CFD investigation of a switched vortex valve for cooling air flow modulation in aeroengine

This item was submitted to Loughborough University's Institutional Repository by the/an author.

Additional Information:

- A thesis submitted in fulfilment of the requirements for the degree of Doctor of Philosophy in the Thermo-Fluids Systems UTC Department of Mechanical Engineering Sciences University of Surrey

Metadata Record: https://dspace.lboro.ac.uk/2134/20146

Publisher: © Bharat Ramesh Koli

Rights: This work is made available according to the conditions of the Creative Commons Attribution-NonCommercial-NoDerivatives 4.0 International (CC BY-NC-ND 4.0) licence. Full details of this licence are available at: https://creativecommons.org/licenses/by-nc-nd/4.0/

Please cite the published version.
CFD Investigation of a Switched Vortex Valve for Cooling Air Flow Modulation in Aeroengine

A thesis submitted in fulfilment of the requirements for the degree of Doctor of Philosophy in the
Thermo-Fluids Systems UTC
Department of Mechanical Engineering Sciences
University of Surrey

by

Bharat Ramesh Koli

February 2015 © 2014 Bharat Ramesh Koli
Statement of Originality¹

This thesis and the work to which it refers are the results of my own efforts. Any ideas, data, images or text resulting from the work of others (whether published or unpublished) are fully identified as such within the work and attributed to their originator in the text, bibliography or in footnotes. This thesis has not been submitted in whole or in part for any other academic degree or professional qualification. I agree that the University has the right to submit my work to the plagiarism detection service TurnitinUK for originality checks. Whether or not drafts have been so-assessed, the University reserves the right to require an electronic version of the final document (as submitted) for assessment as above.

(Bharat Ramesh Koli) Date

¹From the university website http://www.surrey.ac.uk/library/researcher/pgr/information/milestones/thesis/
Acknowledgements

This research work has been funded by the Rolls-Royce plc. and the Engineering and Physical Sciences Research Council of the United Kingdom through the University of Surrey Postgraduate Studentship award. I am indebted to these organizations for providing me this wonderful opportunity and funding my entire research.

I am grateful to Prof. John CHEW who has been my principal supervisor, teacher, mentor and well wisher throughout my PhD research. Without his inputs, guidance, assistance, encouragement and feedback I wouldn’t have reached this far. I am thankful to my co-supervisor Prof. Nick Hills for his valuable inputs and assistance.

Particular thanks goes to Dr. Umesh Javiya who has helped me during the initial stages of my research by teaching how to use Hydra and ICEMCFD. I also take this opportunity to thank Dr. Vladislav Ganine for his invaluable suggestions during the research. I thank my past and present colleagues at the TFSUTC for making this place exciting for research.

I am grateful to my parents and brother for their constant encouragement and support.

Finally, I express my gratitude to my beloved wife, Sneha, for her unconditional support, patience and comfort without which I wouldn’t have come so far.
Summary

This thesis is focused on understanding the flow features associated with a Switched Vortex Valve (SVV) using Computational Fluid Dynamics (CFD) methods for application in aero-engines. In this research the major emphasis was put on detailed flow analysis which was limited in experimental studies of SVVs. Considering the complex geometry of the SVV, for simplicity it was decided to divide the SVV device into two parts before studying the device as whole. These parts are the vortex chamber and flow switching device, which together constitute the SVV device. In this research, different turbulence models were evaluated which are mainly Reynolds average Navier-Stokes (RANS) and unsteady Reynolds average Navier-Stokes (URANS). The different turbulence models are the $k$-$\varepsilon$, SA and the RSM, and for a test case LES was also implemented.

The vortex chamber is an important component of the SVV. The experiments conducted on the SVV at the University of Sheffield by Romero (2014), found that the flow with in the vortex chamber for the application considered here is compressible in nature. The experiments performed by Savino & Keshock (1965) with confined vortex chamber were considered for comparison with the numerical simulations. In these experiments the flow within the vortex chamber was reported to extend up to the compressible regime. As most previous researchers have considered incompressible flow conditions for simulation of vortex chamber flows, the present work was focused on the prediction of compressible flows. The Reynolds stress model (RSM) showed the best agreement with the experimental data, whereas the $k$-$\varepsilon$ model showed the least agreement. The SA model showed better predictions than the $k$-$\varepsilon$ model but was found inferior to the RSM.

The experiments by Tesař (2010) for a flow switching device were considered for the numerical study. The methodology of flow switching in the device experimented by Tesař (2010) is analogous to flow switching requirements of the SVV considered here. The turbulence models considered in this study are LES and URANS. The URANS include the RSM, $k$-$\varepsilon$ and the SA model. While URANS showed qualitative agreement with the measured flow characteristics, LES showed
large differences with measured data. The disagreement of LES was attributed to the insufficient grid resolution due to limited computational resources. As the motivation for the study was to develop methods for everyday use in industry, LES was not considered further.

Based on the performance evaluation of different turbulence models, the SA and the RSM were chosen for simulation of the SVV device. The numerical simulation involved prediction of the flow state from the start-up of the device and mass flow rate through the device for different pressure ratios. The RSM showed better agreement with the measured mass flow rate than the SA model for different flow states and different pressure ratios. The RSM also correctly predicted the start-up state of the device for different pressure ratios, whereas the SA model failed to predict it for a pressure ratio of 2.48. In this part of the research, the numerical studies were further extended to understand the flow switching dynamics using only the RSM. The study revealed that the square root ratio of momentum at nozzle to control port plays an important role in flow switching which is in accordance with experimental work carried by Feikema & Culley (2008). In this study an attempt was also made to understand the effects of vibration (which are due to the dynamic working environment of an aero-engine) on the performance of the device. Different scenarios of vibration were considered which are vibration with maximum acceleration 0.5g and 10g. The study revealed that both scenarios have negligible effect on the performance of the device and hence the device is expected to be safe for operation with vibration levels up to maximum acceleration of 10g.
Contents

Statement of Originality i
Acknowledgements ii
Summary iii
List of Figures ix
List of Tables xiv
Abbreviations xv
Symbols xvii

1 Introduction 1
  1.1 The need for blade cooling ......................... 2
  1.1.1 Functions of secondary air systems .......... 2
  1.2 Cooling air modulation .......................... 5
  1.3 Problem definition ............................... 6
  1.4 Objectives of the present research .............. 7
  1.5 Outline of the thesis ............................ 8

2 Literature Review 10
  2.1 Vortex amplifiers ................................ 12
  2.1.1 Vortex amplifier characteristics .......... 14
  2.2 Wall attached fluidic devices .................... 19
  2.2.1 Load characteristics curves ................. 20
  2.3 Conclusion ....................................... 27

3 Turbulence modeling and Numerical Methods 29
  3.1 Flow governing equations ........................ 30
  3.2 Reynolds Averaged Navier-Stokes ................. 31
  3.2.1 Scale analysis of Reynolds stresses with viscous stresses ... 32
3.3 Turbulence modelling ........................................... 33
3.4 The mean and turbulent kinetic energy equation .......... 36
3.5 Dissipation ....................................................... 38
3.6 Eddy viscosity modeling ....................................... 39
3.7 The Spalart-Allmaras Model .................................. 40
3.8 The $k$-$\varepsilon$ model ....................................... 42
3.9 The Reynolds Stress Model (RSM) ............................ 45
3.10 Computational Implementation in FLUENT ................. 47
3.11 CFD simulations ................................................. 49

4 Numerical analysis of confined swirling flows 51
4.1 Introduction ...................................................... 51
4.2 General Background ............................................ 52
4.3 Physics of confined swirling flows ............................ 53
4.4 Literature review ............................................... 55
  4.4.1 Experimental work ......................................... 55
  4.4.2 Computational work ........................................ 57
4.5 Savino (1965) test case ......................................... 58
4.6 Numerical set up ............................................... 60
4.7 Results .......................................................... 63
4.8 Conclusion ...................................................... 70

5 Numerical study of flow switching device 73
5.1 Introduction ...................................................... 73
5.2 Literature review ............................................... 74
5.3 Tesar (2010) test case .......................................... 77
  5.3.1 Characteristic Curve ....................................... 78
5.4 Numerical set up ............................................... 80
  5.4.1 Turbulence models .......................................... 81
    5.4.1a Spalart-Allmaras Model ................................. 81
    5.4.1b $k$-$\varepsilon$ Model ..................................... 81
    5.4.1c Reynolds Stress Model .................................. 82
    5.4.1d Large Eddy Simulation Model ......................... 82
  5.4.2 Boundary Conditions for Numerical Simulation .......... 83
  5.4.3 Criteria for simulation convergence ...................... 85
  5.4.4 Mesh Dependency Study .................................... 86
  5.4.5 Time Step Study of URANS ................................ 89
5.5 Results of numerical simulation for device ............... 90
  5.5.1 Test case 1 (Flow case at $\mu_Y = 1$) .................... 92
  5.5.2 Test case 2 (Flow case at $\mu_Y \approx 0.86$) .......... 98
  5.5.3 Test case 3 (Flow case at $\mu_Y \approx 0.64$) .......... 99
  5.5.4 Test case 4 (Flow case at $\mu_Y \approx 0.57$) .......... 100
    5.5.4a Numerical prediction of flow switching ............. 101
5.6 Conclusion ...................................................... 103
6 Numerical study of Switched Vortex Valve 105
6.1 Objective of the research ........................................ 106
6.2 Switched vortex valve ........................................... 107
   6.2.1 Introduction .................................................. 107
   6.2.2 Application in Aero-engines ................................. 109
6.3 Experimental Study ............................................. 110
   6.3.1 High flow state ............................................. 110
   6.3.2 Low flow state ............................................. 112
6.4 CFD Modeling .................................................... 113
   6.4.1 Boundary Conditions for Numerical Simulation .............. 114
      6.4.1a Stable high flow state .................................. 115
      6.4.1b Switching from high to low flow state ................. 116
      6.4.1c Stable low flow state .................................. 116
      6.4.1d Switching from low to high flow state ................. 116
6.5 Mesh and time step dependency ................................ 117
6.6 Results of numerical simulation for device ....................... 121
   6.6.1 High flow case with pressure ratio of 1.54 ................ 121
   6.6.2 Low flow case with pressure ratio of 1.54 ................ 122
   6.6.3 Prediction of bi-stable characteristic for pressure ratio of 1.54 .......................................................... 127
   6.6.4 High flow case with pressure ratio of 2.48 ................ 127
   6.6.5 Low flow case with pressure ratio of 2.48 ................ 129
   6.6.6 Prediction of bi-stable characteristic for pressure ratio of 2.48 .......................................................... 129
6.7 Conclusion ....................................................... 130

7 Dynamic Switching Characteristics of an SVV 133
7.1 The objective of the study ........................................ 135
7.2 Numerical Setup .................................................. 135
7.3 Boundary conditions ............................................. 136
7.4 Results ............................................................. 137
   7.4.1 Switching from high to low flow state ..................... 137
      7.4.1a Test case with pressure ratio 1.54 .................... 137
      7.4.1b Test case with pressure ratio 2.48 .................... 144
   7.4.2 Switching from low to high flow state ..................... 145
      7.4.2a Test case with pressure ratio 1.54 .................... 146
      7.4.2b Test case with pressure ratio 2.48 .................... 150
7.5 Conclusion ....................................................... 153

8 CFD study of the effect of vibration on the performance of the device 155
8.1 Objective of the study ........................................... 156
8.2 Numerical set up .................................................. 157
8.3 Results ............................................................. 160
   8.3.1 $a_{max} = 0.5g$ test case .................................. 160
   8.3.2 $a_{max} = 10g$ test case .................................. 162
8.4 Conclusion .......................................................... 163

9 Conclusion and future work ....................................... 164
  9.1 Suggested Future work .......................................... 166

A Dimensions of the SVV device .................................. 167

Bibliography ................................................................ 170
  REFERENCES ............................................................. 170
List of Figures

1.1 Turbine entry temperature for Rolls-Royce engines since 1940 (Cumpsty (2008)) ........................................ 3
1.2 A typical secondary air system of Rolls-Royce Trent engine. Courtesy: Rolls-Royce plc. ............................ 4

2.1 Schematic of a typical vortex amplifier (from Humphrey & Tarumato (1965)) ........................................ 13
2.2 Different states of operation of a vortex amplifier (A) High flow state (B) Low flow state (from Humphrey & Tarumato (1965)) . . . 14
2.3 Circuit symbol of a vortex amplifier ........................................ 15
2.4 A vortex amplifier characteristics curve (From Priestman & Tippets (1984)) ........................................ 16
2.5 A vortex amplifier showing critical geometrical parameters (From King (1985)) ........................................ 17
2.6 A typical wall attached fluidic device showing critical components (From Belsterling (1971)) ...................... 20
2.7 Different size of steps results in preferred flow direction .......... 20
2.8 Bistable characteristic of a wall attached fluidic device ......... 21
2.9 Circuit symbols of a typical wall attached fluidic device ........ 22
2.10 Typical output characteristic curves of a wall attached fluidic device .................................................. 23
2.11 A wall attached fluidic device with conical shape splitter ...... 23
2.12 A typical load characteristic curve of a device experimented by Tesar (2010) ........................................ 24
2.13 A section of jet controlled wall attached fluidic device Feikema & Culley (2008) ........................................ 25
2.14 Different configurations of the jet controlled wall attached fluidic device experimented by Heo et al. (2010) .......... 26
2.15 Schematic arrangement showing essential features of SVV used by Scanlon et al. (2009) ......................... 27

4.1 A typical switched vortex valve numerically tested by Scanlon et al. (2009) showing contours of velocity magnitude from CFD on a mid-section plane .................................................. 51
4.2 Schematic of flow pattern in the confined vortex chamber .... 54
4.3 Schematic arrangement of experimental setup showing essential features of vortex chamber experimented by Savino & Keshock (1965) . 59
List of Figures

4.4 Schematic of mesh used for Savino & Keshock (1965) (a) 2D model (b) 3D 10 degree Sector ............................................... 62
4.5 Radial non-dimensional static pressure distribution on left hand wall of the vortex chamber for different turbulence models with standard FLUENT wall functions. (a) k-ε FLUENT (b) SA-FLUENT (c) RSM-FLUENT .......................................................... 64
4.6 Comparison of non-dimensional static pressure distribution with experimental data .................................................. 65
4.7 Turbulent viscosity variation for different turbulence models using wall functions ...................................................... 65
4.8 Swirl velocity variation for different turbulence models using wall functions ............................................................... 65
4.9 Comparison of computed radial velocity variation with experimental data (a) r/b = 0.52 (b) r/b = 0.256 ......................................................... 66
4.10 Contour of stream functions for different turbulence model using wall functions (a) SA (b) RSM (c) k-ε ........................ 67
4.11 Computed non-dimensional velocity with resolving near wall treatment against $\Delta y^+$ for different turbulence models at r/b = 0.256 of left hand disc ............................................................ 68
4.12 Comparison of computed non-dimensional static pressure distribution on left hand wall of the vortex chamber using resolved nearwall treatment with experimental data ........................................... 68
4.13 Turbulent viscosity variation for different turbulence models using resolved nearwall treatment ........................................ 69
4.14 Comparison of computed radial velocity using resolved nearwall treatment with experimental data (a) r/b = 0.52 (b) r/b = 0.256 ................................. 69
4.15 (a) Comparison of computed flow properties by VP94 with different turbulence models with resolved nearwall treatment (a) Static pressure variation on left hand wall (b) Swirl velocity variation along the centerline of vortex chamber ......................................................... 70
4.16 Comparison of computed radial velocity with experimental data at $r/r_b = 0.17$ ................................................................. 71

5.1 A typical switched vortex valve with contours of predicted velocity magnitude, numerically tested by Scanlon et al. (2009) ........ 74
5.2 Schematic arrangement showing essential features of flow switching device used by Tesař (2010) ................................. 78
5.3 Characteristic curve obtained for flow switching device shown in Figure 5.2 for Re= 27300 ...................................................... 79
5.4 Location of extra geometry (shown by red color) added to original model ................................................................. 84
5.5 (a) The instantaneous variation of total gauge pressure at the inlet of device (b) The variation of magnitude of time averaged total gauge pressure at the inlet of device 86
5.6 Typical mesh configuration in xy plane (a) Mesh for whole device at mid-z plane of the device (b) Mesh for critical region with refinement at steps and splitter area .................................. 87
5.7 Location of plane where velocity flow field for different meshes compared in Figure 5.8 ................................................................. 88
5.8 Contours of velocity magnitude at plane near to splitter as shown in Figure 5.7 for different meshes (a) Coarse (b) Medium (c) Fine, in yz plane ................................................................. 89
5.9 Comparison of numerical results with experimental data (a) with non-dimensional specific energy difference ($\epsilon_Y$) (b) with output specific energy difference between reference ‘V’ and ‘Y’ ($\Delta e_Y$) ........ 91
5.10 (a) Location of measuring probe (shown in blue color) (b) Fluctuation of velocity magnitude recorded by measuring probe for different turbulence models ........................................... 93
5.11 Plot of frequency vs amplitude for velocity magnitude measured at the probe ................................................................. 94
5.12 Contour plot of non-dimensional time averaged total pressure ($\overline{P^*}$) for (a) SA model (b) k-$\epsilon$ model (c) RSM (d) LES ................................................................. 95
5.13 Contour plot of time averaged velocity magnitude (m/s) for (a) SA model (b) k-$\epsilon$ model (c) RSM (d) LES ................................................................. 96
5.14 Contour plot of turbulent viscosity ratio in xy plane at z=0.0264 m for (a) SA model (b) k-$\epsilon$ model (c) RSM ................................................................. 97
5.15 Comparison of numerically obtained $\Delta e_Y$ with measured data (a) With all turbulence models (b) RSM and k-$\epsilon$ model with extra simulations ................................................................. 102
6.1 Schematic arrangement showing essential features of SVV used by Scanlon et al. (2009) ................................................................. 108
6.2 A typical secondary air system in an aero-engine with proposed SVV ................................................................. 109
6.3 Schematic of experimental setup with instrumentation ................................................................. 111
6.4 Variation of measured non-dimensional mass flow rate ($\dot{m}_{non}$) of air with applied pressure ratio in the high flow state (from Romero (2014)) ................................................................. 111
6.5 Variation of measured non-dimensional mass flow rate ($\dot{m}_{non}$) with applied pressure ratio in the low flow state (from Romero (2014)) ................................................................. 112
6.6 Typical mesh configuration in xy plane. General mesh distribution in whole device for (a) 0.5 million mesh size (b) 2.7 million mesh size (c) Mesh refinement at control ports (d) Higher mesh density near walls, such as splitter ................................................................. 114
6.7 CFD domain of SVV with locations of boundary conditions ................................................................. 115
6.8 Comparison of numerically computed mass flow rate by unsteady SA and RSM for different meshes with experimental data ................................................................. 118
6.9 Prediction of time averaged velocity magnitude at point ‘p01 (shown in Figure 6.15)’ for different mesh sizes ................................................................. 119
6.10 Contours of predicted time averaged velocity magnitude (m/s) at the mid-plane of the device in mid xy plane. (a) For 0.5 million mesh size with the RSM (b) For 2.7 million mesh size with the RSM (c) For 0.5 million mesh size with the SA model (d) For 2.7 million mesh size with the SA model ........................................ 120

6.11 Contours of predicted time averaged velocity magnitude (m/s) at mid-plane of the device in xy plane for low flow with pressure ratio of 1.54 (a) RSM (b) SA ......................................................... 123

6.12 Variation of swirl velocity (m/s) in vortex chamber taken at line shown in Figure 6.15 (a) ................................................................. 124

6.13 Variation of Reynolds stresses predicted by the RSM in vortex chamber taken at line shown in Figure 6.15 (a) ........................................ 125

6.14 Variation of Reynolds stresses predicted by the SA model in vortex chamber taken at line shown in Figure 6.15 (a) ........................................ 125

6.15 (a) Location of a sampling probe (pt1) in mid-plane of the geometry (b) Unsteady variation of velocity magnitude (m/s) in vortex chamber taken at ‘pt1’ ................................................................. 126

6.16 Variation of velocity (m/s) in vortex chamber taken at ‘pt1’ Figure 6.15 ................................................................. 127

6.17 Contours of predicted time averaged velocity magnitude (m/s) at mid-plane of device in xy plane for high flow case with pressure ratio of 2.48 (a) RSM (b) SA ......................................................... 128

6.18 Contours of predicted time averaged velocity magnitude (m/s) by RSM at mid-plane of device in xy plane for low flow with pressure ratio of 2.48 ................................................................. 129

6.19 Variation of velocity (m/s) in vortex chamber taken at ‘pt1’ Figure 6.15 ................................................................. 130

7.1 Variation of control pressure ($p_c$) with respect to time for different rates .................................................................................. 137

7.2 Contours of velocity magnitude (m/s) at mid-plane of the device for test case with pressure ratio 1.54 with $C = 0.5$ (a) Stable high flow state at $t = 0$ (b) stable low flow state after $t = 0.5T_f$ ............... 138

7.3 Contours of velocity magnitude (m/s) at mid-plane of the device for test case with pressure ratio 1.54 after (a) $t = 0.1T_f$ (b) $t = 0.2T_f$ (c) $t = 0.3T_f$ (d) $t = 0.435T_f$ ......................................................... 139

7.4 Contours of velocity magnitude (m/s) at mid-plane of the device for test case with pressure ratio 1.54 with $C = 0.5$ (a) flow state at $t = 0.375T_f$ with $p_c$ limited to 75.7% of $P_{in}$ (b) flow state at $t = T_f$ obtained from 7.4(a) (c) flow state at $t = 0.395T_f$ with $p_c$ limited to 73.7% of $P_{in}$ (d) flow state at $t = T_f$ obtained from 7.4(c) ............... 140

7.5 Variation of $\sqrt{J}$ with respect to time obtained from simulation results shown in Figure 7.2 ................................................................. 142

7.6 Variation of control pressure with different flow through times with clear distinction of ‘No switching’ and ‘Stable switching’ regions . 143
7.7 Contours of velocity magnitude (m/s) at mid-plane of the device for test case with pressure ratio 1.54 after (with \( C = 1 \)) (a) \( t = 0 \) (b) \( t = T_f \) ................................. 146

7.8 Contours of velocity magnitude (m/s) at mid-plane of the device for test case with pressure ratio 1.54 (with \( C = 1 \)) after (a) \( t = 0.1T_f \) (b) \( t = 0.2T_f \) (c) \( t = 0.3T_f \) (d) \( t = 0.4T_f \) ................................. 147

7.9 Contours of velocity magnitude (m/s) at mid-plane of the device for test case with pressure ratio 1.54 (with \( C = 1 \)) (a) flow state at \( t = 0.265T_f \) with \( p_c/P_{in} \) limited to 0.88 (b) flow state at \( t = 1.265T_f \) with no-slip high flow control port boundary (c) flow state at \( t = 0.272T_f \) with \( p_c/P_{in} \) limited to 0.87 (d) flow state at \( t = 1.272T_f \) with no-slip high flow control port boundary ................................. 148

7.10 Variation of \( \sqrt{J} \) with simulation time for test case pressure ratio 1.54 149

7.11 Contours of velocity magnitude (m/s) at \( t = 0.5T_f \) obtained from simulation reported in figure 7.7 ................................. 150

7.12 Contours of velocity magnitude (m/s) at mid-plane of the device for test case with pressure ratio 2.48 (with \( C = 1 \)) after (a) \( t = 0 \) (b) \( t = T_f \) ................................. 151

7.13 Variation of \( \sqrt{J} \) with simulation time for test case pressure ratio 2.48 152

7.14 Contours of velocity magnitude (m/s) at \( t = 0.4T_f \) obtained from simulation reported in figure 7.12 ................................. 153

8.1 The direction of vibration imposed on the SVV ................................. 159

8.2 Variation of imposed \( y \) velocity for a single through flow time ................................. 161

8.3 Contours of instantaneous velocity (m/s) at midplane of the device in xy-plane (a) \( t = 0 \) (b) \( t = 4T_f \) ................................. 161

8.4 Contours of instantaneous velocity (m/s) at midplane of the device in xy-plane (a) \( t = 0 \) (b) \( t = 4T_f \) ................................. 162

A.1 Dimensions of the SVV device experimented by Romero (2014) (a) top view (b) front view ................................. 168

A.2 Dimensions of the splitter of the SVV device ................................. 169
## List of Tables

