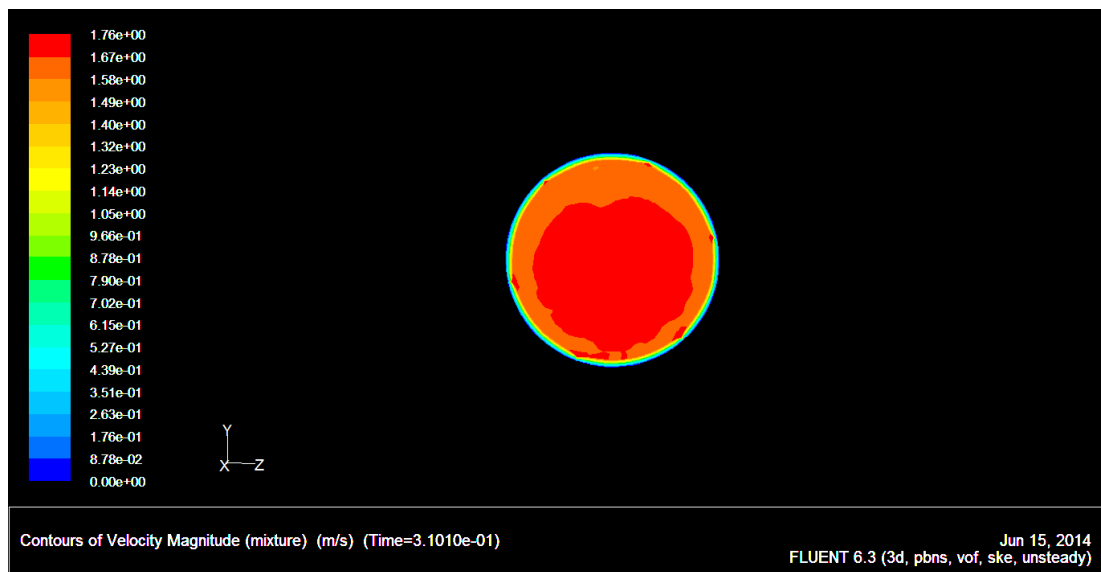


Figure C.12. Velocity magnitude contours of the orifice area only

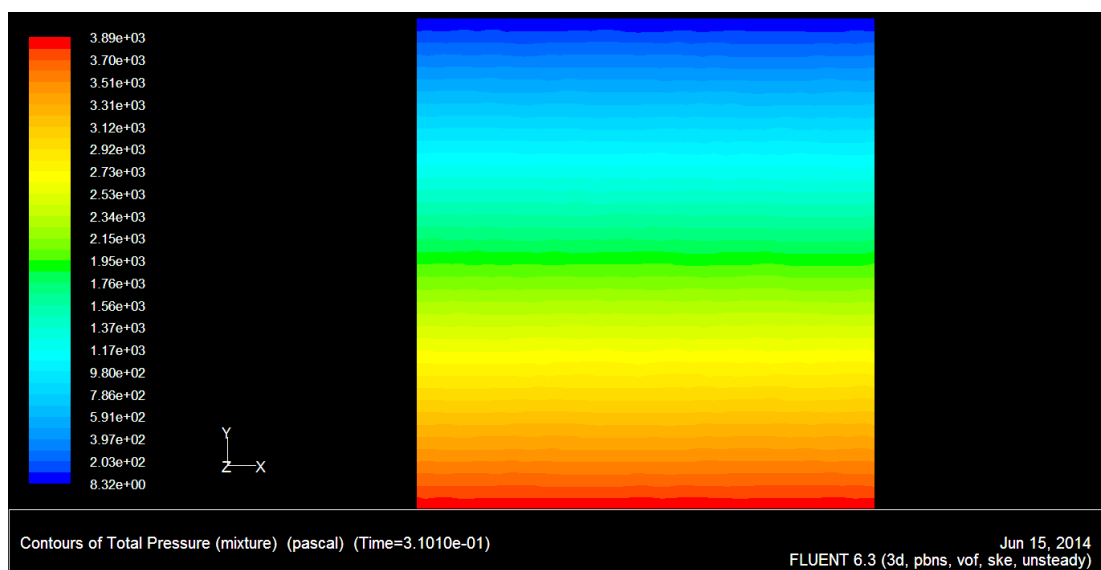


Figure C.13. Pressure contours



Figure C.14. Pressure contours of the orifice area

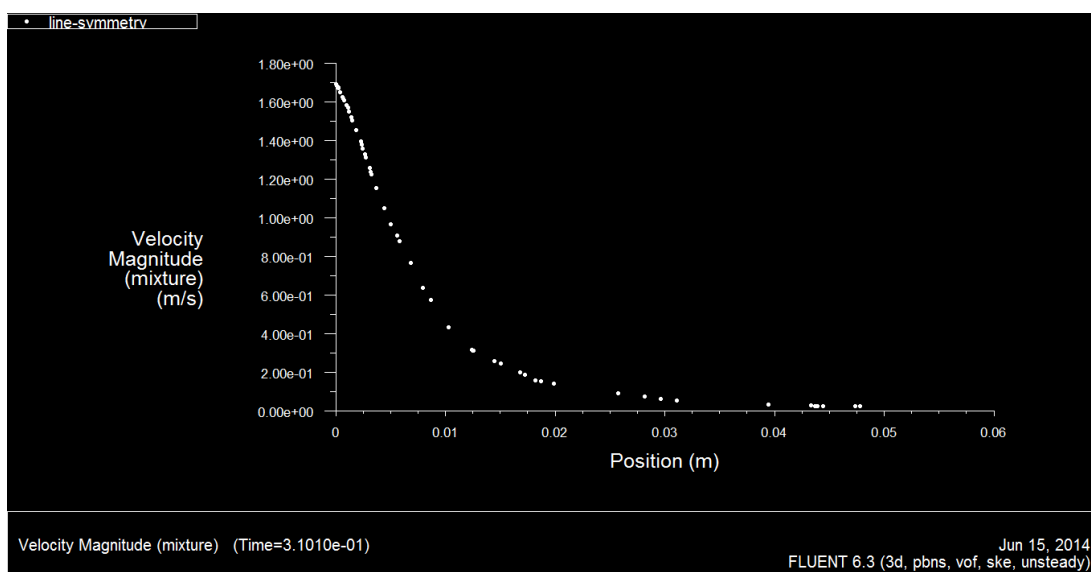