<table>
<thead>
<tr>
<th>Page</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.1</td>
<td>CFD model set up</td>
</tr>
<tr>
<td>4.2</td>
<td>Comparison of viscous moment predicted by the RSM for different mesh size models</td>
</tr>
<tr>
<td>5.1</td>
<td>List of meshes used for mesh dependency study</td>
</tr>
<tr>
<td>5.2</td>
<td>Details of results obtained from different time steps used in URANS</td>
</tr>
<tr>
<td>5.3</td>
<td>Details of numerical results obtained from different turbulence models at no-spillover condition</td>
</tr>
<tr>
<td>5.4</td>
<td>Comparison of numerically computed $\Delta e_Y$ with measured data at $\mu_Y \approx 0.86 \ (M_Y \approx 0.025 \ kg/s)$</td>
</tr>
<tr>
<td>5.5</td>
<td>Comparison of numerically computed $\Delta e_Y$ with measured data at $\mu_Y \approx 0.64 \ (M_Y \approx 0.019 \ kg/s)$</td>
</tr>
<tr>
<td>5.6</td>
<td>Comparison of numerically computed $\Delta e_Y$ with measured data at $\mu_Y \approx 0.57 \ (M_Y \approx 0.017 \ kg/s)$</td>
</tr>
<tr>
<td>6.1</td>
<td>Details of numerical results obtained from different time steps</td>
</tr>
<tr>
<td>7.1</td>
<td>Results of intermediate flow simulations obtained from simulation with $T_f$ flow time for $C = 1$</td>
</tr>
<tr>
<td>7.2</td>
<td>Results of intermediate flow simulations obtained from simulation with $2T_f$ flow time for $C = 2$</td>
</tr>
<tr>
<td>7.3</td>
<td>Results of intermediate flow simulations obtained from simulation with $0.5T_f$ flow time for $C = 0.5$</td>
</tr>
<tr>
<td>7.4</td>
<td>Results of intermediate flow simulations obtained from simulation with $T_f$ flow time for $C = 1$</td>
</tr>
<tr>
<td>7.5</td>
<td>Results of intermediate flow simulations obtained from simulation with $2T_f$ flow time for $C = 2$</td>
</tr>
<tr>
<td>7.6</td>
<td>Results of intermediate flow simulations obtained from simulation with $T_f$ flow time</td>
</tr>
<tr>
<td>8.1</td>
<td>CFD model set up</td>
</tr>
</tbody>
</table>
# Abbreviations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D</td>
<td>2 Dimensional</td>
</tr>
<tr>
<td>3D</td>
<td>3 Dimensional</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CFL</td>
<td>Courant-Friedrichs Lewy condition</td>
</tr>
<tr>
<td>DNS</td>
<td>Direct Numerical Simulation</td>
</tr>
<tr>
<td>FDM</td>
<td>Finite Difference Method</td>
</tr>
<tr>
<td>FEM</td>
<td>Finite Element Method</td>
</tr>
<tr>
<td>FVM</td>
<td>Finite Volume Method</td>
</tr>
<tr>
<td>HP</td>
<td>High Pressure</td>
</tr>
<tr>
<td>HPC</td>
<td>High Pressure Compressor</td>
</tr>
<tr>
<td>HPT</td>
<td>High Pressure Turbine</td>
</tr>
<tr>
<td>IP</td>
<td>Intermediate Pressure</td>
</tr>
<tr>
<td>IPC</td>
<td>Intermediate Pressure Compressor</td>
</tr>
<tr>
<td>IPT</td>
<td>Intermediate Pressure Turbine</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>LP</td>
<td>Low Pressure</td>
</tr>
<tr>
<td>LPC</td>
<td>Low Pressure Compressor</td>
</tr>
<tr>
<td>LPT</td>
<td>Low Pressure Turbine</td>
</tr>
<tr>
<td>MTO</td>
<td>Maximum Take-Off</td>
</tr>
<tr>
<td>NS</td>
<td>Navier-Stokes</td>
</tr>
<tr>
<td>OPR</td>
<td>Operating Pressure Ratio</td>
</tr>
<tr>
<td>rms</td>
<td>root mean square</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds Averaged Navier Stokes</td>
</tr>
<tr>
<td>RSM</td>
<td>Reynolds Stress Model</td>
</tr>
<tr>
<td>Abbreviation</td>
<td>Description</td>
</tr>
<tr>
<td>--------------</td>
<td>-------------</td>
</tr>
<tr>
<td>SA</td>
<td>Spalart Allmaras</td>
</tr>
<tr>
<td>SIMPLE</td>
<td>Semi-Implicit Method for Pressure Linked Equations</td>
</tr>
<tr>
<td>SVV</td>
<td>Switched Vortex Valve</td>
</tr>
<tr>
<td>tsfc</td>
<td>thrust specific fuel consumption</td>
</tr>
<tr>
<td>TDR</td>
<td>Turn Down Ratio</td>
</tr>
<tr>
<td>TET</td>
<td>Turbine Entry Temperature</td>
</tr>
<tr>
<td>TFSUTC</td>
<td>Thermo-Fluid Systems University Technology Centre</td>
</tr>
<tr>
<td>URANS</td>
<td>Unsteady Reynolds Averaged Navier Stokes</td>
</tr>
<tr>
<td>URSM</td>
<td>Unsteady Reynolds Stress Model</td>
</tr>
</tbody>
</table>
## Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Name</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>Amplitude of vibration</td>
<td>m</td>
</tr>
<tr>
<td>$a$</td>
<td>acceleration magnitude</td>
<td>$m/s^2$</td>
</tr>
<tr>
<td>$a_{ij}$</td>
<td>Reynolds stress components</td>
<td>Pa</td>
</tr>
<tr>
<td>$A$</td>
<td>Area</td>
<td>$m^2$</td>
</tr>
<tr>
<td>$C_1$</td>
<td>constant in the RSM model, =1.8</td>
<td></td>
</tr>
<tr>
<td>$C_2$</td>
<td>constant in the RSM model, =0.6</td>
<td></td>
</tr>
<tr>
<td>$C_\mu$</td>
<td>constant in the $k$-$\epsilon$ model, =0.09</td>
<td></td>
</tr>
<tr>
<td>$C_{prod}$</td>
<td>constant in SA model, =2</td>
<td></td>
</tr>
<tr>
<td>$c_{b1}$</td>
<td>constant in SA model, =0.62</td>
<td></td>
</tr>
<tr>
<td>$C_{\epsilon_1}$</td>
<td>constant in $k$-$\epsilon$ model, =1.44</td>
<td></td>
</tr>
<tr>
<td>$C_{\epsilon_2}$</td>
<td>constant in $k$-$\epsilon$ model, =1.92</td>
<td></td>
</tr>
<tr>
<td>$c_{w1}$</td>
<td>constant in SA model</td>
<td></td>
</tr>
<tr>
<td>$c_{w2}$</td>
<td>constant in SA model, =0.3</td>
<td></td>
</tr>
<tr>
<td>$c_{w3}$</td>
<td>constant in SA model, =2</td>
<td></td>
</tr>
<tr>
<td>$D$</td>
<td>Diameter</td>
<td>m</td>
</tr>
<tr>
<td>$E$</td>
<td>Total pressure difference</td>
<td>Pa</td>
</tr>
<tr>
<td>$e$</td>
<td>specific energy</td>
<td>$J/kg$</td>
</tr>
<tr>
<td>$F_b$</td>
<td>body force</td>
<td></td>
</tr>
<tr>
<td>$f_{v1}$</td>
<td>viscous damping function in SA model</td>
<td></td>
</tr>
<tr>
<td>$f_{v2}$</td>
<td>scalar function in SA model</td>
<td></td>
</tr>
<tr>
<td>$f_w$</td>
<td>damping function in the SA model</td>
<td></td>
</tr>
<tr>
<td>$g$</td>
<td>function in SA model</td>
<td></td>
</tr>
</tbody>
</table>
Symbols

$I$  turbulence intensity
$J$  ratio of momentum flux
$L$  length of the device,  $m$
$L_m$  length scale of mean flow  $m$
$l$  turbulent length scale  $m$
$\dot{m}_{\text{expt}}$  mass flow rate of air observed during experiment  $kg/s$
$\dot{m}_{\text{non}}$  non-dimensional mass flow rate ($\dot{m}_{\text{expt}}/\dot{m}_{\text{ref}}$)  $kg/s$ for high flow case
$\dot{m}_{\text{ref}}$  mass flow rate of air measured for pressure ratio 2.48 for high flow case  $kg/s$
$M$  Viscous moment  $N\cdot m$
$N$  Number of nodes
$n$  number of time-steps
$p$  Turbulent kinetic energy production  $m^2/s^3$
$p$  fluid static pressure  $Pa$
$P$  fluid total pressure  $Pa$
$Q$  volumetric flow rate  $m^3/s$
$Q_{C+}$  control flow rate through on side control port  $m^3/s$
$Q_{C-}$  control flow rate through off side control port  $m^3/s$
$Q_{O+}$  Onside outlet volumetric flow rate  $m^3/s$
$Q_s$  supply volumetric flow rate  $m^3/s$
$r$  radial distance  $m$
$r_a$  inner radius of vortex chamber  $m$
$r_b$  outer radius of vortex chamber  $m$
$r'$  non-dimensional distance in the SA model  $m$
$R$  radius of vortex chamber  $m$
$S_{ij}$  deformation tensor  $s^{-1}$
$s$  width of the vortex chamber  $m$
$T$  static temperature  $k$
$T_f$  through flow time  $s$
$t$  simulation time  $s$
$u_i$  instantaneous velocity component in Cartesian components  $m/s$
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$u_i$</td>
<td>time averaged velocity component in Cartesian components</td>
<td>$m/s$</td>
</tr>
<tr>
<td>$u'_i$</td>
<td>velocity fluctuation component in Cartesian components</td>
<td>$m/s$</td>
</tr>
<tr>
<td>$u_r$</td>
<td>radial velocity magnitude</td>
<td>$m/s$</td>
</tr>
<tr>
<td>$u_t$</td>
<td>turbulent velocity scale</td>
<td>$m/s$</td>
</tr>
<tr>
<td>$u_*$</td>
<td>frictional velocity</td>
<td>$m/s$</td>
</tr>
<tr>
<td>$U$</td>
<td>velocity magnitude</td>
<td>$m/s$</td>
</tr>
<tr>
<td>$U_\theta$</td>
<td>swirl velocity magnitude</td>
<td>$m/s$</td>
</tr>
<tr>
<td>$U_m$</td>
<td>mean velocity scale</td>
<td>$m/s$</td>
</tr>
<tr>
<td>$v$</td>
<td>turbulent velocity scale</td>
<td>$m/s$</td>
</tr>
<tr>
<td>$y$</td>
<td>distance normal to the wall</td>
<td>$m$</td>
</tr>
<tr>
<td>$y_p$</td>
<td>distance between the wall and node</td>
<td>$m$</td>
</tr>
<tr>
<td>$W$</td>
<td>width of the vortex chamber</td>
<td>$m$</td>
</tr>
<tr>
<td>$Z$</td>
<td>height of the device</td>
<td>$m$</td>
</tr>
<tr>
<td>$x, y, z$</td>
<td>Cartesian coordinates</td>
<td></td>
</tr>
</tbody>
</table>

**Greek Symbols**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\Delta e_Y$</td>
<td>total pressure drop between on-side and off-side outlet</td>
<td>Pa</td>
</tr>
<tr>
<td>$\Delta e_S$</td>
<td>total pressure drop between supply and off-side outlet</td>
<td>Pa</td>
</tr>
<tr>
<td>$\Delta p$</td>
<td>static pressure drop</td>
<td>Pa</td>
</tr>
<tr>
<td>$\Delta P$</td>
<td>total pressure drop</td>
<td>Pa</td>
</tr>
<tr>
<td>$\Delta y^+$</td>
<td>non-dimensional wall distance ($\rho u_*/\mu$)</td>
<td></td>
</tr>
<tr>
<td>$\delta_{ij}$</td>
<td>Kronecker delta function</td>
<td></td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>turbulent dissipation rate</td>
<td>$m^2/s^3$</td>
</tr>
<tr>
<td>$\kappa$</td>
<td>Von Karman constant</td>
<td></td>
</tr>
<tr>
<td>$\eta$</td>
<td>Kolmogorov length scale</td>
<td>$m$</td>
</tr>
<tr>
<td>$\eta_Y$</td>
<td>non-dimensional specific energy difference</td>
<td>$m^2/s^2$</td>
</tr>
<tr>
<td>$K$</td>
<td>mean kinetic energy</td>
<td>$m^2/s^2$</td>
</tr>
<tr>
<td>$k$</td>
<td>turbulent kinetic energy</td>
<td>$m^2/s^2$</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
<td>Unit</td>
</tr>
<tr>
<td>--------</td>
<td>-----------------------------------------------------------------------------</td>
<td>---------------</td>
</tr>
<tr>
<td>( \mu )</td>
<td>dynamic viscosity</td>
<td>Pa-s</td>
</tr>
<tr>
<td>( \mu_t )</td>
<td>turbulent eddy viscosity</td>
<td>Pa-s</td>
</tr>
<tr>
<td>( \mu_t, 2l )</td>
<td>two layer dynamic turbulent viscosity</td>
<td>Pa-s</td>
</tr>
<tr>
<td>( \mu_Y )</td>
<td>non-dimensional mass flow rate</td>
<td></td>
</tr>
<tr>
<td>( \nu )</td>
<td>dynamic viscosity</td>
<td></td>
</tr>
<tr>
<td>( \tilde{\nu} )</td>
<td>the Spalart eddy viscosity variable</td>
<td>( m^2/s )</td>
</tr>
<tr>
<td>( \rho )</td>
<td>density</td>
<td>( kg/m^3 )</td>
</tr>
<tr>
<td>( \sigma_k )</td>
<td>constant in ( k-\epsilon ) model, =1</td>
<td></td>
</tr>
<tr>
<td>( \sigma_{\epsilon} )</td>
<td>constant in ( k-\epsilon ) model, =1.3</td>
<td></td>
</tr>
<tr>
<td>( \sigma_{ij} )</td>
<td>stress tensor in Cartesian coordinate</td>
<td>Pa</td>
</tr>
<tr>
<td>( \bar{\sigma}_{ij} )</td>
<td>time averaged stress tensor in Cartesian coordinate</td>
<td>Pa</td>
</tr>
<tr>
<td>( \tau_\eta )</td>
<td>Kolmogorov time scale</td>
<td>s</td>
</tr>
<tr>
<td>( \tau_w )</td>
<td>wall shear stress</td>
<td>Pa</td>
</tr>
<tr>
<td>( \tau_{ij} )</td>
<td>Reynolds stress tensor</td>
<td>Pa</td>
</tr>
<tr>
<td>( \phi_{ij} )</td>
<td>pressure strain term of the RSM</td>
<td></td>
</tr>
<tr>
<td>( \chi )</td>
<td>ratio of Spalart variable to kinematic viscosity</td>
<td></td>
</tr>
<tr>
<td>( \omega )</td>
<td>angular frequency</td>
<td>( rad \ s^{-1} )</td>
</tr>
<tr>
<td>( \Omega_{ij} )</td>
<td>mean vorticity tensor</td>
<td>( s^{-1} )</td>
</tr>
</tbody>
</table>

**Subscripts**

<table>
<thead>
<tr>
<th>Subscript</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( a )</td>
<td>values at the inner radius of the cavity</td>
</tr>
<tr>
<td>( b )</td>
<td>values at the outer radius of the cavity</td>
</tr>
<tr>
<td>( c )</td>
<td>values at the end of the control port of SVV</td>
</tr>
<tr>
<td>( co )</td>
<td>difference between values at control port and outlet of the vortex amplifier</td>
</tr>
<tr>
<td>( cs )</td>
<td>time averaged value at the control port</td>
</tr>
<tr>
<td>( n )</td>
<td>values at the end of the SVV nozzle</td>
</tr>
<tr>
<td>( in )</td>
<td>value at the inlet</td>
</tr>
<tr>
<td>( out )</td>
<td>value at the outlet</td>
</tr>
<tr>
<td>( p )</td>
<td>values in the near wall cell centroid</td>
</tr>
<tr>
<td>( ref )</td>
<td>reference value</td>
</tr>
</tbody>
</table>
Symbols

\[ r \] radial coordinate
\[ s \] values at the inlet of the vortex chamber
\[ s_0 \] difference between values at inlet and outlet of the vortex amplifier
\[ wall \] values at the wall of the vortex chamber \((r/R=1)\)
\[ \phi \] tangential coordinate
\[ i, j, k \] subscripts for Einstein’s tensor notation
to my beloved wife . . .

_Sneha_
Chapter 1

Introduction

One of the important tasks of a secondary air system of an aero-engine is to supply the cooling air for different engine components. The quantity of cooling air to be supplied depends upon the operating point in the flight envelope. At take-off the critical components of the engine such as high pressure turbine (HPT) and intermediate pressure turbine (IPT) blades are subjected to high temperatures and high temperature gradients which may result in thermal fatigue. These detrimental effects may lead to loss of blade integrity. Hence, in order to prevent this undesired scenario, a large amount of cooling air is supplied to cool the blades. This cooling air is bled from main annulus flow at different locations in the compressor.

In contrast to take-off, in the cruise flight phase, the engine operating conditions reach a quasi-steady state. In this state the components are at relatively low temperatures and temperature gradients are also reduced hence less cooling air is required compared to the take-off phase. The passages carrying the cooling air are designed for the case of maximum cooling air flow, which is at take-off state. So, in the cruise phase more cooling air is being supplied than is required for cooling at this condition.
The above problem presents an opportunity to introduce a new sub system in secondary air systems so that the cooling air can be modulated in order to save costly compressed air.

1.1 The need for blade cooling

In figure 1.1 shows the history of turbine entry temperature (TET) achieved by Rolls-Royce over the period of 70 years. Over the last few decades engine designers have strived to achieve higher TET. A simple thermodynamic analysis of the Brayton cycle reveals that higher TET leads to higher thermal efficiency. However, TET is limited due to the material properties of turbine blades and the surrounding components.

Figure 1.1 it is evident that up to 1960, the TET was limited due to the limitations of thermal properties of material used for manufacturing turbine blades. However, with introduction of turbine blade cooling in early 1960, it was possible to achieve the TET higher than the limiting material temperature. The introduction of blade cooling directly benefited engine performance. Since then lot of research has been focused on enhancing the effectiveness of blade cooling resulting in increase of temperature difference between operating gas and material limits.

1.1.1 Functions of secondary air systems

Apart from supplying cooling air to turbine blades and the surrounding components, the secondary air system performs other important functions as well. These include pressurization of the aircraft cabin, prevention of hot gas ingestion into turbine disc cavities, bearing chamber sealing and cooling of other important components such as the combustor and high pressure compressor (HPC).

Figure 1.2 shows part of a typical secondary air system of a Rolls-Royce engine. A typical Trent family engine comprises 1 fan, an 8 stage intermediate pressure
Chapter 1. Introduction

Figure 1.1: Turbine entry temperature for Rolls-Royce engines since 1940 (Cumpsty (2008))

compressor (IPC), a 6 stage HPC, a single stage HPT, a single stage IPT and a 6 stage low pressure turbine (LPT). Figure 1.2 only shows the last 5 stages of the HPC, the HPT and the IPT. The figure also shows three different air circuits represented by different colors.

The first circuit, called HP3 air is shown in magenta. The air is extracted from the third stage of the HPC, as shown in Figure 1.2. This extracted air then delivered to the intermediate pressure nozzle guided vanes (IPNGV) through six pipes. The delivered air performs several important functions such as cooling the front part of the IPT blades, sealing of the gap between the rear side of IPNGV and the front of the IPT blades and sealing the gap between back of the HPT blades and front platform of the IPNGV blades. The blue S-shaped arrow shown in Figure 1.2 represents the location of seals through which HP3 air leaks and prevents ingestion of hot gases into the HPT and IPT drums. This air circuit is the prime focus of the research presented in this thesis.

The second air circuit, called HP6 air is shown in red. In this circuit the main annulus air flow is divided into three parts, where each part is used for distinct
purposes as shown in Figure 1.2. The green arrow represents the air flow which passes through and around the combustor, where it is used for burning the fuel. The orange arrow represents the air flow which is used for cooling of the high pressure nozzle guided vanes (HPNGV) and HPT blades. This air is also used to prevent the hot gas ingestion by pressurizing the rim seal between HPNGV and HPT blades. The blue arrow, represents the part of main annulus flow which is used for cabin pressurization.

The third air circuit called IP8 air is shown in yellow. The cooling air is drawn from the eighth stage of IPC (not available in Figure 1.2). This air is used to cool the HPC discs. After passing through the HPC this air is also used to pressurize the LPT drum in order to prevent hot gas ingestion.
1.2 Cooling air modulation

The cooling air delivered by the secondary air system depends upon several parameters such as operating point of flight, ambient conditions and the turbine material capabilities. At the maximum power condition which occurs at take-off, higher cooling air flow is required. The flow rate is fixed by the size of the restricting orifices through which the flow passes. The sizing of these orifices is designed such that the amount of cooling air passed is sufficient to cool the turbine and other components at an acceptable level. At engine power ratings less than the maximum power condition the TET reduces and hence less cooling air is required. Cruise conditions might occur at an altitude of roughly 9200 m, where the atmospheric temperature is around -50°C, the lower inlet temperature and lower power requirement results in lower TET. Hence less cooling air is required than at take-off. Since the orifices are designed for maximum air flow, they supply more cooling air than required for the lower power flight conditions such as cruise.

Modulation of the cooling air, reducing the flow at low power conditions, could give significant benefits in engine performance. Fluidic devices, which have no mechanical parts, have been proposed for this. A 1D network flow analysis of a secondary air system with fluidic devices has indicated reductions in specific fuel consumption around 0.15% (Miniscoulx (2009)). With the introduction of fluidic devices, cooling air flow could be optimized resulting in direct saving of the power required to compress the excess cooling air which is subsequently expanded without doing any useful work.

Fluidic devices are non-moving devices and require no maintenance. Due to absence of any moving part the introduction of fluidic device will not affect the reliability of whole engine system. Hence, considering the potential increase in engine performance, fluidic devices present an attractive method for air modulation compared to conventional valve systems.
The basic concept and working mechanism of the fluidic device considered in this thesis is thoroughly described and explained in chapter 3.

1.3 Problem definition

Considering the benefits offered by fluidic devices compared to conventional flow control methods, Rolls-Royce has recognized fluidic devices as a novel method for modulation of cooling air in aeroengines. Research began with construction of a test rig at the University of Sheffield. A series of experiments were conducted with a ‘Switched vortex valve (SVV)’. The purposes of the experiments were to determine the performance characteristics of this device varying the operating pressure ratio and to examine the stability of the device.

In spite of the broad range of experiments conducted on the SVV, only limited understanding of the flow inside the device was achieved. A clearer understanding of the flow field in the device might be used to optimize the device performance by reducing the aerodynamic losses and in turn making it more efficient. However, due to size and other constraints, no efforts were made to visualize the flow during experiments. This presents the opportunity for application of CFD methodology in order to establish a clear picture of flow behavior inside the device and identify critical flow structure which affects the performance.

A typical Rolls-Royce Trent engine comprises 3 rotors, namely low pressure (LP) rotor, intermediate pressure (IP) rotor and high pressure (HP) rotor. These rotate at different speeds, generating vibration in the engine over a wide range of frequencies. For engine certification all components and subsystems of an engine have to withstand the vibration generated by rotors without compromising performance. Hence, before introducing any new system or when redesigning existing engine systems vibration tests must be conducted. However, these verifications are costly and time consuming due to the complexities of experiment. A further
operational and certification issue is the effect of sand ingestion, which is common for certain engine operators. Any flow control device must be able to withstand such conditions.

Keeping the limitations of the experiments in mind, the broad objective of the present research was defined as development and evaluation of CFD methodology to gain insight of flow behavior occurring in the device. Scanlon et al. (2009) performed CFD analysis for the SVV and reported considerable differences between predicted and measured data. The device considered in their work is similar to the SVV considered in this research but different in dimensions. For the flow predictions, Scanlon et al. (2009) used Reynolds Averaged Navier-Stokes modeling (RANS) with the $k$-$\epsilon$ turbulence model. The accuracy of turbulence models is highly problem dependent, due to their method of formulation. Hence, it is necessary to evaluate the performance of different models before applying CFD as alternative tool to experiments.

1.4 Objectives of the present research

The major focus of this thesis is on the evaluation and application of state of art CFD methodology for flow analysis of SVVs. Chapter 3 reveals that flow prediction in SVVs is a challenging task since it involves flow separation and highly swirling flow in the vortex chamber. From the modeling perspective, the device can be divided into two different flow problems, namely a flow switching device and the swirling flow occurring in vortex chamber. Hence, it was decided to evaluate CFD for a flow switching device and vortex chamber separately before commencing studies on the full SVV. This lead to the following objectives for the research

1. Assessment of CFD for prediction of flow in a flow switching device and for the swirling flows of the vortex chamber.
2. Assessment of CFD for prediction of flow in an SVV using turbulence models examined in the previous study.

3. Application CFD to study the fluid dynamics of the device including such effects as vibration and sand ingestion.

4. Application of CFD to study the stability of the device for proposed changes in geometry.

1.5 Outline of the thesis

Each chapter of this thesis covers a distinct part of the research carried out on the SVV. Chapter 2 reviews previous research carried out on fluidic devices. This chapter also considers the scope of applications of vortex amplifiers and other relevant fluidic devices and explanation of methods of quantifying the performance of fluidic devices. Chapter 3 briefs describes the flow governing equations and the modeling of turbulence equations. These models include the SA, $k-\varepsilon$ and the RSM.

Chapters 4 and 5 present results of CFD analysis of a vortex chamber and a flow switching device. Both studies include comparison of computed results with experimental data. Chapter 6 presents the most important part of the thesis. It reports the evaluation of CFD predictions for the SVV device for different states of operation at different pressure ratios. The comparison of numerical prediction with experimental data is encouraging. With the confidence in CFD predictions generated by this study, CFD investigations are then extended to further problems.

Chapter 7 presents the numerical work carried to understand the dynamics of flow switching. In this the simulation were focused to understand the effect of control port pressure on the flow switching characteristics of the device. Chapter 8 presents the CFD investigation of effect of vibration on the performance of the
device. The vibration study reported in this chapter consists of vibration with several definite frequencies.

Chapter 9 presents the conclusion of the thesis and recommendation for future work.
Chapter 2

Literature Review

Fluidics technology provides sensing and controlling functions with fluid power employing the general fluid phenomenon of wall attachments and fluid stream interactions. According to Humphrey & Tarumato (1965) fluidics was discovered by group of scientists at the US Army Diamond Ordnance Fuze Laboratories in Washington, DC, who introduced fluidic devices as counterparts to analog components used for control under harsh nuclear radiation environments. It was realized that using the same principle the technology could be utilized in several other flow control areas specially in process industries. Today the same technology of controlling the fluid flow using fluidic device is known as ‘Power Fluidic’.

Fluidic devices have captured the imagination of engineers who were attracted by the concept of control devices without moving parts, resulting in high reliability and longer life of systems. Since the inception of fluidic devices, researchers have found wide areas of applications. An extensive review of areas of application of fluidic devices can be found in dedicated references such as Priestman & Tippets (1984) and Weathers (1972).
The major benefit of using fluidic devices for flow control is that they are fail-safe when compared to their counterparts such as actuated valves. This characteristics of fluidic devices favours their application where reliability of a system is paramount. In their early introduction to process industry, many applications have been sought in the nuclear sector, as listed by Zabsky et al. (1970), Grant & Marshall (1975), Tippets et al. (1981) and Francis et al. (2011). The literature reveals that the primary motive of using fluidic devices in nuclear plants is to reduce the human intervention in hazardous areas and replace existing valve flow control systems with reliable fluidic systems. Ranade et al. (2009) and Parker et al. (2011) proposed the use of fluidic devices for radioactive liquid transfer. Tesar (2005) and Tesar (2011) introduced fluidic pumps which would replace conventional pumps.

Fluidic devices have no-moving parts and hence require very little maintenance. Hence, the devices are robust in design and can withstand very harsh operating conditions. This feature of fluidic devices gives an edge over mechanical systems. After the introduction of fluidic devices in the 1960’s researchers proposed and validated use of fluidic devices for flow measurements. The research carried out by Tippets et al. (1973) and Prokopius (1973) showed the feasibility of application of fluidic devices for flow measurement. Tippets et al. (1973) developed and experimentally tested a fluidic device which measures the volumetric flow rate of fluid by using the frequency of oscillations of fluid. Prokopius (1973) extended the concept of flow measurements of single fluid to binary gas mixture. The fluidic device developed in this study was of bistable fluid amplifier category and it successfully measured the gas mixture flow rate within 2% of test flow. The application of fluidic devices has progressed to many fields, for example Wang et al. (1996) and Wang et al. (1997) experiment with a device to operate in the harsh conditions of an oil well to measure the crude oil production.

Due to the ability of fluidic devices to withstand high vibration and high temperature they found different areas of application in aero-engines. Rimmer (1970)
envisaged replacing the conventional pressure traducers of an aero-engine electrical system with fluidic devices. A fluidic device was designed and successfully integrated with the engine control system. Davies (1970) took the same concept one further step designing a control circuit for inlet guide vanes using a fluidic device. Use of fluidic devices in aero-engines was conceptually extended to a new design of combustor by Woolhouse et al. (1998) and Brundish et al. (1999). Chen et al. (1998) experimented with fluidic devices in the fuel injection system of a gas turbine. The new fuel injection system was tested against a conventional system with solenoid valves. The experiment demonstrated that a fluidic gas injector was able to achieve better performance.

The concept of fluidic device was also proposed in several other areas. Scanlon (1997) developed and experimented with a fluidic device for spark ignition engines with the aim of improving fuel air mixture and thus improving the combustion. Dustin & Wauhagen (1969) successfully demonstrated a fluidic design for replacing an electric operated stepper motor. Priestman & Tippets (2000) experimented with a fluidic system for controlling fluid level in gas-fluid separator.

From the review of fluidic devices it can be concluded that they have been proposed and tested for various applications. The basic features of fluidic devices are encouraging. These features include high reliability due to no moving parts, low maintenance, and the ability to work in harsh and hazardous operating conditions. Although there are number of fluidic devices have been developed, this thesis focuses on vortex amplifiers and jet interactions. These are the focus of this research, and are discussed in in section 2.1 ans 2.2.

## 2.1 Vortex amplifiers

The vortex amplifier has been developed in various forms to suit different applications. This is illustrated in Figure 2.1. The vortex amplifier is basically a
Chapter 2. Literature

geometrically modified confined vortex chamber with a radial passage for main supply flow. A control port provides passage for control flow which interacts with the main supply flow. Depending upon the interaction between supply flows, two different flow states occur in the vortex amplifier, namely the high flow state and the low flow state.

![Schematic of a typical vortex amplifier](image1)

**Figure 2.1:** Schematic of a typical vortex amplifier (from Humphrey & Tarumato (1965))

In the high flow state, high mass flow rate of operating fluid is expected at the outlet of the device. This state of the device can be explained in detail with the help of Figure 2.2 (A). This state is characterized by absence flow at the control port. The main supply flow enters radially and leaves through the outlet of the device with minimum resistance.

The low flow state is characterized by the presence of control flow at the control port. The control flow enters the device tangentially and deflects the main supply flow path from its radial pattern to a spiral system as shown in Figure 2.2 (B). The subsequent formation of the vortex lengthens the flow path which eventually increases the pressure drop. This pressure drop can be attributed to two effects of
the vortex flow: the longer stream path and the acceleration of the stream as it gets near to the outlet of the device. Since the supply pressure is usually fixed, the larger pressure drop will result in a reduction of the main supply flow. Increasing the control flow rate imparts higher tangential velocity to the main supply flow at the outer radius of the vortex chamber resulting in higher pressure drop and more reduction in main supply flow.

![Figure 2.2: Different states of operation of a vortex amplifier (A) High flow state (B) Low flow state (from Humphrey & Tarumato (1965))](image)

### 2.1.1 Vortex amplifier characteristics

The performance of the vortex amplifier depends upon the geometry and the interaction of the fluid streams. The operation of the vortex amplifier can be described with the help of functional characteristics. Figure 2.3 shows a circuit symbol of a vortex amplifier.

Tippets (1984) pointed out that the performance of the vortex amplifier can be quantified using following four flow variables

1. $\Delta p_s (= p_{so} - p_o)$ is the static pressure drop between device inlet to outlet
2. $\Delta p_c (= p_{co} - p_o)$ is the static pressure drop between device control port to outlet
3. $Q_s$ is the volumetric flow rate of operating fluid at the inlet of the device.

4. $Q_c$ is the volumetric flow rate of operating fluid at the control port of the device.

King (1985) suggested that the above four flow variables can be reduced to two non-dimensional numbers which are sufficient to describe the performance of the device. These are Control Pressure Ratio (CPR) and Turn Down Ratio (TDR).

a) The CPR is defined as the ratio of the control pressure drop ($\Delta p_c$) to the supply pressure drop ($\Delta p_s$) required to completely restrict the main supply flow to $Q_s = 0$. It represents the effectiveness of the control flow.

b) The TDR is defined as the ratio of the maximum main supply flow rate (when $Q_c = 0$) to the maximum control flow (when $Q_s = 0$) for a constant supply to downstream pressure ratio.

Generally, the designer is looking for high TDR or low CPR, or both, depending upon the application. Higher TDR signifies that low control flow rate is required to switch the device from high flow state to no flow state for a given main supply flow rate and lower CPR signifies that less control pressure is required to switch the device for a given supply pressure drop ($\Delta p_s$).
Chapter 2. Literature

The characteristic of a typical vortex amplifier are often given in the form shown in Figure 2.4 where the supply flow and the control flow are functions of the control port pressure. Generally, during the operations the supply and outlet pressures remain constant and only the control port pressure is varied to obtain the characteristic curve for a vortex amplifier. For any given supply to outlet pressure ratio the supply flow rate is maximum when there is no control flow. With increasing static pressure at the control port, supply flow decreases until it reduces to zero. In this way two unique flow states are achieved in the vortex amplifier.

![Figure 2.4: A vortex amplifier characteristics curve (From Priestman & Tippets (1984))](image)

Belsterling (1971) reported that the performance of the vortex amplifier depends upon a number of parameters. The ratio of vortex chamber to outlet diameters, aspect ratio of the vortex chamber and location of the control ports are some of the numerous factors which affect the vortex amplifier performance. King (1985) conducted comprehensive experimental work on the effects of the geometry on the performance of a vortex amplifier. Figure 2.5 shows a typical vortex amplifier with critical geometrical parameters used in the experiment by King (1985).
The important geometrical parameters varied in the experiment were $A_E$ area of exit throat, $A_S$ area of supply inlet, $A_T$ area of control port, $D_O$ vortex chamber outer diameter, $D_E$ exit throat diameter and $H$ Chamber height. Various geometrical configurations were tested and performance was evaluated in terms of TDR, CPR and Performance Index (PI), which is defined as the ratio of TDR to CPR. The experiments concluded that there are four important geometrical parameters which effect the performance. Some observations from experiments and the list of four geometrical parameters are summarized as follows

1. The most significant geometrical parameter affecting the performance of a vortex amplifier was $A_E/A_T$. It was found that the device showed maximum value of Performance Index for values of $A_E/A_T$ between 2 to 5.

2. Optimizing the ratio $D_O/D_E$ was found useful in improving the performance of the vortex amplifier for a given value of $A_E/A_T$.

3. The device showed best performance when the values of $H/D_E$ were between 0.2 to 0.3 for any given $A_E/A_T$ ratio.
4. It was found that the ratio of $A_S/A_T$ has to sit between 3 to 4 in order to achieve optimum performance. Outside these values the performance starts to deteriorate.

In order to enhance the performance of the vortex amplifier, it is necessary to understand the flow structures occurring in device. Smoke visualization of airflow on a transparent model has been used to study interaction between the main supply flow and the control flow. However, due to the size constraints the smoke visualization finds limited use. In general, in order to optimize the performance of the device a large number of experiments are needed which are time consuming and costly in nature. Recognizing the limitations of experiments, some researchers have focused their attention on application of CFD to understand the flow behavior in the device.

Woolhouse et al. (2001) performed CFD studies on a vortex amplifier using $k-\epsilon$ and RSM turbulence models. The comparison of numerical results with experimental data showed that the $k-\epsilon$ model failed to predict the high swirl cases. However, the RSM model showed good comparison with experimental data for high swirl cases with agreement within 1-2% of measured data. In order to reduce the computational cost, all simulations in this study were run assuming steady state solutions. The authors noted that the unsteadiness of flow field prevented convergence of the solution and hence recommended unsteady solution approaches for future studies. The superiority of the RSM model over the $k-\epsilon$ model for highly swirling flows has also been reported in other publications such as, Kumar et al. (2012) and Jawarneh & Vatistas (2006).

Parker et al. (2011) used CFD to understand the effects of changes of geometry on the performance of the vortex amplifier. Considering the recommendations made by Woolhouse et al. (2001), only a RSM was used to model turbulence. The numerically predicted mass flow rate showed good agreement with the measured mass
flow rate. Considering the confidence in numerical predictions, the CFD studies were extended to optimize the geometry configuration for better performance of the device.

### 2.2 Wall attached fluidic devices

The wall attachment devices are basically flow switching devices utilizing the Coanda effect. Figure 2.6 shows a typical wall attached fluidic device with its critical components. The basic objective of the device is to achieve switching of the flow from one stream to another without using conventional flow control devices. A typical wall attached flow switching device requires a main supply flow at the input of the device. The fluid passes through a restricted channel or nozzle converting pressure energy into kinetic energy. The resulting jet stream then enters the interaction region and passes through a preferred output stream. Flow through preferred outlet can be achieved by asymmetric geometric features as shown in Figure 2.7 or by pressure differential at the outlets. In order to switch the flow to another stream, control flow is added through control ports. The impact of control flow results in deflection of the main supply jet stream in the interaction region causing it to switch to the other output stream.

Wall attached fluidic devices are classified as bistable fluidic devices due to their ability to switch flow from one stable flow state to another stable flow state. The device remains in one particular state unless the flow state is changed through the control flow. The bistable characteristics of the device can be explained though Figure 2.8. After the start up of the device the output flow is received at one of the outlets as shown Figure 2.8(a). In this state there is no control flow and amount of flow flowing through the device depends upon the pressure differential between inlet and outlet. A control flow is injected through a control port, in order to switch the flow to another stream as shown in Figure 2.8(b). Once the
Figure 2.6: A typical wall attached fluidic device showing critical components
(From Belsterling (1971))

Figure 2.7: Different size of steps results in preferred flow direction

flow has switched, the device no longer needs control flow since the flow remains in the stable switched state. By applying the control flow at other control port the device can be switched back to its original state as shown in Figure 2.8(d).

2.2.1 Load characteristics curves

The performance of the wall attached fluidic device is described using load characteristic curves. Load characteristics curves were introduced by Tippets & Royle
Figure 2.8: Bistable characteristic of a wall attached fluidic device

(1971) to compare and characterize different device configurations for circuit design. Tippets & Royle (1971) described load characteristics curves as a graphical representation of performance of the device.

Figure 2.9 shows circuit symbols used by Tippets & Royle (1971) to describe a typical wall attached fluidic device. The ‘on side’ outlet shown in the figure indicates that the supply flow is biased to pass through the lower output unless it is switched to the other stream by control pressures. They argued that the performance of the device can be measured in terms of eight flow variables which are $Q_S$ the main supply flow rate at the inlet of the device, $Q_C^+$ the control flow rate through on side control port, $Q_C^-$ the control flow rate through off side control port, $Q_O^+$ the flow rate at the on side outlet, $E_S$ the specific energy difference between inlet and off-side outlet ($E_S = P_S - P_O^-$), $E_C^+$ the energy difference between on side control port and off-side outlet ($E_C^+ = P_C^+ - P_O^-$), $E_C^-$ the energy difference between off side control port and off side outlet ($E_C^- = P_C^- - P_O^-$) and $E_C^-$ the energy difference between on side outlet and off side outlet ($E_C^- = P_O^+ - P_O^-$).
Figure 2.9: Circuit symbols of a typical wall attached fluidic device

Figure 2.10 shows typical output characteristic curves of a wall attached fluidic device for constant supply flow \( Q_S = q \) and without control flows \( Q_{C^+} = Q_{C^-} = 0 \). The basic objective of the output characteristics is to represent the stable states of the device and to determine whether the device will direct the flow as required. Tippets & Royle (1971), suggested that the output characteristics are obtained by keeping three flow variables constant (In this case \( Q_S, Q_{C^+} \) and \( Q_{C^-} \)) and plotting pressure variables while allowing the fourth flow \( Q_{O^+} \) to vary. At point ‘X’ the supply flow rate matches the on side outlet flow. The on side outlet flow can be varied from a large positive value at ‘A’ to a negative value at ‘B’. At point ‘A’ where \( Q_{O^+} \) is higher than the supplied ‘q’ which is due to the reverse flow occurring at the off side outlet and at point ‘B’ where \( Q_{O^+} \) is negative due to the reverse flow at on side outlet.

Tesař (2010) performed several experiments with a wall attached fluidic device without control ports. The objective of the experiments was to understand the effect of splitter configuration on the performance of the device. Figure 2.11 shows a typical fluidic device with a round shape splitter. Apart from the round splitter, the device was tested with cup shaped and wedge shaped splitter designs.
The experimental results are plotted in terms of a load characteristic, as shown in Figure 2.12. The load characteristic curves represents the dependence of the flow through the on side outlet with respect to the specific energy difference across the device. The mass flow rate at the on side outlet is non-dimensionalised by dividing by the supply mass flow rate

$$\mu_Y = \frac{Q_{O^+}}{Q_S}$$

and, in a similar manner, the output specific-energy difference ($\Delta e_Y$) is non-dimensionalised by being related to the supply specific-energy difference ($\Delta e_S$)
\[ \eta_Y = \frac{\Delta e_Y}{\Delta e_S} = \frac{P_O^+ - P_O^-}{P_S - P_O^-} \quad (2.2) \]

**Figure 2.12:** A typical load characteristic curve of a device experimented by Tesař (2010)

The importance of the load characteristic curve can be explained with the help of Figure 2.12. The point ‘B’ on the curve shown in Figure 2.12 represents that all the supply flow through the inlet passes through the on side outlet. By applying mechanical blockage to the on side outlet flow, the main flow starts to divert to the off side outlet. The point ‘A’ represents the flow state where any further blockage shifts the entire supply flow from the on side to the off side. In this way, the load characteristic curve identifies the point of switching and the specific energy differences required to switch the flow.

Feikema & Culley (2008) performed several experiments to examine the switching characteristics of the jet controlled wall attached fluidic device shown in Figure 2.13. The experiments were performed to develop an understanding of the effect of the control port velocity and the nozzle velocity on the switching characteristics. In order to switch the flow from one state to another, control flow was
introduced to the device through the control port. The control flow was intermit-
tent in nature and controlled by solenoid valves. It was concluded that the ratio
of the control flow velocity to the nozzle flow velocity ratio must be in the range
of 0.2 to 0.3 in order to achieve effective switching.

![Diagram of jet controlled wall attached fluidic device](image)

**Figure 2.13:** A section of jet controlled wall attached fluidic device Feikema
& Culley (2008)

Along with the experiments, Feikema & Culley (2008) also conducted a numerical
study to compare with the experimental observations. The experimental model
was approximated as a 2D geometry and the shear stress transport turbulence
model was used in the numerical study. The numerical solutions were used to
help understand the complex time dependent flow interaction between the control
port and nozzle flow, which is difficult to analyse with experimental methods. The
numerical results were found to be in agreement with the experimental data. Both
experimental and numerical results showed that for the device considered in the
study, the square root of the ratio of momentum flux at nozzle to the momentum
flux at control port is the most important parameter and the value to achieve
switching was found to be approximately 0.25 for different cases.

Heo *et al.* (2010) also investigated the operation characteristics of a jet controlled
wall attached device. Both experimental and numerical studies were conducted
to investigate the area of control port and operation parameters such as the mass flow ratio of control flow and the main supply flow. The experimental work predominately concentrated on the determination of the response time required by the device to switch the flow by control flow.

Figure 2.14: Different configurations of the jet controlled wall attached fluidic device experimented by Heo et al. (2010)

A series of experiments were conducted with several configurations of a device, as shown in Figure 2.14. It was found that the flow can be switched in the device with a control port pressure ranging from 115 kPa to 140 kPa with the constant supply pressure of 300 kPa. The time required to switch the flow was measured and found to be less than 7 msec. It was also found that the shape of the control port has a distinct effect on the switching characteristics of the device. The experimental observations were simulated using the $k$-$\epsilon$ turbulence model. The numerically computed results were compared with the experimental data. The comparison showed that the predictions of supply flow and control flow are in good agreement with the measured data. The response time predicted by the numerical method was also found to be in good agreement with the measured values.
2.3 Conclusion

A detailed review of previously published experimental and computational work related to the wall attached fluidic devices and vortex amplifiers has been presented in this chapter. The wall attached fluidic devices and vortex amplifier are most relevant to the present study due to the following reasons:

1. The fluidic device envisaged in this study for flow modulation will basically work in two states, namely high and low flow state. These two states can be achieved using the device analogues to a vortex amplifier. The high flow state will experience less pressure resistance and in turn supply high flow. Conversely, low flow state will supply less flow rate compared to high flow due the higher resistance to the flow.

2. The transition from high to low state and vice versa can be achieved by flow switching as used in jet controlled wall attached fluidic devices.

Figure 2.15 shows the SVV device which is a combination of vortex chamber and flow switching device. This device is the primary subject of the research reported in this thesis. The details of this device can be found in the chapter 6.

![Figure 2.15: Schematic arrangement showing essential features of SVV used by Scanlon et al. (2009)](image)

Following conclusions can be drawn from the review carried on this chapter
1. Vortex amplifiers are capable of reducing the main supply flow from maximum flow rates to nearly zero mass flow rate. The performance of the vortex chamber is affected by the geometry parameters and certain combinations of parameters can be used to maximize the performance. Few numerical studies have been carried out to understand the flow structure inside the vortex amplifiers. Use of the two equation turbulence models such as the $k-\epsilon$ model result in large discrepancies with measured data. It was also noted that the unsteady calculations may be more realistic considering the observed unsteady flow behavior.

2. The switching characteristics of wall attached fluidic devices were reviewed. The review sheds light on the mechanism of flow switching. It was found that the flow switching can be achieved though applying large flow resistance or by deflecting the jet by control flow. Jet controlled wall attached devices seems more relevant to the present study since they are bistable in nature. Limited attempts have been carried out to predict the flow interaction between and main supply flow and control flow. The numerical results were found to be in good agreement with the experiments. However, very little information is available on the role of the turbulence model in predicting the dynamics of switching.
Chapter 3

Turbulence modeling and Numerical Methods

Computational Fluid Mechanics is considered as an interdisciplinary science with application in several engineering fields such as mechanical, aerospace, automotive, biomedical and civil where flow of fluid plays a crucial role. The most important feature of CFD is that it can be considered as alternative to experimental methods for understanding the flow features associated with engineering flow problems. Basically, CFD is a science of formulating and numerically solving governing equations for the flow in discretized algebraic form. These algebraic equations are solved to obtain the numerical values for the flow variables in space and time. This chapter focuses on the turbulence models used to discretize the governing equations and the numerical methods in the present investigation. It is included for completeness as these methods underpin the research presented in later chapters. Further information on CFD methods and turbulence modeling can be found in standard texts such as those by Versteeg & Malalasekera (2007), Patankar (1980), Pope (2000), Wilcox (2000) and FLUENT (2000).
Chapter 3. Turbulence modeling and Numerical Methods

3.1 Flow governing equations

All numerical methods associated with the CFD are built upon the fundamental governing equations of the fluid dynamics. These fundamental governing equations are the continuity equation based upon the conservation of mass, the momentum equations based upon the Newton’s second law of motion and the energy equation based upon the conservation of energy.

The following equation is a general form of continuity equation valid for incompressible and compressible flows expressed in Cartesian tensor notation

\[
\frac{\partial (\rho)}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0
\]  

(3.1)

where \( \rho \) is density of fluid, \( t \) is time and \( u_i \) with \( i = 1, 2 \) or \( 3 \) represents the Cartesian velocity components, \( x_i \) represents the Cartesian coordinates and \( x_1, x_2 \) and \( x_3 \) can be alternatively be denoted \( i, j \) can represent \( x, y, \) and \( z \).

The conservation of momentum equation is the application of Newton’s second law to the fluid flow. This is the Navier-Stokes equation, valid for incompressible, compressible, laminar and turbulent flows and is written as

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_i} = -\frac{\partial p}{\partial x_i} + \frac{\partial \sigma_{ij}}{\partial x_i} + F_{bi}
\]  

(3.2)

where, \( p \) is static pressure, \( F_{bi} \) is body force acting on the fluid and \( \tau_{ij} \) is a stress tensor given as

\[
\sigma_{ij} = \mu \left[ \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \Theta \right]
\]  

(3.3)

\( \mu \) is dynamic viscosity of the fluid, \( \delta_{ij} \) is a Kronecker delta function and \( \Theta \) is defined as
\[ \Theta = \frac{\partial u_i}{\partial x_i} \]  

Eqn 3.2 is non-linear due to the convection terms and the effects of non-linearity increase with the Reynolds number, resulting in formation of small instabilities in the flow. These instabilities result in the flow fluctuations known as turbulence. Hence, finding an exact solution for Eqn 3.2 is a challenging task for high Reynolds number flows. Only for simple flow problems, such as steady, incompressible, laminar fully developed flow in pipe, do exact solutions exist.

In many engineering applications the fluctuations due to the turbulence are approximated using time averaging process. Flow variables are expressed as the sum of mean and fluctuating components, so that

\[ \phi = \bar{\phi} + \phi' \]  

where, \( \phi \) is any flow variable, \( \bar{\phi} \) is the time averaged quantity of the flow variable and \( \phi' \) is the unresolved or fluctuating component of flow. By applying this approximation the continuity equation is approximated as below

\[ \frac{\partial (\rho \bar{\phi})}{\partial t} + \frac{\partial (\rho \bar{u}_i \phi')}{\partial x_i} = 0 \]  

3.2 Reynolds Averaged Navier-Stokes

Decomposing the flow variables in the Navier-Stokes equation using Eqn 3.5 and time averaging the following equation is obtained for the mean velocity.
\[
\frac{\partial (\rho \bar{u}_i)}{\partial t} + \frac{\partial}{\partial x_i} (\rho \bar{u}_i \bar{u}_j) = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_i} \left[ \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \mu \delta_{ij} \Theta \right] - \rho \bar{u}_i \bar{u}_j \quad (3.7)
\]

This is called the Reynolds averaged Navier-Stokes equation as it was first derived by Osborne Reynolds. A comparison of Eqn 3.2 and 3.7 reveals that the Reynolds averaging process results in new terms involving the averages of products of velocity fluctuations. This new term can be viewed as momentum transfer due to the velocity fluctuations and is called the Reynolds stress tensor \( \tau_{ij} = -\rho \bar{u}_i \bar{u}_j \). The components of the tensor consist of three normal stresses and three shear stresses. The transport due to the velocity fluctuations results in enhanced mixing of flow resulting in energy dissipation and increased heat transfer in the case of thermal flow problems. All the terms of the above equation are resolvable except the Reynolds stresses, and these stresses are the subject of turbulence modeling.

### 3.2.1 Scale analysis of Reynolds stresses with viscous stresses

In this section the order of magnitude of Reynolds stresses \( \tau_{ij} = -\rho \bar{u}_i \bar{u}_j \) compared to viscous stresses \( \nu S_{ij} = \nu (\partial \bar{u}_i / \partial x_j + \partial \bar{u}_j / \partial x_i) \) is considered. It is assumed that the fluctuation velocity scale is \( u_t \), the mean velocity scale is \( U_m \) and the length scale of the mean flow is \( L_m \) which also corresponds to the large eddy length scale. With these scales the magnitude of the Reynolds stress tensor can be interpreted as

\[
\bar{u}_i \bar{u}_j = O \left( u_t^2 \right) \quad (3.8)
\]

and the magnitude of the mean stress tensor can be estimated as
\[ \nu \overline{S}_{ij} = \nu \left( \frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i} \right) = O \left( \frac{\nu U_m}{L_m} \right) \] (3.9)

The ratio of these two order of magnitude is

\[ \frac{u'_i u'_j}{\nu \overline{S}_{ij}} = \frac{u'_i}{\nu} = I^2 R_{em} \] (3.10)

where, \( I \) is turbulence intensity \( (u'_i^2/U_m^2) \) and \( R_{em} \) is mean flow Reynolds number \( (U_m L_m/\nu) \). The scaling analysis shows that the ratio of Reynolds stresses and mean stress tensor is proportional to the turbulence intensity and the mean flow Reynolds number. Therefore, as the Reynolds number increases we get \( u'_i u'_j >> \nu \overline{S}_{ij} \) where the turbulence intensity varies between 5-20% for complex flow geometries. This scale analysis show that the viscous stresses has little effect on the mean flow compared with the induced Reynolds stresses. An exception to this is flow very close to no-slip boundaries where the appropriate flow lengthscale is << \( L_m \).

### 3.3 Turbulence modelling

Most flows of engineering importance are turbulent in nature and this is the case for the flow in the SVV device considered here. The complete understanding of turbulence still stands as a great challenge to scientific community. Many efforts are being made to understand the flow physics of turbulence through experiments and numerical simulations. The most exact approach to simulate the turbulence is to solve the governing equations without any modeling and any approximations. Direct numerical simulations (DNS) is a CFD method that solves the flow governing equations without any model. DNS solves all the relevant time and length scales in the flow field. However, DNS is extremely expensive for complex flow
problems such as those encountered in engineering applications. The computational cost associated with DNS can be roughly estimated using the concept of Kolmogorov microscales.

The present understanding of turbulent scales is due to the pioneering work of Kolmogorov (1941). Kolmogorov’s hypothesis is based on several experimental observations and dimensional analysis. Kolmogorov’s theory hypothesized that turbulent flows are characterized by microscales which are the smallest scale of turbulence. These scales are defined as

\[ \eta \equiv \left[ \frac{\nu^3}{\epsilon} \right]^{1/4} \]  
(3.11)

\[ \tau_\eta \equiv \left[ \frac{\nu}{\epsilon} \right]^{1/2} \]  
(3.12)

\[ u_\eta \equiv [\nu \epsilon]^{1/4} \]  
(3.13)

where \( \eta, \tau_\eta \) and \( u_\eta \) are defined as length scale, time scale and velocity scale, respectively. \( \epsilon \) is the rate of turbulent energy dissipation and \( \nu \) is kinematic viscosity. In DNS, all the scales ranging from the characteristic scale of the flow to the smallest scales or Kolmogorov scales need to be resolved by the computational mesh and time step. The integral length scale of flow is taken as ‘\( L_m \)’ and the approximation \( \epsilon \approx \frac{u_t^3}{L_m} \) is made, where \( u_t \) is rms value of fluctuation. The equation 3.11 implies that a three dimensional DNS requires a number of mesh points \( N^3 \) satisfying following criteria,

\[ N^3 \geq R_{et}^{9/4} \]  
(3.14)
where \( R_{et} = \frac{u_i L_m}{\nu} \) is turbulent Reynolds number. The turbulence is unsteady in nature hence, all DNS simulations has to be time dependent. Considering the Kolmogorov times scale \( (\tau_\eta) \), the time step in numerical simulations has to be less than \( (\tau_\eta) \), in order to achieve physically and numerically accurate solution. Pope (2000) evaluated that the number of time step \( (N_t) \) required are function of turbulent Reynolds number and given by following equations

\[
N_t \geq R_{et}^{1/2} \tag{3.15}
\]

Equation 3.14 and 3.15 indicate that the computing resources such as memory storage and CPU time, needed by DNS grows very fast with the turbulent Reynolds number due to drastic increase in the required number of mesh points and time steps.

The problem of resolving turbulent flow can be circumvented by an alternative approach. This alternative approach consists of deriving and solving equations for the mean flow using approximations and models. Theses models are mathematically formulated in such a way that they simulate the effects of turbulence. Presently, this alternative approach consists of three different classes of turbulence models based upon their method of formulation. These are eddy viscosity, Reynolds stress and large eddy scale models. The first two techniques are based upon the Reynolds averaging approach and are described in detail in following sections.