Figure C.15. Velocity magnitude readings of the centerline velocity

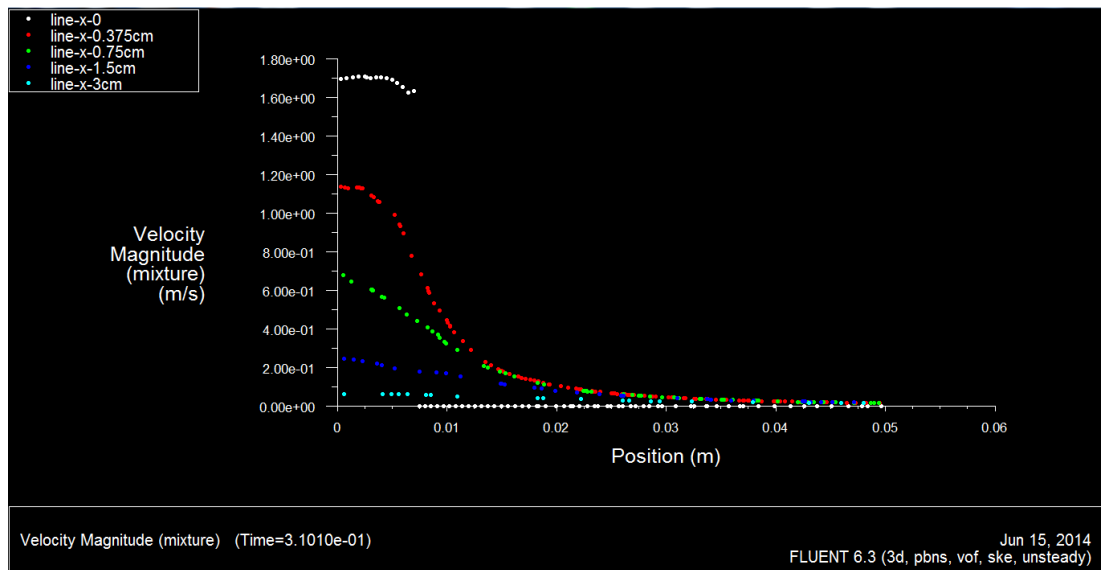


Figure C.16. y-z plane velocity magnitude on z-axis for different x values

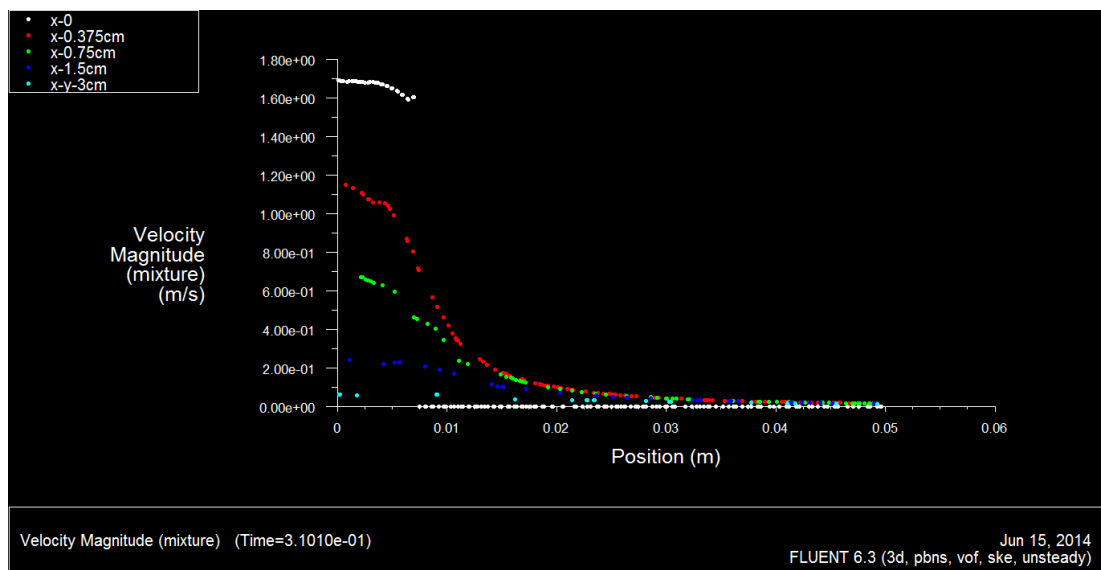


Figure C.17. y-z plane velocity magnitude on y-axis for different x values

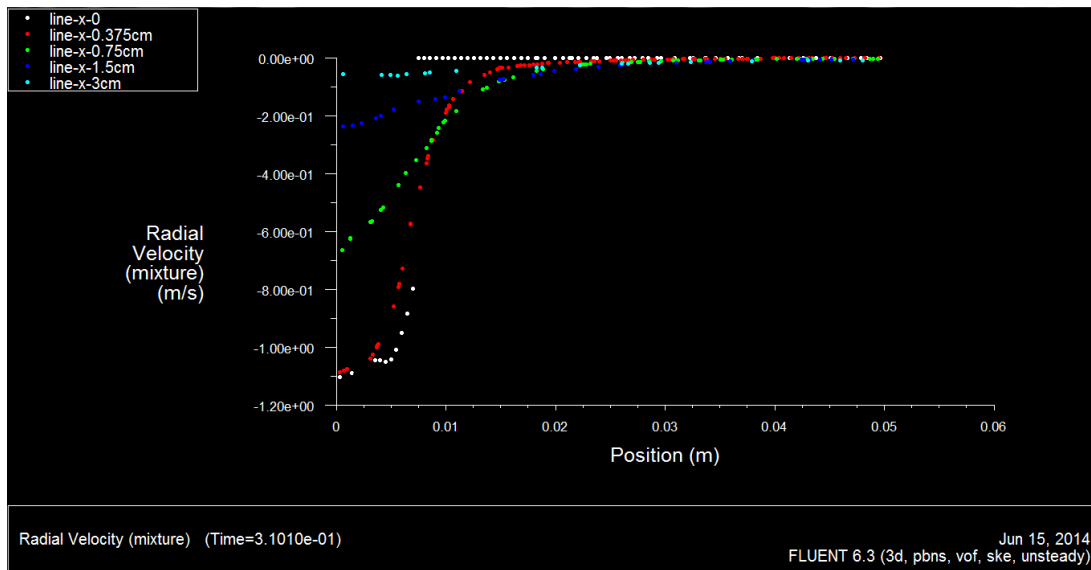


Figure C.18.  $V_r$  vs.  $z$ , for different  $x$  values

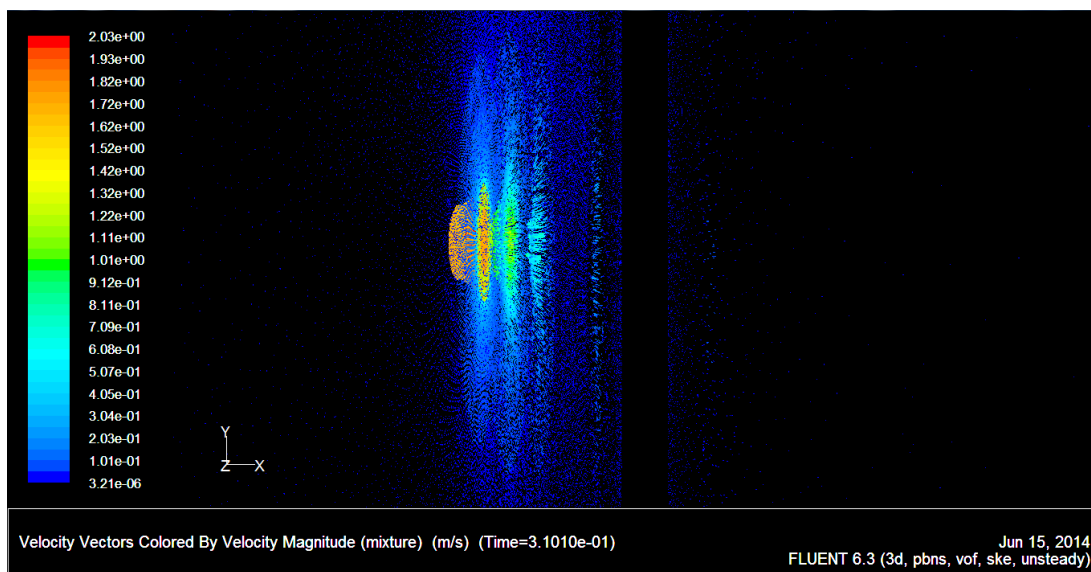


Figure C.19. Velocity vectors for  $x=0, 0.375\text{cm}, 0.75\text{cm}, 1.5\text{cm}, 3\text{cm}$