Before going into details of turbulence modeling, it is important to understand the effect of turbulence on the mean flow. The discussion in section 3.4 and 3.5 serves as an introduction and background for different modeling approaches to be discussed in the further sections.
3.4 The mean and turbulent kinetic energy equation

Equation 3.7 shows the effect of turbulence on the mean momentum equation due to the induced Reynolds stresses. Similarly, the turbulence affects the mean kinetic energy of flow. The governing equation for mean kinetic energy ($K$) can be obtained by multiplying $\vec{u}_i$ with equation 3.7. The resulting equation is

$$\frac{\partial}{\partial t} (\rho K) + \frac{\partial}{\partial x_j} (\rho K \vec{u}_j) = -\vec{u}_j \frac{\partial \vec{p}}{\partial x_j} \quad (I)$$
$$+ \frac{\partial}{\partial x_j} (-\rho \vec{u}_i \vec{u}_j \vec{u}_i) - \rho \vec{u}_i \vec{u}_j \frac{\partial \vec{u}_i}{\partial x_j} \quad (II)$$
$$+ \frac{\partial}{\partial x_j} (\tau_{ij} \vec{u}_i) - \tau_{ij} \frac{\partial \vec{u}_i}{\partial x_j} \quad (III)$$

where $K$ is kinetic energy per unit mass given as

$$K = \frac{1}{2} \sqrt{\vec{u}_x^2 + \vec{u}_y^2 + \vec{u}_z^2} \quad (3.17)$$

As with the mean momentum equation, due to turbulence effects new terms appear in the mean kinetic energy equation. The term (I) represents the rate of change of $K$, (II) represents the transport of $K$ due to convection, (III) represents transport of $K$ by mean pressure, (IV) represents transport of $K$ by Reynolds stresses, (V) represents the production of $K$, (VI) transport of $K$ by viscous stresses and (VII) represents the viscous dissipation of $K$. The term (IV) is of special interest since it shows the effect of the turbulence on the mean kinetic energy. As shown in the scaling analysis with increase in Reynolds number the magnitude of the Reynolds stresses increases and this results in large dissipation of mean kinetic energy of
flow due to the term (IV). In conclusion, in turbulent flows the viscosity has little impact on the mean kinetic energy.

Similarly more information about the turbulence and its interaction with the mean flow can be obtained through the transport equation of turbulent kinetic energy ($k$). Multiplying the instantaneous Navier-Stokes equation by $u_i$ and then decomposition of kinetic energy results in the transport equation for kinetic energy. Subtracting the mean flow kinetic energy equation (3.16) from this equation results in turbulent kinetic energy equation, written as below,

$$
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho ku_j)}{\partial x_j} = -\rho u_i'u_j'S_{ij} - 2\mu S'_{ij}S'_{ij}
$$

where

$$k = \frac{1}{2} \sqrt{u_x'^2 + u_y'^2 + u_z'^2}, \quad S'_{ij} = \frac{1}{2} \left( \frac{\partial u_i'}{\partial x_j} + \frac{\partial u_j'}{\partial x_i} \right)
$$

The mechanism that affect the turbulent kinetic energy on the RHS of equation 3.18 are the production of $k$ (III), dissipation of $k$ due to the viscous effects (IV), transport of $k$ due to pressure fluctuations (V), transport of $k$ due to velocity fluctuations (VI) and transport of $k$ due to viscous stresses (VII).

While the viscous stresses have little impact on the mean kinetic energy flow equation, viscous effects are important in turbulent kinetic energy balance. Above all the terms, the term (IV) is of interest. This term is always positive since it contains the square of $S'_{ij}$ and $a$ indicates positive drain of the turbulent kinetic energy. Term (III) represents the exchange of the energy between mean flow and turbulence showing production of turbulent energy from the mean flow. The
connection between the term (III) and (IV) can be obtained through equation 3.18 by assuming certain flow conditions. Assuming statistically steady homogeneous turbulent flow conditions where all spatial gradients are zero, equation 3.18 can be simplified to the following,

\[- \rho u'_i u'_j S_{ij} - 2 \mu S'_{ij} S'_{ij} = 0 \] (3.20)

This equation shows that the rate of production of the turbulent kinetic energy (III) is balanced by the rate of dissipation (IV). Another observation is that for certain flow conditions the turbulent production is always positive since the dissipation can not be negative.

### 3.5 Dissipation

The rate of dissipation of turbulent kinetic energy is denoted by $\epsilon$. It is the rate at which turbulent energy dissipates through viscous effects and is defined according to following equation (Wilcox (2000)).

$$\epsilon \equiv \nu \frac{\partial u'_i \partial u'_j}{\partial x_k \partial x_k}$$ (3.21)

The transport equation for $\epsilon$ is described as,

$$\rho \frac{\partial \epsilon}{\partial t} + \rho \overline{u_j} \frac{\partial \epsilon}{\partial x_j} = \left[ \frac{\partial \overline{u'_i u'_j}}{\partial x_k} + \frac{\partial u'_k \partial u'_i}{\partial x_i \partial x_j} \right] \frac{\partial \overline{u_i}}{\partial x_j} - 2 \mu u'_k \frac{\partial u'_i}{\partial x_j} \frac{\partial^2 \overline{u_i}}{\partial x_k \partial x_j}$$ (I)

$$- 2 \mu \frac{\partial u'_i}{\partial x_k} \frac{\partial u'_k}{\partial x_m} \frac{\partial^2 \overline{u_i}}{\partial x_k \partial x_m}$$ (II)

$$2 \mu \nu \frac{\partial \overline{u_i^2}}{\partial x_k \partial x_m} \frac{\partial \overline{u_i^2}}{\partial x_k \partial x_m}$$ (III)

$$+ \frac{\partial}{\partial x_j} \left[ \mu \frac{\partial \epsilon}{\partial x_j} - \mu u'_j \frac{\partial u'_i}{\partial x_m} \frac{\partial \overline{u_i}}{\partial x_j} - 2 \nu \frac{\partial p'}{\partial x_m} \frac{\partial u'_i}{\partial x_m} \right]$$ (IV)

(V)

(VI)

(VII) (3.22)
The LHS of the Eqn 3.22 represents the rate of change of $\epsilon$ and transport of $\epsilon$ by convection. The terms (III) and (IV) represent production and dissipation of dissipation, respectively. The term (V) represents the diffusion of dissipation due to fluctuations and the term (VI) represents diffusion of dissipation by molecular interaction and finally, the last term represents the turbulent transport of dissipation. The transport equation for $\epsilon$ is complicated to model since most of the terms involve flow fluctuations. Usually, the modeling of $\epsilon$ is tuned using experimental or DNS data.

### 3.6 Eddy viscosity modeling

Due to the presence of the Reynolds stresses in the equation 3.7, it is unsolvable in its present form. There is no analytical method to determine the value of the Reynolds stresses. Boussinesq (1877) suggested that the Reynolds stresses should be represented as a function of mean velocity gradient. He argued that the analogy of viscous stresses in laminar flows can also be extended to the turbulent flows. In the Boussinesq hypothesis, the constant of proportionality has been called eddy viscosity. According, to this hypothesis the Reynolds Stress tensor linearly relates to the mean strain rate tensor as,

\[ -\rho u'_i u'_j = -\rho \frac{2}{3} k \delta_{ij} + \mu_t \left[ \frac{\partial \pi_i}{\partial x_j} + \frac{\partial \pi_j}{\partial x_i} \right] \quad (3.23) \]

In this equation, $k$ represents the the turbulent kinetic energy. The first term of the RHS of this equation ensures that the sum of the diagonal components of the Reynolds stress tensor equal twice the turbulent kinetic energy. In the above equation the unknown scalars are $k$ and $\mu_t$. In this way the Boussinesq hypothesis forces all components of Reynolds stress tensor to align with the mean velocity gradients. In conclusion, it appears that the eddy viscosity concept is a reasonable and simple method of modeling the turbulent stresses.
The eddy viscosity approach is an approximation and has some limitations. For example, as discussed by Pope (2000), alignment of components of Reynolds stress tensor with mean velocity gradient does not reflect the true behavior of many flows.

### 3.7 The Spalart-Allmaras Model

The Spalart-Allmaras model is a one-equation model. Spalart & Allmaras (1992), formulated the model for prediction of external aerodynamic flows. Due to its robustness in solution convergence, the model has found application in predictions of several flow problems. Spalart & Allmaras (1992) created a model based upon the transport equation for a modified version of turbulent viscosity ($\tilde{\nu}$). The Spalart variable is related to the turbulent viscosity by the following equation.

$$\mu_t = \rho \tilde{\nu} f_{v1}$$

where the viscous damping function $f_{v1}$ is given by,

$$f_{v1} = \frac{\chi^3}{\chi^3 + c_{v1}^3}$$

$\chi$ is the ratio of the Spalart variable to dynamic viscosity ($\equiv \tilde{\nu}/\nu$) and the constant $c_{v1}$ is 7.1. The model uses the transport equation for $\tilde{\nu}$ described by FLUENT (2000) as below

$$\frac{D\tilde{\nu}}{Dt} = G_{\tilde{\nu}} + \frac{1}{\sigma_{\tilde{\nu}}} \left[ \frac{\partial}{\partial x_j} \left( \mu + \rho \tilde{\nu} \frac{\partial \tilde{\nu}}{\partial x_j} \right) + c_{b2} \rho \left( \frac{\partial \tilde{\nu}}{\partial x_j} \right)^2 \right] - Y_{\tilde{\nu}}$$

where $G_{\tilde{\nu}}$ is the production of turbulent viscosity and modeled as,

$$G_{\tilde{\nu}} = c_{b1} \rho \tilde{S} \tilde{\nu}$$

with, the modified source term ($\tilde{S}$) defined as,

$$\tilde{S} \equiv S + \frac{\tilde{\nu}}{\kappa^2 d^2} f_{v2}$$
where $\kappa$ is von Karman constant, $d$ is the distance from the nearest wall and $c_{b1}$ is constant with 0.13 magnitude. The scalar $f_{v2}$ is defined as below,

$$f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}}$$  \hspace{1cm} (3.29)

The source term can be calculated by two different methods. The first method only includes the rotation tensor and the second method accounts for the effects of both rotation and strain tensors. With inclusion of the strain tensor in the calculation of the source term, the model predicts lower turbulence viscosity in swirling flows when compared to the former method and is defined by Mariani et. al. as

$$S \equiv |\Omega_{ij}| + C_{prod} \ min \ (0, |S_{ij}| - |\Omega_{ij}|)$$  \hspace{1cm} (3.30)

where

$$\Omega_{ij} = \frac{1}{2} \left( \frac{\partial u_j}{\partial x_i} - \frac{\partial u_i}{\partial x_j} \right), \ S_{ij} = \frac{1}{2} \left( \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right)$$  \hspace{1cm} (3.31)

and

$$C_{prod} = 2.0, \ |\Omega_{ij}| \equiv \sqrt{2 \Omega_{ij} \Omega_{ij}}, \ |S_{ij}| \equiv \sqrt{2 S_{ij} S_{ij}}$$  \hspace{1cm} (3.32)

FLUENT (2000) have promoted this method of calculating the source term for vortex flows. In this research a great deal of work involves confined swirling flows, hence this particular variant of SA model was applied.

The term $Y_\nu$ appearing in equation 3.26 is associated with destruction of turbulence and is modeled as,
\begin{equation}
Y_v = cw_1 \rho \, f_w \left( \frac{\tilde{v}}{d} \right)^2 \tag{3.33}
\end{equation}

where

\begin{equation}
f_w = g \left[ \frac{1 + cw_3^6}{g^6 + cw_3^6} \right]^{1/6} \tag{3.34}
\end{equation}

\begin{equation}
cw_1 = \frac{cb_1}{\kappa^2} + \frac{1 + cb_2}{\sigma \tilde{v}} \tag{3.35}
\end{equation}

\begin{equation}
g = r' + cw_2(r'^5) \tag{3.36}
\end{equation}

\begin{equation}
r' = \frac{\tilde{v}}{S \kappa^2 d^2} \tag{3.37}
\end{equation}

The constants $cw_2, cw_3, cb_2$ and $\sigma \tilde{v}$ are 0.3, 2, 0.62 and 0.66, respectively.

### 3.8 The $k$-$\epsilon$ model

The standard $k$-$\epsilon$ model was introduced by Launder & Spalding (1972). This model uses two transport equations, one for $k$ and one for $\epsilon$. These two separate transport equations allow the turbulent velocity and length scales to be independently determined, as shown in following equation

\begin{equation}
v = k^{1/2} \quad , \quad l = k^{3/2} / \epsilon \tag{3.38}
\end{equation}

The turbulent viscosity is formulated by dimensional analysis and given as,

\begin{equation}
\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \tag{3.39}
\end{equation}

where $C_\mu$ is constant with magnitude 0.09.

The standard $k$-$\epsilon$ model uses the following transport equations for $k$ and $\epsilon$
\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial}{\partial x_j} (\rho k u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] - \rho u'_i u'_j \frac{\partial u_j}{\partial x_j} - \rho \epsilon \tag{3.40}
\]

\[
\frac{\partial (\rho \epsilon)}{\partial t} + \frac{\partial (\rho \epsilon u_j)}{\partial x_j} = C_\epsilon \frac{\epsilon}{k} + \tau_{ij} \frac{\partial \epsilon}{\partial x_j} - C_{\epsilon 2} \rho \frac{\epsilon^2}{k} \\
+ \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] \tag{3.41}
\]

The \(k\) equation described in 3.40 is basically a modified version of mean turbulent kinetic energy equation (3.18) due to the modeling of the triple correlation appearing in 3.18. The presence of fluctuating terms in equation 3.18 makes it impossible to solve. The Reynolds stresses can be modeled using the Boussinesq hypothesis, based on the gradient diffusion analogy. The turbulent transport and pressure diffusion terms can be approximated in terms of turbulent kinetic energy gradient as,

\[
\frac{1}{2} u'_i u'_i u'_j + p'u_j = -\frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \tag{3.42}
\]

The above approximations and modeling reduces equation 3.18 to its final form described in 3.40. The \(k\) and \(\epsilon\) equations contains five constants that have been adjusted to give reasonable agreement with experimental data for specific flows: \(C_\mu = 0.09\), \(\sigma_k = 1\), \(\sigma_\epsilon = 1.3\), \(C_{\epsilon 1} = 1.44\) and \(C_{\epsilon 2} = 1.92\).

The standard \(k\)-\(\epsilon\) model is valid for fully turbulent flows i.e. usually the flow regions away from the walls. The \(k\) and \(\epsilon\) equations are based on the assumption of isotropy which fails in the near wall region due to the high velocity gradients. In FLUENT, the \(k\) and \(\epsilon\) equations are integrated with the near wall flows by two approaches.
The first approach is applied for coarser grids. For coarser grids ($\Delta y^+ > 30$), a logarithmic profile for mean velocity is used for the wall adjacent cell. From this, the wall shear stress and effective viscosity are estimated. The kinetic energy equation is simplified assuming $\partial k/\partial n = 0$ where $n$ is wall normal coordinate and the production term is calculated as,

$$\rho u'_i u'_j \frac{\partial u_j}{\partial x_j} = \frac{\tau^2}{\kappa \rho C^\mu_{1/4} \kappa_p 1/2 y_p}$$

(3.43)

where subscript $p$ represents the near wall centroid. The dissipation equation is not solved, instead a local equilibrium between the kinetic energy and the dissipation rate is assumed and $\epsilon_p$ is obtained as

$$\epsilon_p = \frac{C^\mu 3/4 \kappa_p 3/2}{\kappa y_p}$$

(3.44)

The second approach is to resolve the flow near the wall into the laminar sublayer using fine grids and to solve modified equations in the near-wall region. In this method, the whole domain is divided into two layers namely, in the viscosity affected region and fully turbulent region, based on the $Re_y$ defined as,

$$Re_y = \frac{\rho y \sqrt{k}}{\mu}$$

(3.45)

where $y$ is distance from the wall and $k$ is the turbulent kinetic energy. For the viscosity affected region, $Re_y < 200$. In the viscosity affected area a separate equation is solved to evaluate the effective viscosity ($\mu_{t,2l}$). The two layer viscosity $\mu_{t,2l}$ smoothly blends the laminar viscosity $\mu$ near the wall adjacent cell to turbulent viscosity $\mu_t$ in the log region ($Re_y < 200$). More details about the wall treatments can be found in the FLUENT (2000) user guide.
3.9 The Reynolds Stress Model (RSM)

The Reynolds stress model is considered the most elaborate RANS turbulence model since it solves the transport equations for each Reynolds stress, hence completely avoiding the Boussinesq hypothesis. Since the RSM solves for every Reynolds stress, it is theoretically well placed to represent anisotropic turbulence. However, this also means that the RSM has to solve more equations than eddy viscosity turbulence models, rendering it a computationally expensive model. The model also solves a transport equation for the rate of turbulent kinetic energy dissipation ($\epsilon$), which is on Achilles heel of RANS turbulence modeling.

The transport equation for Reynolds stress can be represented in the following form

$$
\frac{\partial (\rho \overline{u_i'u_j'})}{\partial t} + \frac{\partial (\rho \overline{u_k'u_j'})}{\partial x_k} =
$$

(I) $- \frac{\partial}{\partial x_k} \left[ \rho \overline{u_i'u_j'} u_k + \rho' (\delta_{jk} u_i' + \delta_{ik} u_j') \right]$

(II) $\frac{\partial}{\partial x_k} \left[ \mu \frac{\partial}{\partial x_k} (\overline{u_i'u_j'}) \right] - 2\mu \frac{\partial u_i'}{\partial x_k} \frac{\partial u_j'}{\partial x_k}$

(III) $p \left[ \frac{\partial u_i'}{\partial x_j} + \frac{\partial u_j'}{\partial x_i} \right] - \left( \rho u_i' u_k \frac{\partial u_j'}{\partial x_k} + \rho u_j' u_k \frac{\partial u_i'}{\partial x_k} \right)$

(IV) (V) (VI) (VII) \hspace{1cm} (3.46)

The term (I) represents the rate of change of $\overline{u_i'u_j'}$, the term (II) ($= T_{ij}$) represents the convection, the term (III) ($= D_{T,ij}$) represents the turbulent diffusion, the term (IV) ($= D_{L,ij}$) represents the diffusion of $\overline{u_i'u_j'}$ into flow due to molecular reaction, the term (V) ($= \epsilon_{ij}$) represents the dissipation, the term (VI) ($= \phi_{ij}$) represents pressure strain and the term (VII) ($= P_{ij}$) the production of the $\overline{u_i'u_j'}$. With these substitutions equation 3.46 can represented in the following way,
\[
\frac{\partial}{\partial t} (u_i' u_j') + T_{ij} = -D_{T,ij} + D_{L,ij} - \epsilon_{ij} + \phi_{ij} - P_{ij} \quad (3.47)
\]

The terms \( T_{ij}, D_{L,ij} \) and \( P_{ij} \) do not require modeling. However, the terms \( D_{T,ij}, \phi_{ij} \) and \( \epsilon_{ij} \) need to be modeled in order to close the equations.

Using the gradient-diffusion analogy the turbulent diffusion term \( (D_{T,ij}) \) can be modeled in following way,

\[
D_{T,ij} = \frac{\partial}{\partial x_k} \left( \frac{\mu_t}{\sigma_k} \frac{\partial (u_i'u_j')}{\partial x_k} \right) \quad (3.48)
\]

where the turbulent viscosity is calculated using equation 3.39 and \( \sigma_k \) is taken as 0.82. The pressure strain term \( (\phi_{ij}) \) uses the following decomposition for modeling,

\[
\phi_{ij} = \phi_{ij,\text{slow}} + \phi_{ij,\text{rapid}} \quad (3.49)
\]

The \( \phi_{ij,\text{slow}} \) is the slow pressure strain term and \( \phi_{ij,\text{rapid}} \) is the rapid pressure strain term, and are modeled in following way,

\[
\phi_{ij,\text{slow}} = -C_1 \rho \frac{\epsilon}{k} \left[ u_i' u_j' - \frac{2}{3} \delta_{ij} \right] \quad (3.50)
\]

\[
\phi_{ij,\text{rapid}} = -C_2 \left[ (P_{ij} - T_{ij}) - \frac{2}{3} \delta_{ij} (P - T) \right] \quad (3.51)
\]

where the model constants are \( C_1 = 1.8 \) and \( C_2 = 0.6 \). The \( P_{ij} \) and \( T_{ij} \) are defined in equation 3.46 and \( P = \frac{1}{2} P_{kk} \), and \( T = \frac{1}{2} T_{kk} \). The other details of pressure strain modeling can be referred through FLUENT (2000).

Another term which requires modeling is the dissipation tensor \( (\epsilon_{ij}) \). It is assumed that the \( \epsilon_{ij} \) is isotropic and related to the scalar dissipation \( \epsilon \) by,

\[
\epsilon_{ij} = \frac{2}{3} \rho \epsilon \delta_{ij} \left( 1 + \frac{2k}{\gamma RT} \right) \quad (3.52)
\]
The RSM in FLUENT requires boundary conditions for individual Reynolds stresses. In this research, most of the RSM simulations were carried out using fine grid where the near wall flow was resolved. For fine meshes, the laminar stress strain relationship is applied to near wall cells to estimate the Reynolds stresses. The following equations links the Reynolds stresses at the near wall cells with the wall shear stress,

\[
\frac{u_{\tau}r^2}{u_{\tau}^2} = 5.1, \quad \frac{u_{\eta}r^2}{u_{\tau}^2} = 1, \quad \frac{u_{\lambda}r^2}{u_{\tau}^2} = 2.3, \quad \frac{u_{\tau}u_{\eta}}{u_{\tau}^2} = 1 \tag{3.53}
\]

where \(u_\tau\) is friction velocity defined as, \(u_\tau = \sqrt{\tau_w/\rho}\) where \(\tau_w\) is wall shear stress. \(\tau, \eta\) and \(\lambda\) are local coordinates where, \(\tau\) is tangential coordinate, \(\eta\) is normal coordinate and \(\lambda\) is binormal coordinate.

### 3.10 Computational Implementation in FLUENT

The process of obtaining a numerical solution from FLUENT can be divided into three elements. These are pre-processor, solver and post-processor. The first step in the CFD study is pre-processing. The pre-processing involves the geometry preparation and meshing. The geometry basically defines the domain of a problem under study, in space. The meshing involves dividing the whole domain into cells. Depending upon the connectivity between cells the mesh can be structured or unstructured. Most engineering problems are complex in geometry, as is the SVV device considered here. For such complex geometries an unstructured or blocked structured meshing approach is preferred. In this research, quadrilateral cells for 2D cases and hexahedral cells for 3D cases, were used for all simulations. Another important step in the pre-processing is defining the boundary conditions for the discretized domain. In this step, available experimental information is imposed on the model, in such a way that the CFD model mimics the experimental set-up conditions.
The next step is to solve the governing equations over all the cells. The numerical solution of the governing equations can be obtained using several discretization methods such as, finite volume method (FVM), finite element method (FEM) and finite difference method (FDM) etc. The FVM is most commonly used in the industry since it is mathematically conservative and it can be conveniently adopted for unstructured meshes. In this research, for all simulations the FLUENT solver was used. The FLUENT solver is based on a finite volume approach and uses the cell center approach where the flow variables are calculated and stored at cell centers.

In FLUENT, the discretized equations are solved using either a pressure based or a density based approach. The density based approach is well suited for compressible problems where density changes are highly significant. In this research pressure based approach was used for all simulations since the effect of compressibility was limited. The pressure based solver solves the continuity and momentum equations in either a coupled or a segregated manner, and energy and turbulence equations are solved sequentially. In this research, for all simulations the coupled approach was preferred. In the coupled approach, the continuity and momentum equations are solved simultaneously. Due to this the convergence of this approach is faster than the segregated approach where momentum equations and pressure correction (applied through continuity) equations are solved sequentially.

For most of the simulations presented in this thesis the spatial discretization was achieved using a second order upwind scheme and a second order implicit formulation was used for time marching for unsteady simulations. However, for the vibration study test cases in Chapter 8, the spatial discretization for Reynolds stresses was reduced to first order since the second order showed convergence difficulties.
The last phase of the CFD process is post-processing the numerical results. FLUENT allows visualization of results through plotting of graphs and contours over the surfaces. Post-processing plays an important role since it helps to understand the flow features through several flow visualization techniques.

3.11 CFD simulations

An objective of the CFD is to understand the flow structures occurring in a SVV. These are almost impossible to visualize through experiments. A clear understanding of flow behavior could be also used to increase the performance of the device. In this study, the primary aim was to show that CFD techniques are suitable for prediction of flow within the SVV device. Due to the complex flow patterns within the device, the numerical study was divided into three main parts. Firstly, the capabilities of CFD to determine the flow within the confined vortex chamber were investigated using the experimental data obtained by Pivirotto (1967). This part of the study considered the suitability of the selected turbulence models for prediction of swirling flows. The details of the experiment and the numerical results are reported in Chapter 4. The second part of the numerical study is mainly devoted to predictions of flow switching phenomena. In this work the experiment conducted by Tesař (2010) was used as a test case for numerical simulations. The details of experiment procedure and results of numerical simulations are reported in chapter 5. The studies reported in the Chapters 4 and 5 form the basis for the numerical study of the SVV device since the SVV incorporates the important features of both the confined vortex chamber and flow switching phenomenon. Chapter 6 reports the performance of different turbulence models for prediction of experimental data and the capability of CFD to predict the distinct behavior of flow occurring in the SVV. Chapter 7 reports the results of numerical simulations carried out to determine the dynamic flow switching characteristics of the device.
Subsequently, chapter 8, reports the results of numerical simulations performed to predict the effects of vibration on the performance of the device.
Chapter 4

Numerical analysis of confined swirling flows

4.1 Introduction

Figure 4.1 shows the early design of the SVV device which was numerically and physically tested by Scanlon et al. (2009). The present SVV device which has been studied in this work has been evolved through various design configuration such as shown in Figure 4.1.

Figure 4.1: A typical switched vortex valve numerically tested by Scanlon et al. (2009) showing contours of velocity magnitude from CFD on a mid-section plane
In the initial stage of this research, the experimental data for the SVV device was not available, hence it was decided to study the mechanism of operation by breaking-down the whole device in two parts, namely a flow switching device and a vortex chamber. Figure 4.1 shows the conceptual breakdown of a device into two parts. The flow switching mechanism governs the path of flow through the device with the help of control flow through control ports and the vortex chamber determines the resistance offered to flow through the device.

This chapter is focused on numerical simulation of confined swirling flows, typically of those found in the vortex chamber of fluidic devices. The main objective was to assess the accuracy of currently available numerical methods and various turbulence models in predicting such flows. As most previous researchers have considered incompressible flow, the present effort focused on the computation of compressible swirling flow. Simulations were run in Fluent using three different turbulence models namely the Spalart-Allmaras (SA), $k$-$\epsilon$ and Reynold stress models (RSM) with standard wall functions and with resolved near wall modeling for some cases. The numerical results were compared with experimental data published by Savino & Keshock (1965). An important conclusion emerged from this study that RSM and SA showed good agreement with the experimental data but $k$-$\epsilon$ failed to capture the expected flow field. The Rolls-Royce plc code VP94 (Chew & Snell (1988)), which is based on momentum integral solutions was also used to predict the flow field in the vortex chamber. The numerical solution obtained from VP94 agreed with the experimental data.

### 4.2 General Background

Many examples of swirling flows are found in nature and engineering devices. According to the nature of their occurrence swirling flows are divided into two categories, confined swirling flows and unconfined swirling flows. Unconfined swirling
flows are those generated in a large fluid body without confinement. The swirling flows, which occur in nature as wind whirls, tornadoes and dust devils are examples of unconfined swirling flows. In contrast to unconfined swirling flows, confined swirling flows are bounded by wall boundaries, which form a vortex chamber. Cyclone separator, fluidic vortex valve, swirl combustor and swirl atomizer are engineering devices whose performance is based on confined swirling flows. The swirling motion of a fluid may be achieved in a vortex chamber by injecting the flow with a high tangential velocity component at the outer circumferential wall of vortex chamber and constraining it to leave the chamber at an inner flow exit. In this study attention is focused on numerical analysis of a vortex chamber which is an integral part of fluidic flow switching devices. Work done by Vinodkumar & Chew (2010) at the TFSUTC has evaluated CFD methods for incompressible and low Mach number flow for such chambers. This evaluation is extended to compressible flow here, with particular relevance to flow in fluidic devices.

### 4.3 Physics of confined swirling flows

Confined swirling flows possess unique flow characteristics which are not found in non-swirling flows. A significant property of strong swirling flow is the balance of centrifugal force by pressure gradient in the radial direction. Since, centrifugal force depends upon radial distance from the axis of symmetry, it produces a strong adverse pressure gradient close to the symmetry axis of a vortex chamber. This decelerates the flow and leads to production of a three dimensional highly complex recirculation zone (Scanlon et al. (2009)). Figure 4.2 shows a typical flow pattern of confined swirling flow in the $r$-$\theta$ plane in a vortex chamber. The flow field is dependent on inlet and outlet flow variables but usually the overall flow pattern is of the general form shown in Figure 4.2.
Figure 4.2: Schematic of flow pattern in the confined vortex chamber

Referring to figure 4.2, as the fluid tangentially enters the vortex chamber through the inlet it experiences a strong centrifugal force due to swirling motion. Fluid divides into two streams and flow along the end walls of the vortex chamber in a spiral manner forming the Ekman layers. The region between the Ekman layers is the rotating core region. Static pressure, density and swirl velocity in the rotating core region depend on radial distance from the axis. Most of the fluid flows to the sink region though the Ekman layers, with negligible flow of fluid through the rotating core. The experiment performed by Savino & Keshock (1965) confirms the absence of the radial velocity in the rotating core. The sink region can be identified from the presence of the severely curved streamlines near to the outlet of the vortex chamber. The flow in the sink region and near the axis of the vortex chamber is complex. It involves high velocity and pressure gradients and reversal of axial velocity. This high three dimensional straining of fluid creates a major
challenge to turbulence models in predicting the turbulence and flow features.

4.4 Literature review

This section of the chapter briefly reviews the past experimental and computational research carried on confined compressible swirling flows. The most important objective in conducting the survey was to find suitable experimental data to be used for validation of available CFD methods. In reviewing the literature attention was primarily focused on compressible characteristics of confined vortices, since this is most relevant to the fluidic vortex chamber. The literature surveyed during the course of research falls in two parts. The first part is experimental investigations with regard to the confined vortex chambers, specially for short vortex chambers having aspect ratio less than unity with high pressure ratio. The second part of the survey consists of computational work done to predict the flow features and details of turbulence properties in vortex chambers using various turbulence models.

4.4.1 Experimental work

Ragsdale (1961), conducted experiments with a confined vortex chamber of aspect ratio \((s/r_a) = 1\), using air with high tangential velocity at the inlet of vortex chamber for different pressure ratios where pressure ratio is defined as static pressure at inlet to outlet. The pressure ratio range was from 1.1 to 3.4. Total and static pressure of air were measured at various location from the outer radius \((r_b)\) to the inner radius \((r_a)\). A rapid variation of static pressure near to the axis of the vortex chamber was found. Savino & Keshock (1965) conducted experiments on a vortex chamber with an aspect ratio of 0.213. The experiments were carried out for a constant pressure ratio of 1.5. They made the first attempt to measure the radial velocity and tangential velocity together using a pitot yaw probe at several radial stations. The resulting measurements showed that, at high swirl to
radial velocity ratio, all the radial inflow at the outer radius of the vortex chamber was diverted axially to the walls of the chamber. The radial velocity plot showed no flow in the radial direction except near to the wall, proving that all radial inflow occurred in the near wall region of the vortex chamber. An asymmetrical radial velocity distribution was observed near to the exit of the vortex chamber. The observations and measurements from this experiment confirm the proposed different flow regions in rotating cavities proposed by Hide (1968). Hide proposed four different regions of flow: (i) An inlet source region, (ii) The Ekman boundary layer near to walls of rotating disc, (iii) The inviscid rotating core separated by two Ekman layers, and (iv) A sink region near to the outlet.

Pivirotto (1967) carried out an experiment on a long vortex chamber of aspect ratio 11 with pressure ratio up to 10 using Nitrogen as the working fluid. The objective of the experiment was to determine the effect of fluid mass flow rate and the exit orifice diameter on radial static pressure distribution. The radial static pressure distributions were measured on the closed end of the vortex chamber and found to be a function of the exit hole dimensions. The study revealed that with an increase in mass flow rate or reference pressure at inlet, the pressure distribution curve in the vortex chamber approached a constant limiting distribution due to the choking of the exit orifice. Wilkinson et al. (1988) conducted several experiments on another confined vortex chamber to study the effects of compressibility due to application of high pressure difference across a vortex chamber. This work is similar to work done by Pivirotto (1967) except the working fluid was superheated steam in these experiments. Pressure tapping and thermocouples were used to measure the radial distribution of static pressure and temperature along the wall of the vortex chamber. The key conclusion emerging from this study was that, due to compressibility effects, the mass flow rate through vortex chamber reduced significantly as the exit orifice of vortex chamber choked. Further increase of mass flow rate or inlet pressure of steam did not change the pressure variation in the
radial direction. Similar observation was also noted by Scanlon et al. (2009) in their experiments with the vortex chamber.

4.4.2 Computational work

Abujelala & Lilley (1985) considered the modeling of confined swirling flow in a circular cylinder in the context of combustion chambers. The incompressible $k-\epsilon$ model was used to predict the turbulence. Computed velocity profiles were compared with experimental data at different locations. The most important outcome of the comparison was the inability of the $k-\epsilon$ model to predict swirling turbulent flow. The shortcoming of the $k-\epsilon$ model was attributed to the poor choice of the closure coefficients in the dissipation equation. Jones & Pascau (1989) evaluated the $k-\epsilon$ and Reynolds stress transport equation turbulence models for prediction of confined swirling flow. Comparison of the numerical results with measurement showed a superiority of the stress transport model over the two equation model. Jones & Pascau noted that due to the absence of any mechanism to account for swirl and its effect, the $k-\epsilon$ model gave poor agreement with measurements. The study concluded with recommendation of the Reynolds stress transport model for confined swirling flows, due to their inherent ability to account for effects caused by swirling flow. The failure of the $k-\epsilon$ model is reported in other publications as well, such as those by Saqr et al. (2009), Sharif & Wong (1994) and Scanlon et al. (2009). The results of these computational studies demonstrated that the $k-\epsilon$ model works well for some cases, however it fails to predict effects due to severe curving of streamlines near to the axis of rotation. Other researchers such as Chen & Lin (1999) and Jawarneh & Vatistas (2006) concluded from their numerical work that the Reynolds stress model was able to predict the salient features of swirling flow such as the peak of tangential velocity and the pressure drop, due to its ability to account for the effects caused by severe streamline bending.
Several publications were also found on application of momentum integral equations for predicting the flow features in confined vortex chambers. All the integral methods reported here inspired from free disc theory by Karman (1924). Based on this integral momentum methodology Wormley (1969) developed an analytical model for the incompressible, steady and axisymmetric flow in vortex chambers. Wormley (1969) considered the interaction of main core vortex flow and the end wall boundary layers. The model developed by Wormley (1969) successfully predicted the pressure variation in a vortex chamber from inlet to the exit radius of the chamber, which demonstrated the applicability of integral methods to complex flow problems such as swirling flow in a confined vortex chamber. In similar work to that of Wormley (1969), Kotas (1975) studied the effect of different profiles for radial velocity in the solution to the integral momentum equations. Chew & Snell (1988) studied the problem of radial inflow between two co-rotating discs using momentum integral methods. Setting the rotation of the two co-rotating disc nearly to zero, the co-rotating disc problem reduces to the conditions of a vortex chamber. Chew & Snell (1988) formulated the continuity, momentum and energy equation to develop an analytical model for compressible, steady and axisymmetric flow. This model is best suited for estimating the pressure drop occurring in a vortex chamber with compressible swirling flow since it accounts for the effects of compressibility with the inclusion of the energy equation in the model formulation.

4.5 Savino (1965) test case

As mentioned above Savino & Keshock (1965) conducted several experiments on a vortex chamber measuring radial velocity along the length of the vortex chamber at several planes located at different radii. This case was selected for further study here as it is reasonably representative of the flow conditions and geometrical similarity of vortex chambers in the fluidic device. The flow similarity is that flow in fluidic device vortex chamber is compressible with aspect ratio less than
1. Figure 4.3 shows the schematic arrangement of the experimental setup showing essential features of vortex chamber experimented by Savino & Keshock (1965).

The radius ratio \((r_a/r_b)\) and the axial gap to outer radius ratio \((s/r_b)\) of the vortex chamber are 0.17 and 0.213 with an outer radius \((r_b)\) of 0.148 m. Air was injected tangentially into the chamber through a series of 48 guide vanes around the periphery of the vortex chamber and discharged to the atmosphere through the exhaust tube. The static pressure measurements were taken on the top wall of the vortex chamber at different radial locations. The radial velocity was measured at seven radial stations \((r/r_b = 0.17, 0.25, 0.34, 0.42, 0.51, 0.59 \text{ and } 0.76)\) scanning in the axial direction. All experiments were carried out for mass flow rate of 0.0948 kg/s at absolute static pressure of 142325 Pa at \(r/r_b = 0.95\).

**Figure 4.3:** Schematic arrangement of experimental setup showing essential features of vortex chamber experimented by Savino & Keshock (1965)
In numerical simulations, the mass flow rate and outlet pressure values were specified through boundary conditions and numerically computed static pressure variations and radial velocity were compared with the measured values.

### 4.6 Numerical set up

In this study, a stress transport model and two eddy viscosity models, namely the two equation $k$-$\epsilon$ model developed by Launder & Spalding (1974) and one equation model developed by Spalart & Allmaras (1992) model, were used to simulate the swirling flow in a confined vortex chamber. The numerical results were compared with the experimental data obtained from the publication by Savino & Keshock (1965). An inhouse Rolls-Royce built code, VP94 which is based on an integral method formulation, developed by Chew & Snell (1988) was also used to evaluate flow variables in vortex chamber.

The axisymmetric behavior of steady swirling flow in confined chambers is reported in publications such as, Savino & Keshock (1965), Wilkinson et al. (1988) and Pivirotto (1967). These publications confirmed that the variation of any flow property in the circumferential direction ($\theta$) is negligible. This observation allows simulation of the flow in vortex chambers with simple 2D models, instead of time consuming 3D models. However, 3D model results are also considered here. In this study, for all 2D CFD simulations axisymmetric, steady and compressible flow was assumed. ANSYS ICEMCFD software was used to create a quadrilateral cell mesh and ANSYS FLUENT software was used as the CFD solver. In most cases the non-dimensional near wall distance ($\Delta y^+$) was kept between 30 and 100. In some simulations flow was resolved in near the wall region by increasing the number of mesh points near the wall such that the $\Delta y^+$ value was less than 2. The frictional torque was monitored to judge convergence of the solver. Table 4.1 shows the details of the CFD setup.
Table 4.1: CFD model set up

<table>
<thead>
<tr>
<th>Item</th>
<th>Descriptions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solver</td>
<td>FLUENT 6.3, Pressure based steady solver</td>
</tr>
<tr>
<td>Fluid</td>
<td>Air, compressible, Operating pressure=101325 Pa</td>
</tr>
<tr>
<td>Pressure-velocity coupling</td>
<td>Coupled</td>
</tr>
<tr>
<td>Discretization</td>
<td>Second order for all equations</td>
</tr>
<tr>
<td>Turbulence modeling</td>
<td>RSM (Linear pressure strain without wall effect)</td>
</tr>
<tr>
<td></td>
<td>SA (Strain vorticity based)</td>
</tr>
<tr>
<td></td>
<td>k-ε (Standard and two layer model)</td>
</tr>
<tr>
<td>Inlet boundary condition</td>
<td>Mass flow rate with specified swirl angle</td>
</tr>
<tr>
<td>Outlet boundary condition</td>
<td>Pressure outlet. A gauge pressure of 0 Pa for all simulations</td>
</tr>
<tr>
<td>$y^+$</td>
<td>30-100 (standard wall functions)</td>
</tr>
<tr>
<td></td>
<td>&lt; 2 (Resolved near wall)</td>
</tr>
</tbody>
</table>

Figure 4.4 shows the 2D model and a 3D 10 degree sector model which were created for these simulations. Mesh independence was demonstrated by comparing solutions obtained from the RSM with different number of mesh cells for both 2D and 3D models. Viscous moment ($M$) was considered as the parameter for mesh dependency where viscous moment is calculated by following equation.

$$M = \int_0^{r_b} \tau_w r dr$$  \hspace{1cm} (4.1)

Mesh independent solution was achieved for a 2D model containing 23900 quadrilateral cells and a 3D model containing 181500 hexahedral cells. Table 4.2 shows the comparison of viscous moment obtained from different mesh size models. These results were obtained with standard wall functions.

Using the wall function approach it was ensured that the first computational node away from the walls lay in the range $30 < \Delta y^+ < 100$ and in near wall resolved layers approach $\Delta y^+$ for the first computational node was kept less than 2. The
2D model which was used with near wall resolved layers models contains 94400 quadrilateral cells. No mesh dependency study was carried out for near wall resolving models. Flow boundary conditions were taken from the available experiment information. The boundary conditions were,

1. At the walls: \( u_\theta, u_r \) and \( u_z = 0 \) and \( \partial T/\partial n = 0 \) where \( n \) is normal to local wall. The k-equation is solved over the entire domain with \( \partial k/\partial n = 0 \) at the
walls. Simulations with wall functions use the following equation to compute the dissipation $\epsilon$,

$$\epsilon = C_\mu k^{(3/2)}/\kappa y$$  \hfill (4.2)

at the near wall mesh nodes with resolved nearwall treatment the following equation is applied,

$$\epsilon = k^{(3/2)}/\iota_t$$  \hfill (4.3)

where $\iota_t$ is the turbulence length scale.

2. **At the inlet:** The mass flow rate was specified as $\dot{m} = 0.0948$ kg/s and the swirl to radial velocity ($u_\theta/u_r$) ratio was adjusted by iterating the solution till the absolute static pressure at radial location $r/r_b = 0.95$ was equal to 142325 Pa. Turbulence intensity was set to 5% and the turbulence length scale was calculated from following equation,

$$\iota_t = 0.07D_h$$  \hfill (4.4)

3. **At the outlet:** In the experiment, the operating air was vented to atmosphere hence the gauge static pressure was taken as 0 Pa.

### 4.7 Results

Figure 4.5 shows the variation of non-dimesional static pressure along the left hand wall of the vortex chamber for 2D and 3D solutions with different turbulence models. The comparison shows that there are small differences between the 2D and 3D model results for RSM and $k-\epsilon$ models where as no difference was found for the SA model. The similarity of 2D and 3D results confirms the axisymmetric flow
behavior in the vortex chamber reported by Savino & Keshock (1965), Wilkinson et al. (1988) and Pivrotto (1967). Hence for further analysis it was concluded that 2D model is sufficient to understand the flow features of the vortex chamber.

![Graph](image)

**Figure 4.5:** Radial non-dimensional static pressure distribution on left hand wall of the vortex chamber for different turbulence models with standard FLUENT wall functions. (a) $k$-$\varepsilon$ FLUENT (b) SA-FLUENT (c) RSM-FLUENT

As shown in Figure 4.6 the standard $k$-$\varepsilon$ model over-predicted the pressure drop while the SA and RSM showed close agreement with experiment data with almost the same predictions. All turbulence models predicted nearly the same pressure drop in the outer part of chamber but showed difference at inner radii. This variation in pressure can be understood by considering the variation of turbulent viscosity in the radial direction as shown along the centerline of vortex chamber in Figure 4.7. The RSM does not calculate the turbulent viscosity since it is not
Chapter 4. Numerical analysis of confined swirling flows

Figure 4.6: Comparison of non-dimensional static pressure distribution with experimental data

based on eddy viscosity hypothesis, however, turbulent viscosity can be evaluated from following equation.

\[ \mu_t = \rho C_\mu \frac{k^2}{\epsilon} \]  \hspace{1cm} (4.5)

Figure 4.7: Turbulent viscosity variation for different turbulence models using wall functions

Comparison of the predicted turbulent viscosities shows that the \( k-\epsilon \) model predicts excessive turbulent diffusion compared to other models. The effect of over-prediction of diffusion in the flow field by \( k-\epsilon \) can be seen in the plot of swirl velocity
along the centerline of the vortex chamber in Figure 4.8. This is consistent with the over-prediction of pressure drop in Figure 4.8.

Figure 4.8: Swirl velocity variation for different turbulence models using wall functions

Figure 4.9 shows a comparison of computed radial velocity distribution with experimental data at different radii of the vortex chamber. The comparison shows that the $k$-$\epsilon$ completely failed to capture the variation of radial velocity in the axial direction, whereas RSM and SA models successfully captured the strong variation of radial velocity near to the walls.

Figure 4.9: Comparison of computed radial velocity variation with experimental data (a) $r/b = 0.52$ (b) $r/b = 0.256$
The absence of Ekman layers in the $k$-$\epsilon$ solution can be seen in Figure 4.10 which shows flow streamlines. RSM and SA models show the bifurcation of flow of fluid into two Ekman layers separated by a rotating inviscid core. The $k$-$\epsilon$ solution shows very weak Ekman layer formation.

![Figure 4.10: Contour of stream functions for different turbulence model using wall functions (a) SA (b) RSM (c) $k$-$\epsilon$](image)

The above results were computed using wall functions. In this approach turbulence quantities in the area near to the wall are given by semi empirical functions known as wall functions. The wall functions basically link the near wall cell solutions variables and flow quantities to the wall. Their use reduces the computation time required by resolved wall layer treatments where flow is resolved into the laminar sublayer. Results are presented below using models with resolved wall treatments. Figure 4.11 shows that for these solutions the first cell near to the wall lies in the laminar sublayer and a sufficient number of cells lie in transition zone. This confirms that near wall mesh is satisfies to the modeling guidelines.

Figure 4.12 Shows the comparison of computed non-dimensional pressure variation on the left hand wall of the vortex chamber with experimental data. SA and RSM
Chapter 4. Numerical analysis of confined swirling flows

Figure 4.11: Computed non-dimensional velocity with resolving near wall treatment against $\Delta y^+$ for different turbulence models at $r/b = 0.256$ of left hand disc

show almost identical solutions which are in good agreement with experimental data. The $k-\epsilon$ model solution over-predicted the pressure drop similar to solution obtained using wall functions.

Figure 4.12: Comparison of computed non-dimensional static pressure distribution on left hand wall of the vortex chamber using resolved nearwall treatment with experimental data

Figure 4.13 shows the variation of turbulent viscosity along the centerline of the vortex chamber. The turbulent viscosity prediction for $k-\epsilon$ is at least 2 orders higher than the SA and RSM turbulent viscosity predictions. The effect of high diffusivity on static pressure can be seen in Figure 4.12. The high predictions of
turbulent viscosity by the $k$-$\epsilon$ model compared to other models, results in higher energy dissipation which in turn show higher static pressure drop.

Figure 4.13: Turbulent viscosity variation for different turbulence models using resolved nearwall treatment

Figure 4.14 shows the comparison of computed radial velocity with experimental data at two different radii. The solutions obtained with wall functions and resolved wall layer treatment are similar, with the SA model and RSM showing good agreement with experimental data and the $k$-$\epsilon$ model completely failing to capture the experimental trend, especially in the near wall region.

Numerical solutions were also obtained with the Rolls-Royce integral method based code VP94. Figure 4.15 shows the comparison of computed flow properties by
Chapter 4. Numerical analysis of confined swirling flows

VP94 with different turbulence models and experimental data. Computed static pressure variation is in good agreement with experimental data except near to the inner radius \((a/b = 0.17)\). The integral method assumes equal mass flow rates through the Ekman layers, and hence the same radial velocity profile. However, the experiment showed slightly asymmetric radial velocity profiles.

Figure 4.15: (a) Comparison of computed flow properties by VP94 with different turbulence models with resolved nearwall treatment (a) Static pressure variation on left hand wall (b) Swirl velocity variation along the centerline of vortex chamber

Figure 4.16 shows the comparison of computed radial velocity variation with experimental data at the inner radius. The difference in the radial velocity profile on left and right hand wall can be seen in plot. This asymmetrical nature is attributed to the fact that it is easier for flow to exit through chamber from the right hand side Ekman layer than the left hand side.

4.8 Conclusion

Compressible swirling flow in a vortex chamber was numerically simulated using three different turbulence models, two eddy viscosity based models (SA and \(k-\epsilon\)) and a Reynolds stress model (RSM). Use of both wall functions and resolved near wall treatments was also explored. The numerical flow simulations were compared with the experimental data obtained from Savino & Keshock (1965).
The following conclusions can be deduced from the computational studies reported in this chapter.

- Several experiments conducted on confined swirling flows confirm that in general flow can be divided into four regions, namely source, rotating core, Ekman layer and sink. The SA and RSM model successfully predicted all four regions.

- CFD study showed that the accuracy in prediction of swirling flows is strongly affected by the accuracy in prediction of the rotating core which occupies the majority of the flow region in the chamber. The SA and RSM showed good agreement with all experimental data and observations. The $k$-$\epsilon$ model failed to predict the expected flow pattern since it overpredicted of turbulent viscosity in the core region.

- Comparisons of computed radial velocity with measured data showed good agreement, for the SA model and RSM. The SA and RSM also predicted asymmetrical radial velocity profiles near to the inner radius of vortex chamber. This observation was absent in the $k$-$\epsilon$ model predictions.
The numerical simulation reported in this study can be divided into two parts, namely with wall functions and with resolving flow near the wall approach. Both methods of simulations resulted in almost identical results, however it was found that the simulations with wall function approach were less time consuming than the latter approach.

The predictions of the integral method are encouraging. Despite the simplicity of method, the integral method showed good comparison with experimental data. The accuracy in prediction can be attributed to the assumptions made in model formulation specially for the rotating core.
Chapter 5

Numerical study of flow switching device

5.1 Introduction

Figure 5.1 shows the early design of an SVV device which was tested experimentally and numerically tested Scanlon et al. (2009). As explained in section 4.1, in order to facilitate the numerical study of the SVV, the device was conceptually divided into two parts. The vortex chamber flow was thoroughly studied and reported in chapter 4.

This chapter is focused on the flow switching mechanism which is another major subsystem of the SVV. The main objective of this study was to analyze the performance of different turbulence models for prediction of complex flows in fluidic flow switching devices. The test case considered is taken from the flow switching device experiments of Tesař (2010). The methodology of flow switching in this device is analogous to the flow switching requirements of the SVV with the exception of the method of flow switching. This exception is explained in section 5.2.
Figure 5.1: A typical switched vortex valve with contours of predicted velocity magnitude, numerically tested by Scanlon et al. (2009)

The best practices learnt from this study were implemented in detailed numerical study of the SVV which is reported in chapter 6. The computational methods used in this study include large eddy simulations (LES) with the Smagorinsky-Lilly subgrid-scale model and unsteady Reynolds-Averaged Navier-Stokes (URANS) models. The URANS models include, the Reynolds stress model (RSM), the $k$-$\epsilon$ model and the Spalart-Allmaras (SA) model. Due to the limited computational resources the grid resolution was used for URANS was also used for LES simulations. While URANS showed qualitative agreement with measured device characteristics, unexpectedly LES showed large differences from the measured data. The SA model gave the best agreement with measurements and the URANS models all reproduced the switching phenomena to some extent.

### 5.2 Literature review

Accurate prediction flow of flow switching fluidic devices is a challenging task for turbulence models. The flow is complex in nature since it involves the separation of flow due to its geometric similarity with backward facing step and diffuser flows. The difficulties in predicting separated flows with RANS models were reported by Jang & Leschziner (2001). This led to the conclusion that the eddy
viscosity based turbulence models, $k-\epsilon$ and $k-\omega$ showed a poor comparison with experimental data. Breuerl et al. (2003) numerically investigated the one equation Spalart-Allmaras turbulence model for prediction of separated flow past an inclined plate with high angle of attack. Comparison of numerical results with experimental data showed the inaccuracy of numerical predictions for naturally unsteady separated flow cases. Unlike previous authors, Saric & Tropea (2006) used unsteady RANS turbulence models to predict the flow over a backward facing step and a hill hump. They employed the Spalart-Allmaras and $k-\omega$ turbulence models in numerical simulations, which failed to reproduce all the important effects observed in experiments. This conclusion is in harmony with other authors findings. The disagreement between URANS/RANS numerical results and experimental data can be found in other flow cases such as these reported by Rodi (1997) and Claudio & Schewe (2010). Rodi (1997) used $k-\epsilon$ and Reynold stress (RSM) turbulence models to predict the flow around a square cube, where the comparison of numerical data with experimental findings showed the inability of both turbulence models to reproduce the fluctuations caused by shedding. The disagreement with measurements of the $k-\epsilon$ model was attributed to over prediction of turbulent viscosity. Claudio & Schewe (2010) used the $k-\omega$ model and the Reynold stress based explicit algebric stress model (EASM), unsteady RANS turbulence models for modeling the flow over a square cylinder. The numerical study led to the conclusion that the $k-\omega$ failed to capture the details of dynamics of flow separation whereas, EASM gave satisfactory agreement with experimental data.

The above findings lead to the conclusion that the URANS/RANS methodology is not sufficiently accurate and reliable to simulate the flows where large scale unsteadiness dominates, such as separated flow, bluff body flows and flows involving vortex shedding. The limitations of URANS/RANS have motivated the development of large eddy simulation (LES). LES is regarded as an intermediate approach between URANS and direct numerical simulation (DNS) with respect to accuracy.
and computation cost. In contrast to URANS, LES resolves the large turbulent eddies directly and only the smaller eddies are modeled. Since the smaller eddies present the quasi-isotropic turbulence properties, they are less affected by the boundaries and hence may be less difficult to model. By virtue of its ability to capture the dynamics of large eddies, LES may provide more accurate results than URANS in complex flows such as separated and bluff body flows. However, LES is computationally much more demanding than URANS.

Superiority of LES over RANS in modeling flow separation was demonstrated by Rodi (1997). LES with a Smagorinsky sub-gridscale model was reported to capture the turbulent flow features and the separation details. Despite the better performance than RANS, LES gave large differences with experimental data. Similar conclusions were drawn by several authors. For example, Breuerl & Jovicic (2001), Breuerl et al. (2003) and Young & Ooi (2007) all used LES with the Smagorinsky model for highly separated flows. Lubcke & Thiele (2001) preferred a dynamic subgrid scale approach over the Smagorinsky model in predicting the flow structure for a bluff body flow case. The LES approach has been applied to other complex flow cases which are difficult to predict with URANS models. Cases considered include flow over a back facing step, separation in a diffuser and at a wall bump. Grigoriadis & Goulas (2004) used the Smagorinsky model to simulate flow over a back step. The numerical results were found to be in good agreement with experimental results using a relatively coarser mesh compared to the usual requirements of LES. The Smagorinsky model was used to simulate the separation in a diffuser by Schneider & Rodi (2009). The details of separation were captured by LES whereas RANS failed to predict some flow features. You & Moin (2005) used a dynamic subgrid scale and standard Smagorinsky models to numerically simulate the flow over a bump. The dynamic subgrid scale model performed better than the Smagorinsky model but both models were able to predict the major flow features.
From the literature it is concluded that, the present URANS/RANS method based
turbulence models are not able to predict flows where large separation occurs.
However the computational efficiency of URANS models compares favourably with
LES. Previous work done by researchers on LES sheds light on two major impor-
tant conclusions. The first conclusion is that LES performs better than URANS
and successfully captures the unsteadiness of flow. However, the discrepancies of
numerical results compared with experimental data are also appreciable. Another
conclusion is that, depending upon modeling of small scale turbulence, numerical
results differ. This chapter presents a summary of results obtained from URANS
and LES for simulation of the flow switching fluidic device, designed and tested
by Tesař (2010).

5.3 Tesař (2010) test case

Tesař (2010) designed and tested a flow switching device whose sole purpose is to
switch the flow from one stream to another without using any mechanical means.
Unlike the SVV where flow is switched due to interaction between control flow and
main nozzle jet, in this device flow was switched by blocking one of the outlets
(explained in later part of this section). However, the interaction of the nozzle jet
with the applied pressure differences is expected to be similar to the main nozzle
jet flow of the SVV during the switching. Another reason for selecting this test
case was the similarity of the geometrical features with the SVV device. The basic
geometrical configuration of the device is schematically shown in Figure 5.2. The
flow through the device was characterized by the Reynolds number (\(Re\)) which is
defined as below,

\[
Re = \frac{\rho b \bar{u}}{\mu}
\]  

(5.1)
where, ‘b’ is the width of the nozzle, ‘$\overline{v}$’ is the average velocity of air at nozzle exit, ‘$\rho$’ denotes fluid density and ‘$\mu$’ denotes fluid viscosity. Different Reynolds numbers were obtained by changing the mass flow rate at inlet of the device and different geometrical configurations were investigated by changing the shape of the splitter. In this study the numerical simulations were done for one particular Reynolds number ($Re=27300$) with a cup shaped splitter geometry.

### 5.3.1 Characteristic Curve

The operation of the flow switching valve can be explained by reference to the characteristic curve. This is a graphical representation of the dependence of the output specific energy difference $\epsilon_Y$ on the ratio of output to input mass flow rates ($\mu_Y$). The output flow rate through branch ‘Y’ is non-dimensionalised by relating it to mass flow rate at the inlet as shown in the following equation,
Figure 5.3: Characteristic curve obtained for flow switching device shown in Figure 5.2 for Re= 27300.

\[
\mu_Y = \frac{M_Y}{M_S} \quad (5.2)
\]

Similarly, the output specific energy \( (\epsilon_Y) \) is non-dimensinalised by being related to the supply specific energy difference \( ((\Delta e_S)_{ns}) \) shown in following equation,

\[
\epsilon_Y = \frac{\Delta e_Y}{(\Delta e_S)_{ns}} \quad (5.3)
\]

where, \( \Delta e_Y (= P_Y - P_V) \) is the specific energy difference between locations ‘Y’ and ‘V’ and \( (\Delta e_S)_{ns} \) is specific energy difference between inlet and reference ‘V’ in the no-spillover condition when \( \mu_Y = 1 \). This condition is identified as point ‘A’ on the characteristic curve in Figure 5.3. At this point the entire flow from
the inlet of the device goes through outlet(Y). The point ‘A’ is achieved by closing the outlet(V) so the entire flow can pass through the outlet(Y) which is vented to atmosphere. Different flow rates through outlets are achieved by applying a mechanical blockage on outlet (Y) which causes the flow to reduce in stream ‘Y’, and by opening the outlet (V) flow is allowed to go through the stream ‘V’. The measurements of specific energy at inlet and both outlets at different outlet flow rates form the characteristic curve. With decrease in mass flow rate in stream ‘Y’ the flow conditions reach point ‘B’. Point ‘B’ signifies that small addition of blockage on outlet ‘Y’ will shift almost all the air flow from stream ‘Y’ to ‘V’ and this flow condition is termed as ‘switched flow’.

Note that the specific energy is related to fluid static gauge pressure and velocity magnitude. The precise definition of specific energy at measuring locations can be explained through following relationship

\[ e = \frac{p_s}{\rho} + \frac{1}{2}v^2 \]  

(5.4)

where, ‘e’ is specific energy of flow, ‘\( \rho \)’ is density of flow, ‘\( p_s \)’ and ‘\( v \)’ are static pressure and velocity magnitude at measuring locations. All pressure values reported in this paper are gauge pressures considering atmospheric pressure equal to 101325 Pa.

5.4 Numerical set up

In this study, all simulations were performed with the commercial CFD code FLUENT 6.3. The code is based on a 3D finite volume numerical method solving filtered and unsteady RANS equations. All meshes were created with commercial meshing software\(^1\), ICEM CFD which is a multi block structured meshing tool creating hexahedral cells.

\(^1\)Detail of dimesions of the device can be found from Tesar (2010)
For all simulations, the spatial discretization is of 2\textsuperscript{nd} order. The solutions are
time advanced using a 2\textsuperscript{nd} order implicit method. The coupled implicit pressure
based solving approach was used to solve the pressure velocity coupling which is
faster in convergence compared to the standard SIMPLE algorithm. For all sim-
ulations, between two consecutive time steps 5 inner iterations were performed to
find the next time step flow solution with CFL number equal to one. Incompress-
ible flow conditions were assumed in all simulations since the maximum velocity
occurring in the device is approximately 30 m/s at nozzle exit at standard room
air temperature. The magnitudes of air density ($\rho$) and molecular viscosity ($\mu$)
were respectively $1.175 \text{ kg/m}^3$ and $2.05 \times 10^{-05} \text{ Pa.s}$.

5.4.1 Turbulence models

5.4.1a Spalart-Allmaras Model

The SA model is a one-equation model in which eddy viscosity is determined by
solving a transport equation for an eddy viscosity variable ($\tilde{\nu}$). The strain/vortic-
ity based production model was used over the default vorticity based production
model. FLUENT (2000) recommends the strain/vorticity based production model
over the vorticity based production model, since the latter model over-predicts the
turbulent viscosity in flow field. The accuracy of prediction of flow field by this
strain/vorticity based production model has been proved in swirling flows reported
in chapter 4. For all simulations run with the SA model, a time-step of $15 \times 10^{-05}s$
was used and the boundary layer was resolved ensuring the near wall mesh spacing
nodes $\Delta y^+ < 1.5$.

5.4.1b $k$-$\epsilon$ Model

Calculations were carried out with the $k$-$\epsilon$ model. This is a two equation model
which solves for kinetic energy ($k$) and the rate of its dissipation ($\epsilon$). This $k$-$\epsilon$ model
uses a two layer near-wall methodology when the near wall region is resolved into
the viscous layer. Here the first near wall nodes were ensured to have values of
$\Delta y^+$ near to one ($\Delta y^+ < 1.5$). For all simulations with this model a time-step
of $15 \times 10^{-05}$ s was used. The model constants were taken from Fluent default
settings which represent standard empirical values.

5.4.1c Reynold Stress Model

The RSM does not use the eddy viscosity concept but solves the transport equa-
tions for individual Reynolds stresses. However, the accuracy of prediction of the
flow field is marred by the approximations made in closure assumptions employed
to model various terms of the transport equation for Reynolds stress. For all
simulations run with the RSM a time-step of $15 \times 10^{-05}$ s was used. In order to
resolve the near wall regions, mesh $\Delta y^+$ values were kept near to 1 at the near
wall points. The RSM used in this study consists of linear pressure strain without
wall effects. FLUENT (2000) recommends that the linear pressure strain without
wall effects ensures the simulation convergence for complex geometry problem like
flow switching device considered in this study.

5.4.1d Large Eddy Simulation Model

The LES solver uses the Smagorinsky-Lilly model to calculate the eddy viscosity
modeling the unresolved small scale motion with, using a Smagorinsky constant
of 0.1. Turbulence was added at the inlet boundary condition in terms of velocity
fluctuations by using a spectral synthesizer. The near-wall normal mesh $\Delta y^+$
value was kept near to 1. The grid resolution in the other directions were $\Delta x^+ <$
140 and $\Delta z^+ < 30$ where x and z denote the streamwise and spanwise directions,
respectively. These grid resolutions were calculated at all the walls of the model.
The time step used for LES was kept smaller than the URANS time step in order
to resolve the variation of fluctuations. The time step used for all LES simulation
was $3 \times 10^{-05}$ s. Generally, LES uses finer meshes compared to URANS but in this
study the same mesh was used for LES which was used for URANS. For all flow cases the SA solution was used to initialize the LES simulations.

5.4.2 Boundary Conditions for Numerical Simulation

In the experiment different flow conditions were achieved by partially blocking and unblocking the outlets while maintaining constant air mass flow rate at the inlet of device. From the available flow details the following boundary conditions were applied to the CFD model.

**Inlet:** A turbulent velocity profile was assigned at the inlet of the device which is given as below,

\[
    u = u_{cl}(1 - \frac{r}{R})^{(1/7)}
\]

where, \( u \) is velocity in \( x \) direction, \( u_{cl} \) is centerline velocity, \( R \) is radius of the inlet pipe and \( y \) is distance from wall to centerline. The centerline velocity \( u_{cl} \) was derived such that the nozzle Reynolds number \( (Re) \) based on average velocity was 27300. An inlet flow turbulence intensity of 5\% was assumed for all URANS simulations. The turbulence length scale is estimated as 0.07 times the inlet diameter of the device. For all LES simulations a spectral synthesizer with turbulent intensity of 5\% was used to generate fluctuations in velocity components at inlet.

**Outlet\,(Y):** In the experiment, for the no-spill over case \((\mu_Y=1)\), the outlet ‘Y’ was vented to atmosphere. Hence for this case, atmospheric pressure was assigned at the boundary. For all other points of the characteristic curve, outlet ‘Y’ was blocked mechanically in the experiment, in order to increase the pressure. In the simulations uniform static pressure was assigned at outlet ‘Y’. Values used for different cases are given later.

**Outlet\,(V):** At the no-spillover condition there was no flow through the stream ‘V’ hence the outlet was modeled as a wall for \( \mu_Y=1 \). For all other cases uniform static pressure was applied at outlet ‘V’.
Note that for some flow simulations reverse flow occurred at the both outlets. Presence of reverse flow at any outlet affects the convergence of the simulations to a large extent and hence it is usual practice to avoid the reverse flow with small geometrical changes at the flow outlets. In this study a small converging pipe was added to the actual flow outlets. These extra converging pipes are shown in Figure 5.4 in red. Due to the converging shape the problem of reverse flow at flow outlets was completely removed. However, due to the extra addition of geometry in the model, the numerical boundary conditions were assigned on the surfaces $S_1$ and $S_2$. The surface $S_1$ and $S_2$ are deemed as Outlet ‘$V$’ and Outlet ‘$Y$’ in discussing the boundary conditions.
5.4.3 Criteria for simulation convergence

The convergence criteria used in URANS and LES are different from RANS usual practice. Generally, convergence of flow parameters such as viscous forces, momentum balances or numerical parameters such as continuity residual are considered as conditions for convergence of steady numerical simulations. However, since URANS and LES are unsteady numerical methods, these methods do not give the same behavior of convergence of flow and numerical parameters that can be seen in steady RANS. In this study, URANS and LES simulations are deemed as converged when they reach a statistically converged condition. That is where the mean or average values of data generated by flow field monitors do not change with respect to time. The time required to achieve statistically converged state in simulations was different for each flow case simulated.

An example of convergence is given in Figure 5.5 (a) which shows the variation of total pressure at the inlet of device in the LES for flow case $\mu_Y = 1$. The data captured was time averaged after 0.5, 1.5, 2 and 2.36 sec with step of 0.5 sec except for the last point. The variation of time averaged total pressure can be seen in Figure 5.5 (b). The magnitude of time averaged total pressure fairly achieved the statistical convergence after 2 sec and hence solution was deemed as converged. Data from outlet monitors showed similar behavior. Similar methods have been used to determine the convergence and magnitude of required flow parameters for other conditions.

The CFD analysis started with a study of mesh dependency which was carried out using the RANS approach. In the following sub-section the effect of time step on flow solution derived from URANS simulations is reported. Subsequently, in further sub-sections the URANS and LES results are presented for various flow conditions at different non-dimensional output flow rates ($\mu_Y$).
5.4.4 Mesh Dependency Study

The purpose of this study was to determine the mesh dependency of the solutions. Steady simulations were carried out using three different meshes for the no-spillover ($\mu = 1$) condition. The critical part of the device is the region between the nozzle and splitter where flow is expected to be separated and unsteady. This region includes two steps in the direction of the flow which cause separation. Mesh refinement was concentrated on the steps and the splitter. The mesh refinement was achieved by increasing the number of points in all three directions in critical regions. Figure 5.6 (a) shows the computational grid in the xy plane for the medium resolution mesh of Table 5.1.

The SA model was used to perform the mesh dependency study. Convergence was examined for several physical aspects such as, continuity residual and viscous force ($F_x$) in the x direction. The simulations were judged to be converged viscous force on the splitter ($F_x$) reached a converged value. In all simulations near wall mesh $\Delta y^+$ values was kept near to one. Table 5.1 lists the different meshes used.

The results obtained from the different meshes can be seen in Table 5.1. Refinement of the coarse mesh to the medium mesh led to significant changes in
non-dimensional output specific energy ($\epsilon_Y$) with results moving closer to the experimental value ($\epsilon_Y = 0.375$). Further refining the mesh to 2.8 million cells showed only a small change in numerical results.

The effect of mesh size was also checked by comparing the flow field for different mesh sizes. Figure 5.7 shows the location of an observation plane in the device.
This is near to the splitter where large variation of velocity is expected due to splitting of the flow coming through the nozzle.

Figure 5.7: Location of plane where velocity flow field for different meshes compared in Figure 5.8

Figure 5.8 shows the contours of velocity magnitude at the plane shown in Figure 5.7 for different meshes. The overall flow field structure is almost identical for the medium and fine meshes but appreciable differences can be seen between coarse and medium mesh. The phenomenon of flow splitting can be seen in all three simulations but significant variation of the velocity field can be observed. It may be noted in Figure 5.8 that the results are not completely symmetric about the mid z plane despite symmetry of geometry and boundary conditions. The steady state solution for this case also showed some oscillations which suggest that the problem is unsteady in nature. Hence, an unsteady solution approach was preferred for further analysis and is described in following part of this chapter.

It can be concluded that the results obtained from medium mesh are reasonably mesh independent and require almost half the computing time of the fine mesh. Hence, considering the computational cost savings associated with medium mesh without compromising quality of solution, further analysis was conducted using the medium mesh.
5.4.5 Time Step Study of URANS

The accuracy of simulations of flows with large scale unsteadiness using steady RANS methods is always questionable. Application of steady RANS methods in unsteady flows always results in loss of information about the evolution of flow with respect to time. In order to capture the time effects on flow structure, industry has focused their attention on using URANS models. URANS methods are simply extended versions of the steady RANS methods where time dependent terms are included in the governing equations.

Accurate numerical solution requires that both mesh spacing and time step are sufficiently small, so the unsteady features of the flow can be resolved with better accuracy. Section 5.4.4 demonstrated the effect of mesh size on the solution. In this section, the effect of different time steps on solution are considered. Two different time steps were used in this study, with the SA model for predictions.

Figure 5.8: Contours of velocity magnitude at plane near to splitter as shown in Figure 5.7 for different meshes (a) Coarse (b) Medium (c) Fine, in yz plane
of the no-spillover condition. The medium size mesh was used to represent the fluid domain. The simulations were run till they reached a statistical steady state which took roughly 2.5 s of simulation time in both cases. For both simulations, 5 inner iterations were used to compute the flow field between consecutive time steps. Table 5.2 reports the time averaged non-dimensional output specific energy ($\epsilon_Y$) and the magnitude of drag force in the x direction, obtained from simulations using small time step of $3 \times 10^{-05}$ s and a relatively large time step of $15 \times 10^{-05}$ s. The numerical results obtained from the two time steps are almost identical and hence for further URANS simulations, time step of $15 \times 10^{-05}$ s was used. For all simulations the through flow time ($T_f$) was estimated about 0.1 sec and solutions were ran for at least $10T_f$ i.e. 1 sec.

<table>
<thead>
<tr>
<th>Time step (s)</th>
<th>$\epsilon_Y$</th>
<th>$F_x$ (N)</th>
<th>$y^+$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$15 \times 10^{-05}$</td>
<td>0.36</td>
<td>2.05</td>
<td>&lt; 1.5</td>
</tr>
<tr>
<td>$3 \times 10^{-05}$</td>
<td>0.358</td>
<td>2.05</td>
<td>&lt; 1.5</td>
</tr>
</tbody>
</table>

### 5.5 Results of numerical simulation for device

This section summarizes the numerical results obtained from the different turbulence models for different flow conditions. Figure 5.9(a) shows the comparison of numerically computed time averaged non-dimensional output specific energy ($\epsilon_Y$) with the experimental data. Numerical simulations included four different flow conditions for the device. Different flow conditions represent different states of the flow switching device which include the no-spillover condition represented by point ‘A’, the state just before the experimentally observed flow switching represented by point ‘B’, and intermediate states.
From figure 5.9(a), it can be seen that URANS turbulence models were qualitatively able to capture the trend of the characteristic curve where LES showed some departure from this trend. The SA model shows the best agreement with experiments compared to other turbulence models. The RSM and $k$-$\epsilon$ model produced very similar results and show appreciable discrepancy with experiments.

The comparison in Figure 5.9 (a) lacks information about the magnitude of specific energy differences, since $\epsilon_Y$ is a ratio. Hence, results for the magnitude of output specific energy differences between reference ‘Y’ and reference ‘V’ ($\Delta e_Y$) are also presented in Figure 5.9(b). This shows the comparison of numerically computed time averaged $\Delta e_Y$ with experimental data. It plots the variation of specific energy difference between reference ‘V’ and ‘Y’ with respect to mass flow rate in stream ‘Y’. The comparison of numerical results with experiments shows significant differences, for all turbulence models. All URANS models predicted high specific energy
difference (i.e high total pressure difference) compared to experimental data. The results of simulations are analyzed at particular non-dimensional output mass flow rates ($\mu_Y$) in the following sub sections.

### 5.5.1 Test case 1 (Flow case at $\mu_Y = 1$)

Table 5.3 shows, the comparison of numerically computed time averaged $\Delta e_Y$ with experimental data. The SA model gave the least discrepancy and LES showed the largest deviation from experimental data. The $k-\epsilon$ model and RSM showed almost identical magnitude of $\Delta e_Y$ but they are not in agreement with experiments. It is important to mention that for this flow case in the $k-\epsilon$ and RSM URANS simulations, the unsteady flow motion was completely absent. Hence steady RANS models gave similar results. Unlike the $k-\epsilon$ and RSM, the SA model showed unsteady variations in flow field having discrete frequencies.

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>$\Delta e_Y (\text{J/kg})$</th>
<th>$\Delta y^+$</th>
</tr>
</thead>
<tbody>
<tr>
<td>SA</td>
<td>109</td>
<td>$&lt; 1.5$</td>
</tr>
<tr>
<td>$k-\epsilon$</td>
<td>172</td>
<td>$&lt; 1.5$</td>
</tr>
<tr>
<td>RSM</td>
<td>181</td>
<td>$&lt; 1.5$</td>
</tr>
<tr>
<td>LES</td>
<td>239</td>
<td>$&lt; 1.5$</td>
</tr>
<tr>
<td>Tesar Expt</td>
<td>79</td>
<td>NA</td>
</tr>
</tbody>
</table>

Figure 5.10(a), shows the location of a measuring probe placed in the vicinity of the stream ‘Y’ flow step where unsteady flow behavior may be expected due to the separation. The existence of unsteady fluctuations in the recirculation area due to step in flow was reported by Spazzini & Cicca (2001). From their experimental
study of flow over a back step, they concluded that the zone of recirculation caused by the step displays a broad range of frequency fluctuations. Figure 5.10(b), shows the variation of velocity magnitude with time, recorded by the measuring probe for different URANS models.

![Figure 5.10: (a) Location of measuring probe (shown in blue color) (b) Fluctuation of velocity magnitude recorded by measuring probe for different turbulence models](image)

The magnitude of fluctuations predicted by the SA model and LES were not compared with findings of Spazzini & Cicca (2001) due to differences in flow geometry such as step size. The SA model and LES, both qualitatively agrees with the unsteady flow observations reported by Spazzini & Cicca (2001). Figure 5.11, shows the comparison of predicted frequencies and amplitudes of frequencies at the measuring probe from the SA and LES models. LES predicted a broad range of frequencies at the probe. This contrasts with the dominant frequency given by the SA model (which might be attributed to the flow phenomenon like vortex shedding) which is also evident in figure 5.10(b). The steadiness predicted by the $k$-$\epsilon$ model and RSM is probably due to an overestimation of eddy viscosity in the separation zone which damps out the vortices formed in separation zone and at the edge of splitter.
The disagreement of numerical results with experiment as shown in Table 5.3 can be confirmed by studying the non-dimensional total pressure ($P^*$) variation in the device where non-dimensional total pressure ($P^*$) is defined as follows:

$$P^* = \frac{P_t - P_{ti}}{0.5 \rho (u_{ne})^2}$$  \hspace{1cm} (5.6)

where $P_t$ is time averaged total pressure, $P_{ti}$ is time averaged total pressure at inlet and $u_{ne}$ is time averaged velocity magnitude at the nozzle exit.

Figure 5.12 shows contour plots of non-dimensional time averaged total pressure on the mid z plane of the device ($z=0.0264$ m). The total pressure predictions in stream ‘V’ of the device by URANS models are almost identical for all four solutions whereas the LES shows a significant drop of pressure in stream ‘V’ when compared to URANS predictions. There are appreciable differences between URANS models in stream ‘Y’. The SA model predicted lower total pressure than the RSM and the $k$-$\epsilon$ model. The LES shows significantly higher total pressure. Further examination of the LES results showed total pressures greater than the inlet value
at some interior points. This is a cause for concern and brings the validity of the LES into question.

The differences between numerical results obtained from different turbulence models are further explored by analyzing the mean flow field in the device. Figure 5.13, shows contours of time averaged velocity magnitude in the mid \( z \) plane \( (z=0.0264 m) \) of the device. The flow velocity in the lower branch just downstream of the...
splitter is higher for SA and LES models than for the $k$-$\epsilon$ and RSM model. The acceleration due to the splitter is relatively small for the $k$-$\epsilon$ and RSM model.

A separation bubble at the leading edge of the splitter in the stream ‘Y’ was found only in the SA model solution and LES and this leads to flow acceleration downstream. LES shows the vortex formation just near to the splitter (at entry
of stream ‘V’) which may be the reason for the large total pressure drop in branch ‘V’ shown in Figure 5.12.

Considering the SA model results as reference results since this model gave the closest agreement with experimental data, other models predicted higher total pressure drops as shown in Table 5.3. The higher total pressure drop may be due to the over-prediction of turbulent diffusion. Figure 5.14, shows the comparison of time averaged turbulent viscosity ratio ($\mu_t/\mu_l$) for the URANS models. This comparison does not include the LES for which turbulent viscosity is computed only on subgrid scale. The eddy viscosity predicted by the SA model in the critical region is smaller than that given by the $k$-$\epsilon$ model and the RSM. Absence of any unsteady flow fluctuations in the $k$-$\epsilon$ model and RSM could be attributed to

**Figure 5.14:** Contour plot of turbulent viscosity ratio in xy plane at z=0.0264 m for (a) SA model (b) $k$-$\epsilon$ model (c) RSM
over prediction of turbulent diffusion since higher magnitude of turbulent viscosity tends to damp out unsteady flow fluctuations.

The only experimental parameter which is available for comparison of numerical results is specific energy difference between stream ‘Y’ and ‘V’ ($\Delta e_Y$) at respective measuring probes. Since ($\Delta e_Y$) is the difference of two quantities, it is not possible to know whether the disagreement between the numerical results and experimental data is because of over-prediction of total pressure in stream ‘Y’ or under-prediction of total pressure in stream ‘V’.

5.5.2 Test case 2 (Flow case at $\mu_Y \approx 0.86$)

This flow case was achieved by changing the outlet boundary conditions for streams ‘Y’ and ‘V’. Stream ‘V’ outlet which was considered as a wall in the previous flow case was changed to a static flow pressure boundary with magnitude of 800 Pa (gauge pressure). Stream ‘Y’ outlet which was treated as outlet vented to atmosphere in previous simulations was changed from atmospheric static pressure to static pressure of a 300 Pa (gauge pressure). It may be noted that for all simulations flow was assumed incompressible hence only the difference between two outlet pressures is significant. For all turbulence models, respective previous flow case solutions were used to initialize the simulations.

The rise in static pressure at the outlet of stream ‘Y’ reduces the flow in stream ‘Y’ and increases that in stream ‘V’. In this case all turbulence models approximately gave the same mass flow rate through stream ‘Y’ ($\dot{M}_Y \approx 0.025 \, \text{kg/s}$) and in turn approximately the same value of ratio of output mass flow rate ($\mu_Y \approx 0.86$).

As shown in Table 5.4, it is clear that all turbulence models failed to predict the measured $\Delta e_Y$. Like previous case, in this case also the SA model close to measurements but shows a larger discrepancy. The other turbulence models failed
Table 5.4: Comparison of numerically computed $\Delta e_Y$ with measured data at $\mu_Y \approx 0.86$ ($\dot{M}_Y \approx 0.025$ kg/s)

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>$\Delta e_Y$</th>
<th>$\mu_Y$</th>
<th>$y^+$</th>
</tr>
</thead>
<tbody>
<tr>
<td>SA</td>
<td>214</td>
<td>0.86</td>
<td>&lt; 1.5</td>
</tr>
<tr>
<td>$k-\epsilon$</td>
<td>220</td>
<td>0.86</td>
<td>&lt; 1.5</td>
</tr>
<tr>
<td>RSM</td>
<td>215</td>
<td>0.86</td>
<td>&lt; 1.5</td>
</tr>
<tr>
<td>LES</td>
<td>261</td>
<td>0.88</td>
<td>&lt; 1.5</td>
</tr>
<tr>
<td>Tesar Expt</td>
<td>134</td>
<td>0.86</td>
<td>NA</td>
</tr>
</tbody>
</table>

to show any improvement. Overall, in this case, no encouraging agreement was found.

5.5.3 Test case 3 (Flow case at $\mu_Y \approx 0.64$)

In this case, static pressure on stream ‘Y’ outlet was increased from 300 Pa to 800 Pa and the static pressure on stream ‘V’ outlet was kept at 800 Pa. The increase in static pressure on stream ‘Y’ outlet decreased the air mass flow rate through this stream.

Table 5.5, shows the comparison of numerically computed $\Delta e_Y$ with measured data at $\mu_Y \approx 0.64$ ($\dot{M}_Y \approx 0.019$ kg/s). The SA model exactly predicted the measured magnitude of $\mu_Y$ whereas the prediction of the $k-\epsilon$ and the RSM model is fairly accurate with measured data with deviation of only 1.5% error. LES predicted a larger drop in flow through stream ‘Y’ with the $\mu_Y$ equal to 0.53 ($\dot{M}_Y = 0.0157$ kg/s). The continued trend of large differences from measured data can also be seen in this case. All URANS turbulence models over-predicted the magnitude of $\Delta e_Y$. LES predicted a large drop in $\Delta e_Y$ and hence under-predicted the measured data. According to experiments, a sudden drop in $\Delta e_Y$ represents
Chapter 5. Numerical study of a flow switching device

Table 5.5: Comparison of numerically computed $\Delta e_Y$ with measured data at $\mu_Y \approx 0.64$ ($M_Y \approx 0.019 \text{ kg/s}$)

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>$\Delta e_Y$</th>
<th>$\mu_Y$</th>
<th>$y^+$</th>
</tr>
</thead>
<tbody>
<tr>
<td>SA</td>
<td>252</td>
<td>0.64</td>
<td>&lt; 1.5</td>
</tr>
<tr>
<td>$k-\epsilon$</td>
<td>235</td>
<td>0.63</td>
<td>&lt; 1.5</td>
</tr>
<tr>
<td>RSM</td>
<td>241</td>
<td>0.63</td>
<td>&lt; 1.5</td>
</tr>
<tr>
<td>LES</td>
<td>55</td>
<td>0.53</td>
<td>&lt; 1.5</td>
</tr>
<tr>
<td>Tesar Expt</td>
<td>164</td>
<td>0.64</td>
<td>NA</td>
</tr>
</tbody>
</table>

the flow condition where the rise in air mass flow rate through stream ‘V’ is rapid even though a small amount of static pressure increased on stream ‘Y’ outlet. The curves obtained from experiment do not show the flow characteristics after switching. Compared to experiment as shown in Figure 5.9(b), LES predicted the drop in $\Delta e_Y$ well before it occurred experimentally. Overall, in this case URANS again failed to predict the energy differences in two streams and LES showed premature switching of flow.

5.5.4 Test case 4 (Flow case at $\mu_Y \approx 0.57$)

In this case for all turbulence models the static pressure on stream ‘Y’ outlet was increased from 800 Pa to 865 Pa and pressure on stream ‘V’ outlet was kept at 800Pa. The flow case explained in this section is unique in nature because according to experiment, when $\mu_Y$ reaches a magnitude of 0.51, further increase of outlet pressure for stream ‘Y’ the flow switches from stream ‘Y’ to stream ‘V’. To see if this was reproduced by the CFD, simulations were performed for URANS models with the static pressure on stream ‘Y’ outlet increased to to 900 Pa from 865 Pa (Since LES showed switching in Test case 5.5.3, further simulations were
not carried out for the modified boundary condition). It was found that for the SA model with the addition of 35 Pa, the flow increased from 0.012 kg/s to 0.015 kg/s in stream ‘V’. RSM and $k$-$\varepsilon$ models showed considerably smaller changes in flow rate. The switching behavior is discussed further in Section 5.5.4a.

Table 5.6, shows the comparison of numerically computed $\Delta e_Y$ with measured data at $\mu_Y \approx 0.57$ ($\dot{M}_Y \approx 0.017$ kg/s). All URANS models showed good agreement with the measured magnitude of $\mu_Y$ with maximum deviation of 3.5% but LES showed a difference in prediction of $\mu_Y$ which is nearly 0.46 ($\dot{M}_Y \approx 0.014$ kg/s). Comparing the numerical results with measured data all URANS models over-predicted the magnitude of $\Delta e_Y$.

### 5.5.4a Numerical prediction of flow switching

As explained in Section 5.3, the point ‘B’ represents flow condition just before switching. According to experimental findings presented by Tesař (2010), when the device is at flow condition ‘B’, a small addition of pressure at outlet ‘Y’ results in large increment in mass flow-rate through stream ‘V’. This large increase

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>$\Delta e_Y$</th>
<th>$\mu_Y$</th>
<th>$y^+$</th>
</tr>
</thead>
<tbody>
<tr>
<td>SA</td>
<td>224</td>
<td>0.58</td>
<td>&lt;1.5</td>
</tr>
<tr>
<td>$k$-$\varepsilon$</td>
<td>215</td>
<td>0.59</td>
<td>&lt;1.5</td>
</tr>
<tr>
<td>RSM</td>
<td>208</td>
<td>0.58</td>
<td>&lt;1.5</td>
</tr>
<tr>
<td>LES</td>
<td>-6</td>
<td>0.46</td>
<td>&lt;1.5</td>
</tr>
<tr>
<td>Tesar Expt</td>
<td>177</td>
<td>0.57</td>
<td>NA</td>
</tr>
</tbody>
</table>
in massflow rate through stream ‘V’ also causes a large drop in specific energy difference ($\Delta e_Y$).

Figure 5.15(a), shows the comparison of numerical predictions of output specific energy difference ($\Delta e_Y$) with measured data. The comparison shows that the LES and the SA model display a very steep drop in output specific energy difference ($\Delta e_Y$). In the LES this occurs at a higher flow rate $\dot{M}_Y$ than in the experiment (which is $\dot{M}_Y = 0.015$ kg/s for experiment). Note, however, that the exact point of switching has not been determined in the CFD solutions. The magnitude of rise in mass flow rate though stream ‘V’ and the magnitude of pressure increment at outlet ‘Y’ required to switch the flow is not reported for the experiment. The results for the RSM and the $k$-$\epsilon$ model in Figure 5.15(a) show a relatively small
drop in $\Delta e_Y$. Hence, further numerical simulations were performed for the RSM and the $k-\epsilon$ model.

Figure 5.15(b), shows further comparison of $\Delta e_Y$ given by the RSM and the $k-\epsilon$ model with measured data. As in previous simulations, the flow conditions were changed by changing the static pressure at outlet ‘Y’. The same set of boundary conditions were applied to both models. Figure 5.15(b), shows that RSM predicted a higher drop in $\Delta e_Y$ with increased static pressure at outlet ‘Y’ when compared to the $k-\epsilon$ model.

Overall it can be concluded that SA model showed the best agreement with measured data compared to other turbulence models. LES showed the largest deviations from the experimental data and predicted premature switching in the device. All URANS models showed some qualitative agreement experiment but quantitative disagreement. The SA model showed the sharpest drop in $\Delta e_Y$ close to the experimentally observed switching point.

### 5.6 Conclusion

In this investigation, a flow switching device which is an integral part of a SVV was numerically studied using Tesar (2010) case. The objective of the study was to understand the behavior of flow switching and the prediction capabilities of available turbulence models. In this study URANS and LES methods were employed to simulate the test cases. The quality of numerical predictions was evaluated by comparing with experimental data. The effect of mesh size and time step on solution quality was also studied. Due to the complex geometry of the test case, unsteadiness in the flow was expected and hence for all simulations an unsteady solution method was employed. The following conclusions can be drawn from the present investigation
• All URANS turbulence models qualitatively predicted the experimental behavior. However, for majority of flow cases they quantitatively failed to predict measured data.

• Among all turbulence models, the SA model showed better agreement. The important observation was that the SA model predicted lower magnitude of ratio of turbulent viscosity to laminar viscosity($\mu/\mu_t$) compared to the $k-\epsilon$ model and RSM. This higher prediction of $\mu/\mu_t$ resulted in prediction of steady flow for $k-\epsilon$ model and RSM. However, the unsteadiness predicted by the SA model can be attributed to effects like vortex shedding.

• The complete disagreement of LES with experiments suggest that the LES are not capable of producing quality results using URANS grid resolution. The general guidelines for LES simulations such as $\Delta y^+ < 1$, $\Delta x^+ < 140$ and $\Delta z^+ < 30$ failed in this test case. However these LES results should be regarded as preliminary since no effort was put on understanding the effect of improved grid resolution on solution due to the limited computational resources.

• It was found that much care is needed in applying CFD to fluidic switching devices although URANS models are capable of predicting the qualitative behavior and indicating the approximate point at which switching occurs
Chapter 6

Numerical study of Switched Vortex Valve

Methods of flow control that avoid mechanically moving parts are of considerable interest in industry since such controls are more reliable and require no maintenance. The application of such flow control devices has potential in the gas turbine and aero-engine industry where reliability is of paramount importance. Scanlon et al. (2009), explored the possibility of using these flow control devices in the secondary airflow system of an aero-engine. They proposed and tested a device described as a ‘Switched Vortex Valve’ (SVV), to control the high pressure air lost through a burst pipe or duct. In order to gain better understanding of the flow behavior in the SVV, the device was conceptually broken down into two parts. In chapter 4, the vortex chamber was studied numerically where it was found that the RSM and SA model are capable of predicting the general flow features of confined vortex with good accuracy. In chapter 5, the flow switching mechanism was numerically simulated which is an integral and important sub system of the SVV. It was found that the URANS models can qualitatively predict the flow switching characteristics, especially the SA model.
The numerical studies conducted in chapters 4 and 5, can be considered the building blocks for the numerical study of the SVV. The conclusions that emerged from the previous numerical simulations helped to form a methodology for numerical study of the SVV. Considering the computational cost and accuracy of predictions, the SA and RSM are considered the best choices for prediction of turbulence in the SVV. LES was not considered for this study since it is highly costly compared to URANS. In this study \(^1\) major emphasis was put on the numerical study of the SVV. The numerical study includes the comparison of predicted results with available experimental data and prediction of switching characteristics of device. Two turbulence models namely the SA and RSM were used. The RSM showed a good agreement with measured data and qualitative agreement with other experimental observations.

### 6.1 Objective of the research

Romero (2014) conducted several experiments on the SVV device shown in figure 6.1. The main objective of the experiments was to observe the applicability of the device for expected operating conditions in an aero-engine application. The device was tested for several parameters such as variation of air mass flow rate in different flow states for different pressure ratios, and the effect of splitter and nozzle design. Through several experiments the experimental study concluded that the present device (figure 6.1) was the optimum design.

Though the device was found suitable for expected operating points, very little was known about the detailed flow behavior in the device. Due to the size constraints it was almost impossible to visualize the flow structures through flow visualization. Hence, in order to gain a better understanding of flow structures occurring

\(^1\)Much of the content of this chapter was presented at the 2014 ASME Turbo Expo, Dusseldorf, paper GT2014-26221.
Chapter 6. Numerical study of Switched Vortex Valve

6.2 Switched vortex valve

6.2.1 Introduction

The SVV is a fluidic device based on the wall reattachment Coanda effect. It has two control ports, namely the high flow control port and low flow control port. These are located on each side of the input nozzle as shown in Figure 6.1. The splitter divides the channel passage into two channels which are described as, high flow and low flow passages. In Figure 6.1, ‘A’ shows the low flow passage and ‘B’ shows the high flow passage.

The device works in two different states namely, the high and the low flow state. In the high flow state the jet of air formed from the inlet nozzle is attached to the wall of the high flow channel. The flow path in this state is shown by the blue arrow passing though channel ‘B’ in Figure 6.1. Similarly, in the low flow state the jet of air is attached to the wall of the low flow channel. The flow path in this state is shown by red arrow passing though channel ‘A’ in Figure 6.1.

The device under consideration in this study is biased to start-up in the high flow state. This was achieved by designing an asymmetrical nozzle as shown in figure 6.1. Unlike the bottom edge of nozzle, the top edge is angled in a such way that the flow coming out of the nozzle is biased to pass through the high flow state during the the start-up.

By extracting air from the low flow control port the jet from the inlet nozzle can be deflected so that it attaches to the wall of the low flow passage. This causes
the flow to enter the vortex chamber tangentially forming a vortex and inducing the low flow state. Conversely, extracting flow from the high flow control returns the device to the high flow state. Hence the device can be switched to either state according to user requirements. Once the flow is switched the device remains stable in either state and no longer needs air extraction at the control ports. The ability of the device to remain stable in either state until air is extracted from the control port is described as a bi-stable characteristic. The opening of control ports can be achieved using solenoid valves.

The device responds to the applied inlet-to-outlet pressure differences according to the state in which it is. When the device is in the high flow state the air jet coming out of the inlet nozzle impacts on the vortex chamber wall and passes through the outlet. In the low flow state the air jet coming out of the inlet nozzle enters the vortex chamber tangentially and forms a confined vortex which offers high resistance to flow and reduces the mass flow rate of air passing through the device. Hence for a given pressure ratio \( \frac{P_{in}}{P_{out}} \) the high flow state allows a higher mass flow rate than with low flow state.

Figure 6.1, shows a typical geometry of a SVV which has been extensively studied in this chapter. This device is similar in general features to the device studied by
Scanlon et.al. (Shown in figure 4.1). However, small geometrical differences can be observed at inlet port, the size of splitter and shape of control ports.

6.2.2 Application in Aero-engines

An aero-engine secondary air system performs several important functions such as providing cooling air to compressors and turbines, pressurizing seals and supplying cabin air. A typical aero-engine uses high pressure compressor (HPC) bleed air to cool intermediate pressure turbine (IPT) blades since the bleeding air temperature is lower than the IPT blade temperature. The cooling air required depends on the phase of flight. During the take-off phase high cooling air flow is required to minimize the temperature of the IPT blades but in the cruise phase less cooling air flow is required.

The integration of a SVV into an engine is envisaged as follows 1. During the take-off phase the SVV will allow high cooling air operating in the high flow state. 2. During the cruise phase the SVV will reduce the cooling the air flow by switching air jet to the low flow state. Figure 6.2 shows the proposed location of SVV in the HPC/IPT cooling air flow.

Figure 6.2: A typical secondary air system in an aero-engine with proposed SVV
6.3 Experimental Study

The experimental work was conducted at the University of Sheffield and will be reported more fully elsewhere by Romero (2014). Only a brief outline of the experiment is given here.

The experimental work focused on the steady state flow delivery by a SVV under the high and low flow states. The device was tested under different pressure ratios. Experiments were also conducted on the switching of the device from one state to another. Figure 6.3 shows a schematic of the experimental setup. In all experiments the mass flow rate through the device was measured at the outlet of device using a rotameter (R). A mechanical pressure gauge (P) was used to record the inlet pressure and the outlet was vented to atmosphere. The flow can be switched from one state to another by venting air at control port through the solenoid valve (S). The uncertainty of the measurements is not included in the reported experimental results.

6.3.1 High flow state

The SVV was tested in the high flow state in order to determine the flow delivery capacity under pressure ratios ranging from 1.5 to 2.5. Figure 6.4 shows the variation of measured non-dimensional mass flow rate \( \dot{m}_{\text{non}} \) of air at the applied pressure ratios. The non-dimensional mass flow rate \( \dot{m}_{\text{non}} \) is defined as ratio of measured mass flow rate for a particular pressure ratio \( \dot{m}_{\text{expt}} \) to the mass flow rate of air measured for pressure ratio 2.48 for high flow case \( \dot{m}_{\text{ref}} \). With increasing pressure at the inlet port, the mass flow rate of air through the device increases. It was also found that the device is capable of operating in the high flow state at all pressure ratios which is a primary requirement in design of the SVV. When the device was in the high flow state, high levels of noise were observed, compared to the low flow state. The reason behind the noise generation could be
the impact of the air jet coming from the inlet nozzle on the vortex chamber wall and unsteady flow in the chamber.

**Figure 6.3:** Schematic of experimental setup with instrumentation

**Figure 6.4:** Variation of measured non-dimensional mass flow rate ($\tilde{m}_{\text{non}}$) of air with applied pressure ratio in the high flow state (from Romero (2014))
6.3.2 Low flow state

The same device was tested in the low flow state. The low flow state was achieved by switching the air jet from the high flow state by venting the low flow control port to atmosphere using a solenoid valve. As flow switches to the low flow passage, flow resistance rises due to swirling flow in the vortex chamber and the mass flow rate of air reduces drastically. In this set of experiments the mass flow rate of air was again measured for different applied pressure ratios as shown in Figure 6.5.

The device was also switched from low to high flow state and it was found that the device reproduced the high flow state measured data. This confirms the bi-stable flow characteristic of the device.

The ratio of high flow rate to low flow rate at a given pressure ratio is known as turn down ratio (TDR). For this device the TDR is about 2. King (1985), has shown that the TDR for vortex amplifiers is sensitive to geometric parameters. Such dependencies might also be expected for the present device, but this has not been investigated here.

![Figure 6.5: Variation of measured non-dimensional mass flow rate ($\dot{m}_{non}$) with applied pressure ratio in the low flow state (from Romero (2014))](image)
6.4 CFD Modeling

In this study, all simulations were performed with the commercial CFD code FLUENT 6.3. The code is based on a 3D finite volume numerical method solving unsteady RANS equations. All meshes were created with the commercial meshing software, ICEM CFD which is a multi block structured meshing tool creating hexahedral cells. Emphasis was put on mesh quality. Measures of grid quality include skewness of mesh, adequate aspect ratio and high resolution near the wall boundaries. All simulations in this study were run on dual core Opteron processor where each processor have 4 GB RAM. Figure 6.6 shows a typical mesh distribution in the device with mesh refinement near the walls.

For all simulations, the spatial discretization was 2nd order. The solutions were advanced in time using a 2nd order implicit method. The coupled implicit pressure based approach was used to solve the equations. This converged faster than the SIMPLE algorithm. For all simulations, 5 inner iterations were performed at each discrete time point to find the next time step flow solution. Time steps of $1 \times 10^{-05}$ s and $5 \times 10^{-06}$ s were used. In order to resolve the near wall region, mesh $\Delta y^+$ values were kept near to 1 at the near wall points. In this study two turbulence models were used. These are the eddy viscosity based SA model and the RSM. The SA model considered is a variant which uses strain and vorticity based production. The RSM considered a linear pressure-strain relationship near the wall without wall reflection effects. It was found that use of the RSM employing wall reflection effects in the pressure-strain relationship lead to divergence of the solution. All simulations in this study were deemed to be converged when they reached a statistically steady condition as discussed in section 6.4.1. It was found that all solutions achieved statistical convergence after four through flow times. In this study the flow through time was estimated to be about 0.04 s for the test case with pressure ratio 1.54 and 0.033 s for the test case with pressure ratio 2.48.
All of the simulations were run for at least four flow through times in order to observe any significant unsteady features in flow.

6.4.1 Boundary Conditions for Numerical Simulation

The operation of a SVV is dynamic in nature. Different phases of the operation were simulated separately. This is explained below along with the boundary conditions applied in the CFD model. Figure 6.7 shows the CFD domain with the different boundaries indicated.
6.4.1a Stable high flow state

In this state the air entering through the inlet port exits the device through the outlet. Both control ports are closed hence they are modeled with no-slip boundary conditions. Total pressure is assigned at the inlet of the device. Atmospheric static pressure is applied at the outlet which matches with the experimental conditions. In the experiment, the start up of the device is a transient process hence attention was given to replicating this process. This was achieved by linearly increasing the total pressure applied at the inlet of the device, as shown in Equation (6.1).

\[ P = \begin{cases} 
  t \left( \frac{P_{in}}{0.5T_f} \right) & \text{if } t \leq 0.5T_f \\
  P_{in} & \text{if } t > 0.5T_f 
\end{cases} \]  

(6.1)

where ‘\( P \)’ is magnitude of inlet total gauge pressure at any flow time, ‘\( t \)’ is flow time, ‘\( P_{in} \)’ magnitude of inlet total gauge pressure after 0.5\( T_f \), ‘\( T_f \)’ is flow through
time and the reference pressure was 101325 Pa (atmospheric pressure). At the beginning of the experiment the device was at atmospheric conditions hence all high flow test cases were initialized from atmospheric conditions \((P = 0 \text{ Pa at } t = 0)\).

### 6.4.1b Switching from high to low flow state

In this state the physical boundaries at inlet, outlet and high flow control port remain the same as in the high flow state. In order to achieve switching, the solenoid valve on the low flow control port is opened momentarily allowing air to vent to atmosphere. This was modeled by reducing the static pressure with respect to time on the low flow control port. Numerical studies showed that reducing the static pressure at the control port to 80% of the inlet total pressure was not sufficient to switch the flow state in the device. However, switching occurred for 70% and so the static pressure was linearly reduced to 70% of total inlet pressure in half through flow time. In experiments, no arrangement was made to quantify the opening time required by the solenoid valve. Hence a pragmatic decision was taken to approximate the opening time of solenoid valve by half the through flow time \((T_f)\). A stable high flow case is a precursor to switching hence the switching test case uses initial flow conditions obtained from the high flow case.

### 6.4.1c Stable low flow state

Once the flow switches from high flow to low state, the boundary condition for the low flow control port is changed to that for a wall. All other boundary conditions remain the same as in the high flow state. Flow variables predicted by the switching test case are used as initial solution for the low flow state simulations.

### 6.4.1d Switching from low to high flow state

The boundary conditions for this case are exactly the same as used in Section 6.4.1b except the transient pressure variation is applied at the high flow control
port linearly reducing the static pressure to 70% of total inlet pressure in half the through flow time.

### 6.5 Mesh and time step dependency

This section presents the numerical results obtained from simulations of the SVV carried out at different pressure ratios. It includes a study of grid dependence and the effect of the magnitude of the time step specified for unsteady simulations. The results obtained from two turbulence models are compared with the experimental data. Calculations were conducted on a computing cluster with fast networking. Typically the SA model took 4 seconds for a time step using 2 processors with 8 cores having 1 GB RAM each where RSM took 5 seconds for a single timestep for the same number of processors.

The total error occurring in the numerical simulation is due to two types of error. These errors are due to the differencing scheme and errors due to the modeling. Before quantifying the errors due to modeling it is necessary to quantify the errors due to differencing. This is achieved by running otherwise identical simulations for different grid sizes and time steps. In order to quantify the effect of grid size, four different grids were created. The size of these four grids was 0.5, 1.5, 2.7 and 5 million nodes respectively.

The RSM and SA turbulence model were used in the mesh dependency study. The test case selected had a pressure ratio of 1.54. The high flow case was considered for the study since it is highly unsteady in nature when compared to the low flow case. The boundary conditions employed in this study are explained in the section 6.4.1a.

Figure 6.8 shows the comparison of the numerically computed mass flow rate at the outlet of the device from unsteady calculations with the RSM and SA model with the measured non-dimensional mass flow rate ($\dot{m}_{non}$) of 0.59. All four meshes
used with the RSM showed consistency in the prediction of mass flow rate and similar consistency can also be seen in mass flow rates predicted by the SA model. Due to a convergence problem results from the SA model are not available for the 5 million node mesh. The comparison confirms that for the SA model and the RSM 0.5 million node mesh produces similar results to the 2.7 and 5 million node meshes.

![Figure 6.8](image)

**Figure 6.8:** Comparison of numerically computed mass flow rate by unsteady SA and RSM for different meshes with experimental data

Figure 6.9 shows the variation of time averaged velocity magnitude at point ‘po1 (shown in figure 6.15)’ for different mesh sizes. The SA model showed variation less than 1.5% in velocity magnitude for different meshes when compared to average where as the RSM predicted differences in velocity magnitude less than 5%. Hence, the 0.5 million node mesh was considered for all other simulations.

Figure 6.10 shows a comparison of the predicted time averaged velocity magnitude for different meshes using the SA model and the RSM. Figure 6.10(a) and (b) compare the flow field predicted by the RSM for 0.5 and 2.7 million node meshes respectively. The xy plane represents the mid-plane of the device without inlet and outlet pipes. The predicted flow fields in the vortex chamber from the two meshes are almost identical. With the fine mesh, flow is better resolved in the area
between the splitter and the inlet port compared to the coarse mesh. However, the flow in the high flow passage is almost the same for both meshes which is consistent with to the mass flow rate predictions. Figure 6.10(c) and (d) show the predicted flow field with the SA model for 0.5 and 2.7 million node meshes respectively. Comparison shows almost identical predictions with some minor differences in flow magnitude near the center of the vortex chamber. In conclusion it can be confirmed that the mesh size of 0.5 million is sufficient for flow predictions.

The quality of unsteady numerical simulation results is dependent on the mesh size and the time step used. The factors involved in choosing a time step include the total computational time required for simulation and the degree of flow resolution. A smaller time step results in better flow resolution, however smaller time steps require more computational time. Hence the choice of time step is a trade-off between computation cost and flow resolution. In this study, three different time steps were considered for simulation of the high flow state condition of the device for a pressure ratio 1.54 using the RSM. The predicted non-dimensional mass flow rate is compared with the measured non-dimensional mass flow rate of 0.59.

From Table 6.1 it is clear that with the reduction in time step the predicted mass flow rate moves slightly closer to the measured value. It can be seen that the
Figure 6.10: Contours of predicted time averaged velocity magnitude (m/s) at the mid-plane of the device in mid $xy$ plane. (a) For 0.5 million mesh size with the RSM (b) For 2.7 million mesh size with the RSM (c) For 0.5 million mesh size with the SA model (d) For 2.7 million mesh size with the SA model

 Obtained mass flow rate with the coarsest time step ($5 \times 10^{-05}$ s) is still within 2.3% of experimental data.

It is useful to characterize the time step in terms of the CFL number. It was observed that the maximum CFL number occurred at the outlet of the vortex chamber due to high outgoing fluid velocity. However, away from the outlet of the device, the CFL number was roughly $1/20^{th}$ of the maximum for all above mentioned simulations. Considering the computational cost, it was decided to use a time step of $1 \times 10^{-05}$ s for all test cases with the pressure ratio of 1.54.
Table 6.1: Details of numerical results obtained from different time steps

<table>
<thead>
<tr>
<th>Times step (s)</th>
<th>Predicted non-dimensional mass flow rate ($\dot{m}_{\text{non}}$)</th>
<th>Maximum CFL Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>$5 \times 10^{-05}$</td>
<td>0.573</td>
<td>4190</td>
</tr>
<tr>
<td>$1 \times 10^{-05}$</td>
<td>0.574</td>
<td>740</td>
</tr>
<tr>
<td>$2 \times 10^{-06}$</td>
<td>0.575</td>
<td>147</td>
</tr>
</tbody>
</table>

From similar reasoning a time step of $5 \times 10^{-06}$ was used for all test cases with the pressure ratio of 2.48.

6.6 Results of numerical simulation for device

After establishing the mesh size and time step, studies were conducted to analyze the performance of the RSM and SA model considering different states of the device for pressure ratios of 1.54 and 2.48.

6.6.1 High flow case with pressure ratio of 1.54

At start up, the device is expected to operate in the high flow state. Hence the first objective of this test case was to note the flow path of air predicted by different turbulence models.

Figure 6.10 (a) and (c) show the contours of predicted time averaged velocity magnitude after four through flow times for the RSM and SA model, respectively. Both turbulence models successfully predicted the air to flow through the high flow passage. The performance of both turbulence models is quantified by comparing the numerically obtained mass flow rate with measured data. The RSM predicted non-dimensional mass flow rate of 0.574 is offset $\approx 2\%$ (over-prediction) from experimental data (0.59). The SA model predicted a non-dimensional mass flow
rate of 0.492 which is offset of \( \approx 16\% \) (under-prediction) from experimental data. Hence, it can be concluded that the RSM performs better than the SA model in the high flow case.

The large offset in prediction by the SA model can be attributed to the prediction of a vortex in the vortex chamber shown in Figure 6.10 (c) which is not present in the flow predictions by the RSM shown in Figure 6.10 (a). It is well known that the presence of a vortex in flow field results in lost of flow energy in terms of total pressure drop in flow field. Due to the predictions of vortex flow field by the SA model, the flow experiences pressure resistance which in turn results in lower mass flow rate through the device and hence, under-prediction by the SA model in this case.

Figure 6.10 (a) and (c) reveal an important flow feature in the region of bottom left corner of the splitter. Both turbulence models predicted splitting of part of the jet stream from the main jet of air coming through the inlet nozzle. This splitting of the main jet results in the loss of energy which may be avoided by redesigning the splitter.

### 6.6.2 Low flow case with pressure ratio of 1.54

This section presents numerical results for the low flow case which were obtained by switching the high flow case as explained in section 6.4.1b. With both turbulence models the device reacted to switching by shifting the flow from the high to the low passage. After switching, the simulation was run for at least four through flow times in order to allow the flow to stabilize. The RSM predicted non-dimensional mass flow rate of 0.411 has an offset of \( \approx 3\% \) from the measured non-dimensional flow of 0.4. The SA model predicted a non-dimensional mass flow rate of 0.333 with an offset of \( \approx 16\% \) from measured data.
Figure 6.11: Contours of predicted time averaged velocity magnitude (m/s) at mid-plane of the device in xy plane for low flow with pressure ratio of 1.54
(a) RSM (b) SA

Figure 6.11 shows contours of predicted time averaged velocity magnitude for the low flow case with the SA model and the RSM. The tangential entry of air flow through the low flow passage into the vortex chamber creates the swirling effect. As the flow progresses towards the exit of the vortex chamber the velocity increases. The comparison shown in Figure 6.11 clearly shows differences in the prediction of velocity in the vortex chamber by the two turbulence models.

Figure 6.12 shows the variation of swirl velocity in the vortex chamber along the line shown in Figure 6.15 (a) where position \( r/R = 0 \) represents the axis of the vortex chamber and \( r/R = 1 \) represents a point on the wall of vortex chamber in the mid-plane of the device. The comparison shows that the SA model predicts higher swirl velocity in majority of the vortex chamber. The prediction of the higher swirl velocity explains the under prediction of mass flow rate by the SA model. For confined swirling flows equation (6.2) is often a good approximation for the radial pressure gradient.
\[ \frac{\partial p}{\partial r} = \frac{\rho U_\theta^2}{r} \]  

(6.2)

According to this equation, higher swirl velocity in the vortex chamber leads to high pressure gradient. This represents higher resistance to the flow, and hence the mass flow rate through the device is reduced.

Figures 6.12 and 6.13 show variation of Reynolds stresses predicted by the RSM and SA model respectively, in the vortex chamber.

According to the Boussinesq hypothesis used in the SA model, the Reynolds stresses \( a_{ij} \) are proportional to the mean rate of strain,

\[ a_{ij} = -\rho u_i' u_j' + \frac{2}{3} k \delta_{ij} = \nu_t \rho \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \]  

(6.3)

Compared to the stresses predicted by the RSM, the SA model generally gives higher values. The lower magnitude of stresses predicted by the RSM in the core region of vortex chamber suggests that the core flow is essentially inviscid. This observation is in agreement with the experiment carried out by Wormley (1969).
for a vortex chamber and the numerical study done by Kumar et al. (2012) for vortex chambers.

The higher magnitude of stresses predicted by the SA model results in higher turbulent kinetic energy production ($P$). The production of turbulent kinetic energy is given as,

$$P = u_i u_j \left( \frac{\partial U_i}{\partial x_j} \right)$$  \hspace{1cm} (6.4)
Assuming axisymmetric flow conditions in the core region of the vortex chamber, all mean velocity gradients are negligible except \((\partial U_\theta / \partial r)\). Hence, over prediction of stresses specially \(a_r\theta\) by the SA model, results in higher production of kinetic turbulent energy that in turn increases the Reynolds stresses. This leads to much higher turbulent dissipation in the SA model.

Figure 6.15 shows the location of a sampling probe (pt1) and the unsteady variation of velocity magnitude at the probe predicted by the RSM. The variation of velocity magnitude confirms the unsteadiness in vortex chamber and hence justifies selection of the unsteady solver. The lower velocity magnitude at the probe in the high flow case is due the its location in an area of recirculation which can be seen from Figure 6.10(a). In the low flow case, due to the vortex, the velocity magnitude predicted at the probe is higher in magnitude.

![Figure 6.15: Location of a sampling probe (pt1) in mid-plane of the geometry](image)

![Figure 6.15: Unsteady variation of velocity magnitude (m/s) in vortex chamber taken at ‘pt1’](image)
6.6.3 Prediction of bi-stable characteristic for pressure ratio of 1.54

Figure 6.16 shows the variation of swirl velocity in the vortex chamber taken at ‘pt1’ in Figure 6.15 during the different states of operation of the SVV predicted by the RSM. Line ‘A’ represents the high flow state of the device. Line ‘B’ shows the variation during the switching from high to low flow state with increased magnitude of swirl velocity suggesting that flow is entering the vortex chamber tangentially through the low flow passage. Line ‘C’ shows the stable low flow state in the device. Line ‘D’ shows the variation of magnitude of velocity during the switching from low to high flow. Line ‘E’ shows again shows the stable high flow case.

Figure 6.16 shows that, the device is stable in any state after switching and the device can be successfully switched from one state to another. This confirms the bi-stable characteristic of the device.

![Figure 6.16: Variation of velocity (m/s) in vortex chamber taken at ‘pt1’](image)

6.6.4 High flow case with pressure ratio of 2.48

For the this case, except for the magnitude for the total absolute pressure at the inlet (\(P_{in} = 251314\) Pa), the numerical setup remains the same as that in section 6.4.1.
Figure 6.17 shows the contours of predicted time averaged velocity magnitude. The RSM successfully predicted the flow to pass through the high flow passage. The flow structure near the exit of the vortex chamber shows the presence of weak vortex flow. This vortex was not observed in the flow predicted by the RSM for the test case with pressure ratio of 1.54, explained in section 6.6.1. Here the predicted non-dimensional mass flow rate of 0.936 is offset $\approx 6.5\%$ from the measured data of 1. This difference is more than that for the pressure ratio of 1.54 and might be attributed to formation of the vortex.

![Figure 6.17: Contours of predicted time averaged velocity magnitude (m/s) at mid-plane of device in xy plane for high flow case with pressure ratio of 2.48](image)

(a) RSM (b) SA

The RSM predicted splitting of the main jet stream near the region of the left bottom corner of the splitter, which can also be observed in the test case with a pressure ratio of 1.54.

From Figure 6.17(b) it is clear that SA model failed to predict the flow through the high flow passage, instead it predicted a low flow state. This failure can be attributed to the vortex flow field predicted by the SA model shown in the Figure 6.10(c). With increase in the pressure ratio, it is believed that the strength...
and the spread of the vortex field increases to that extend where the flow finds it easier to pass through low flow passage rather than high flow passage.

A point to be noted is that although Figure 6.4 only includes data up to pressure ratio 2.48, further experiments showed that the device started in the high flow state for pressure ratios up to 3.8.

### 6.6.5 Low flow case with pressure ratio of 2.48

Figure 6.18 shows the contours of predicted time averaged velocity magnitude using the RSM. The model successfully reacted to the switching and predicted stable low flow. The predicted non-dimensional mass flow rate was 0.627 with an offset $\approx 2\%$ from measured data of 0.64.

![Figure 6.18: Contours of predicted time averaged velocity magnitude (m/s) by RSM at mid-plane of device in xy plane for low flow with pressure ratio of 2.48](image)

### 6.6.6 Prediction of bi-stable characteristic for pressure ratio of 2.48

The bi-stable characteristic of the SVV was simulated for a pressure ratio 2.48. Figure 6.19 shows the different states of the device through variation of velocity magnitude at measuring probe ‘pt1’ shown in figure 6.15. Line ‘A’ represents the stable high flow state of the device. Line ‘B’ shows the variation during the switching from high to low flow state with increased magnitude of swirl velocity.
suggesting that flow is entering the vortex chamber tangentially through the low flow passage. Line ‘C’ shows the stable low flow state in the device. Line ‘D’ shows the variation of magnitude of velocity during the switching from low to high flow. Line ‘E’ shows again shows the stable high flow case. The simulations showed that the device was stable in both states and successfully responded to switching.

![Graph showing velocity variation](image)

**Figure 6.19:** Variation of velocity (m/s) in vortex chamber taken at ‘pt1’

### 6.7 Conclusion

In this study various flow states of a SVV were numerically simulated using two turbulence models, namely the SA model and RSM. These states include high flow, low flow and switching of flow. Generally, the flow structure occurring in these states is complex in nature due to flow separation, interaction with control flows and highly swirling flow in the low flow case. Particular attention was paid to the mesh size, time step and flow resolution near the walls. The best practices learnt from chapter 4 and 5 were implemented in this study since the flow switching mechanism and vortex chamber are integral parts of the SVV.

Various test cases were systematically studied in order to find mesh independent solutions and optimize the time step. In all test cases the near wall mesh spacing
\( y^+ \) was kept less than 1.5 to ensure full flow resolution in boundary layers. The test cases analyzed in this study correspond to device pressure ratios of 1.54 and 2.48. The performance of the turbulence models was quantified by comparing numerically predicted mass flow rate with measurements. The following observations were made in this study:

- Both the SA model and RSM predicted the air flow path through the high flow passage during the transient start-up for pressure ratio 1.54. However, for test case with 2.54, the SA model failed to predict the high flow path.

- For all flow states of test cases with pressure ratios 1.54 and 2.48, the RSM showed better agreement with the measured data. The RSM showed over-prediction for all flow states for pressure ratios 1.54 and 2.48, except the high flow state of pressure ratio 2.48. The SA model showed under-predictions for all simulations.

- For the test case with pressure ratio 1.54 both the SA model and RSM responded successfully to the switching procedure, from high to low flow state. The RSM was further extended to simulate the flow switching from low to high flow states for both pressure ratios 1.54 and 2.48. The RSM predicted stable flow states after switching which is in agreement with the experimental observations.

- In order to switch the flow from high to low state the low port control pressure was reduced to 70% of total inlet pressure for test cases with both pressure ratios. Similarly, it was found that the 90% of inlet total pressure reduction at high flow control port was sufficient to switch the flow from low to high flow state.

The observations overall showed that the RSM preformed better than the SA model. A detailed analysis was carried out to find out the reasons due to which the
SA model showed poor agreement. The conclusions from comparison of numerical results due to the SA model and RSM with experimental data and observations are as follows

- For the test case with pressure ratio 1.54, the SA model predicted a vortex flow field in the vortex chamber for high flow state predictions. Unlike the RSM predictions, the presence of the vortex flow field in the SA model predictions, led to large pressure resistance to flow and hence poor agreement with measured data. It is believed that the same vortex flow field prediction can be attributed to the failure of the SA model to predict the high flow state in transient start-up for test case with a pressure ratio of 2.48.

- The RSM showed better agreement than the SA model for the low flow state case for a pressure ratio of 1.54. The higher magnitude of swirl velocity in the vortex chamber could be attributed to the poor predictions of the SA model. Detailed analysis of Reynolds stresses in the vortex chamber showed that the RSM predicted the general flow feature of confined vortex flow such as lower magnitude of Reynolds stresses whereas, the SA model was limited by the eddy viscosity hypothesis.
Chapter 7

Dynamic Switching

Characteristics of an SVV

In the numerical simulations described in chapter 6, section 6.4.1b and 6.4.1d, the flow state of the device was switched from one state to another by reducing the control port static pressure with respect to time. In experiments the device was switched by momentarily venting the air into the atmosphere, through the control port. This switching procedure fails to provide important information regarding the minimum pressure reduction required at the control port in order to ensure switching of the flow state. The applied pressure at the control port is important as it affects the ability of the device to switch or remain in a particular state. Hence, in order to understand the effect of the magnitude of pressure applied at the control port on the ability of device to switch the flow state, several numerical simulations were conducted and the results of those simulations are reported in this chapter.

Feikema & Culley (2008) performed several unsteady numerical simulations on a fluidic actuator. This fluidic actuator is similar in operation to the SVV except that in the SVV control flow is vented whereas, in the fluidic actuator, control flow
was forced onto the nozzle jet to change the state. The numerical study pointed out that the square root of the momentum flux ratio between the control port flow and the nozzle flow is a controlling parameter in order to produce reliable switching of flow. The square root of the momentum flux ratio is described by the following equation

$$\sqrt{J} = \sqrt{\frac{\rho_n v_n^2 A_n}{\rho_c v_c^2 A_c}}$$  \hspace{1cm} (7.1)

where, $\rho$ is density, $v$ is velocity magnitude of fluid and $A$ is area. The subscripts ‘$n$’ and ‘$c$’ represent nozzle and control port, respectively. In the numerical simulations Feikema & Cully calculated the value of square root of the momentum flux ratio ($\sqrt{J}$) was required for the control port flow to achieve successful switching in the device and found out that the critical magnitude of this parameter was approximately 4. Heo et al. (2010) conducted unsteady numerical simulations on a fluidic valve which is similar in operation and geometry to the device studied by Feikema & Cully. The objective of the study was to understand the dynamics of the flow switching in the device by investigating the effect of static pressure and mass flow rate of control flow at the control port. The numerical study showed that for fluidic device like those tested by Feikema & Cully a critical control pressure magnitude exists which clearly distinct the ability of device to switch the flow or not. If the control pressure is less than the critical pressure the device fails to switch the flow whereas, if the control pressure is more than the critical pressure device successfully switches. Heo et al. (2010), also studied the effect of size of the control port on the switching ability of the device. Their numerical simulations and experiments confirmed that with the reduction in the control port size the pressure required at the control port to switch the flow increases.

The numerical and experimental studies conducted by Feikema & Cully and Heo et al. shed light on the importance of the flow parameters such as control pressure and control flow mass flow rate required at the control port in order to achieve
reliable switching of the state. Considering these observations, several numerical simulations were conducted to understand the dynamics of flow switching and the importance of the flow parameters for the SVV namely, control pressure and control flow mass flow rate.

7.1 The objective of the study

In this study the major objective was to understand the effect of control pressure and in turn control flow mass flow rate on the ability if the SVV to switch the flow from high to low state and vice versa. This study is analogous with the work carried out by Feikema & Cully and Heo et al., however different in switching procedure. Feikema & Cully and Heo et al. injected control flow (compressed air) to switch the nozzle jet, whereas in this study the control flow was vented to atmosphere. In this numerical study, the flow was switched from high to low state according to section 6.4.1b and from low to high flow state according to section 6.4.1d. With the help of this study, it was possible to identify the critical magnitude of pressure which governs the switching ability of the device. The magnitude of control flow and nozzle flow mass flow rate was also monitored and the square root of the momentum ratio was calculated.

7.2 Numerical Setup

Considering the agreement of numerical results using the RSM with experimental data, shown in the chapter 6, in this study only the RSM was used to model the flow turbulence. For all simulations, the spatial discretization was 2\textsuperscript{nd} order. The solutions were advanced in time using a 2\textsuperscript{nd} order implicit method. FLUENT’s coupled implicit pressure based approach was used to solve the equations. For all simulations, 5 inner iterations was performed at each discrete time point to find the next time step flow solution. Time steps of \(1\times10^{-05}\) s for test case with
pressure ratio of 1.54 and $5 \times 10^{-06}$ s for test case with pressure ratio of 2.48 were used. In order to resolve the near wall region, mesh $\Delta y^+$ values were kept near to 1 at the near wall points. In this study the flow through time was estimated to be about 0.04 s for test case with pressure ratio 1.54 and 0.035 s for the test case with pressure ratio 2.48. The mesh dependency study carried out in chapter 6 revealed that the mesh size of 0.5 million was sufficient for flow predictions and hence in this study for all test cases, the 0.5 million mesh was used to model the device.

7.3 Boundary conditions

The simulations considered in this study can be divided into two parts mainly, dynamic flow predictions of flow switching from high to low flow state and low to high flow state. The boundary conditions adopted in this study are similar to those reported in chapter 6, where the transient pressure boundary condition was applied at either of the control ports, depending upon the switching. In this study, the effect of the rate of pressure reduction at control port on the switching behavior was also considered. The following equation represents the pressure reduction considered in this study.

$$p_c = p_{cs} - \left\{ \frac{p_{cs} - 0.7P_{in}}{C \ast T_f} \right\} t$$

(7.2)

where ‘$p_c$’ is control port static pressure during the simulation, ‘$p_{cs}$’ is time averaged static pressure at the control port before switching, ‘$P_{in}$’ is total inlet pressure, ‘$T_f$’ is flow through time, ‘$t$’ is time of simulation with magnitude of $0.5T_f$, $T_f$ and $2T_f$, and ‘$C$’ is constant with magnitude of 0.5, 1 and 2, representing the different rates of pressure reduction at control port, as shown in Figure 7.1.
7.4 Results

7.4.1 Switching from high to low flow state

The first scenario considered in this study was switching from high to low flow state for test cases with pressure ratio of 1.54 and 2.48. The high flow state flow solutions were considered as the initial flow conditions in the fluid domain. The next step was to reduce the control pressure to 70\% of total inlet pressure, at different rates as explained in Equation 7.2. In all the cases, flow was successfully switched to low flow state at the end of simulations. In all simulations, the square root of the momentum flux ratio ($\sqrt{J}$) was monitored and its variation with respect to control pressure was observed. The main emphasis in this study was to pin point the magnitude of $\sqrt{J}$ at the low flow control port for which flow switching occurs.

7.4.1a Test case with pressure ratio 1.54

Figure 7.2 (a) shows the contours of velocity magnitude at the mid-plane of the device for the test case with pressure ratio 1.54 with a stable high flow state.
Figure 7.2 (b) shows the switch to low flow at $t = 0.5T_f$ with $C = 0.5$. The device successfully predicted the flow switching after $t = 0.5T_f$. However, it was observed that the flow had already switched before the flow simulation time reached $0.5T_f$. This shows that the flow within the device reacts to the control pressure reduction well before it reduces to the 70% of total inlet pressure.

![Figure 7.2: Contours of velocity magnitude (m/s) at mid-plane of the device for test case with pressure ratio 1.54 with C = 0.5](a) Stable high flow state at $t = 0$ (b) stable low flow state after $t = 0.5T_f$

Figure 7.3 shows the flow field within the device while switching for different flow times. At $t = 0.435T_f$, figure 7.3(d) clearly showing the switching of the flow before the control port pressure could reach to 70% of inlet pressure. Further numerical experiments were carried out to investigate the effect of control pressure on switching of flow within the device.

Figure 7.4(a) shows the flow state in the device after $t = 0.375T_f$ from the initial flow solution shown in figure 7.2 (a). At this flow time the nozzle jet has started to shift from the high to low flow passage, indicating the commencement of flow switching. In this this simulation the control port static pressure was limited to 75.7% of inlet total pressure. At $t = 0.375T_f$ (with $C = 0.5$) the low flow control
port boundary condition was changed to no-slip condition from unsteady static pressure, mimicking the instantaneous shut-off of solenoid valve. After changing the control port boundary condition the unsteady flow simulation was continued.

Figure 7.4(b) shows the flow state within the device after a single flow through time obtained from flow state shown in Figure 7.4(a). This shows that the flow returned to the high flow state, signifying that the reduction in control port pressure to 75.7% of inlet total pressure is not enough to change the flow state.

Figure 7.4(c) shows the flow state (for $C = 0.5$) in the device after $t = 0.395T_f$ from the initial flow solution shown in figure 7.2 (a). At this flow state the bending of the nozzle jet is clear which was not observed in Figure 7.4(a). At this flow time, the
low flow control port boundary conditions was then changed to no-slip. The low flow control port static pressure was 73.7% of total inlet pressure and simulation was further continued until stable flow condition was achieved. Figure 7.4(d) shows the flow state within the device after after a single flow through time obtained from flow state shown in Figure 7.4(c). The flow field clearly shows switching to low flow state.

These simulations quantify the minimum pressure reduction required at the low flow control port which is \( \approx 74\% \) of inlet total pressure in this test case. The result of the simulation suggests that once the nozzle jet gains sufficient inertia, the control flow has less effect on the device’s ability to switch the flow.

![Figure 7.4: Contours of velocity magnitude (m/s) at mid-plane of the device for test case with pressure ratio 1.54 with \( C = 0.5 \) (a) flow state at \( t = 0.375T_f \) with \( p_c \) limited to 75.7% of \( P_{in} \) (b) flow state at \( t = T_f \) obtained from 7.4(a) (c) flow state at \( t = 0.395T_f \) with \( p_c \) limited to 73.7% of \( P_{in} \) (d) flow state at \( t = T_f \) obtained from 7.4(c)](image)

Figure 7.5 shows the variation of \( \sqrt{J} \) with respect to computational time for simulation results shown in Figure 7.2. The reduction in magnitude of \( \sqrt{J} \) with time indicates the increase in mass flux at the control port.

The results shown in Figure 7.4 can be used to identify the critical magnitude of \( \sqrt{J} \) at which reliable and stable flow switching can be obtained. Based on the
flow switching shown in Figure 7.4(d) the variation of $\sqrt{J}$ can be divided in to two regions, namely no switching and stable switching. The no switching region signifies that shutting off the control flow in this region will result in stable high flow as shown in Figure 7.4(b). The stable switching region signifies that in order to switch the flow, no further control flow is required. In this region the flow is able to switch due to the nozzle jet inertia itself. The critical magnitude of $\sqrt{J}$ at which the curve shown in Figure 7.5 can be divided in to two separate regions, is found to be approximately 2.73. In the numerical studies conducted by Feikema & Cully on the fluidic actuator, this parameter was approximately 4. The differences in computed magnitude of $\sqrt{J}$ can be attributed to following reasons

1. The geometry of fluidic device considered in this study is different compared with the fluidic actuator experimented by Feikema & Cully.

2. Feikema & Cully switched the nozzle jet by injecting the control flow, whereas in this study air was extracted through the control port to switch the flow. The higher magnitude of $\sqrt{J}$ compared to computed by Feikema & Cully, signifies that present device needs lower non-dimensional momentum-flux at the control port to switch the flow.

3. Feikema & Cully’s test case had a pressure ratio of 1.1 which is substantially different than the present simulated test case.

It may be noted that a higher magnitude of $\sqrt{J}$ for the present device suggests that the method of venting air through the control port in order to switch the flow is more efficient than injecting the air into device. Based on these results less control flow is required to switch the SVV flow compared with Feikema & Cully’s fluidic actuator.

The numerical studies were further extended to understand the effect of rate of pressure reduction at the control port. In the results of simulations reported below, the control port pressure was reduced to 70% of inlet total pressure in
flow times of $T_f$ and $2T_f$ with $C = 1$ and 2, respectively. Both simulations successfully predicted the flow switching to low flow state which suggest that the rate of pressure reduction has no effect on the switching ability of the device. As in simulations for $C = 0.5$ the control pressure and $\sqrt{J}$ was measured. Tables 7.1 and 7.2 show the results of simulations of the intermediate flow states.

Table 7.1: Results of intermediate flow simulations obtained from simulation with $T_f$ flow time for $C = 1$

<table>
<thead>
<tr>
<th>Flow time</th>
<th>$p_c/P_{in}$</th>
<th>$\sqrt{J}$</th>
<th>Remark</th>
</tr>
</thead>
<tbody>
<tr>
<td>$0.72T_f$</td>
<td>0.741</td>
<td>2.97</td>
<td>No switching</td>
</tr>
<tr>
<td>$0.74T_f$</td>
<td>0.739</td>
<td>3.34</td>
<td>Stable switching</td>
</tr>
</tbody>
</table>

The results shown in Table 7.1 and 7.2, indicate that the flow can be switched before the control port pressure reaches to 70% of inlet pressure, this is in agreement with the intermediate flow states obtained from simulations with $0.5T_f$ flow time. Irrespective of the rate of pressure reduction at control port the device showed...
stable switching at approximately 74% of inlet pressure. However, the magnitude of $\sqrt{J}$ varied for different switching rates.

Figure 7.6 shows the variation of control pressure applied at low flow control port for different rates. The study indicates that the graph can be divided into two regions, namely ‘No switching’ and ‘stable switching’. The ‘no switching’ region indicates that irrespective of rate of pressure reduction at control port, the shutting off of control flow results in stable high flow state i.e. no switching of flow. Similarly, the ‘stable switching’ region highlighted in blue colour, indicates that if the control pressure reduces more than $\approx 74\%$ the flow state changes to low flow and no further control flow is required.

![Figure 7.6: Variation of control pressure with different flow through times with clear distinction of ‘No switching’ and ‘Stable switching’ regions](image-url)
7.4.1b Test case with pressure ratio 2.48

The study reported in section 7.4.1a was further extended to the test case of pressure ratio 2.48. In this study the low flow control port pressure was again reduced to 70% of inlet total pressure through flow times of $0.5T_f$, $T_f$ and $2T_f$. All simulations with different flow times, showed the switching of the flow to low flow state near to 70% reduction. Tables 7.3, 7.4 and 7.5 show the results of the simulations for flow time $0.5T_f$, $T_f$ and $2T_f$, respectively with the magnitude of critical ratio of $p_c/p_{cs}$ and $\sqrt{J}$.

The ratio $p_c/p_{cs}$ and $\sqrt{J}$ showed little variation with respect to the rate of pressure change at the control port for simulations with $T_f$ and $2T_f$. Both simulations predicted reliable switching at $\approx 70\%$ of inlet pressure. Comparing these results with the results obtained in section 7.4.1a for test case of pressure ratio 1.54, the following observations are noted

1. With the increase in pressure ratio, the magnitude of pressure reduction at control port required for reliable switching increases.

2. The magnitude of $\sqrt{J}$ noted for the stable switching condition showed variation with respect to pressure ratio for all flow simulation time.

Table 7.3: Results of intermediate flow simulations obtained from simulation with $0.5T_f$ flow time for $C = 0.5$

<table>
<thead>
<tr>
<th>Flow time</th>
<th>$p_c/P_{in}$</th>
<th>$\sqrt{J}$</th>
<th>Remark</th>
</tr>
</thead>
<tbody>
<tr>
<td>$0.42T_f$</td>
<td>0.719</td>
<td>2.88</td>
<td>No switching</td>
</tr>
<tr>
<td>$0.44T_f$</td>
<td>0.715</td>
<td>3.35</td>
<td>Stable switching</td>
</tr>
</tbody>
</table>
Chapter 7. Dynamic Switching Characteristics of an SVV

Table 7.4: Results of intermediate flow simulations obtained from simulation with $T_f$ flow time for $C = 1$

<table>
<thead>
<tr>
<th>Flow time</th>
<th>$p_c/P_{in}$</th>
<th>$\sqrt{J}$</th>
<th>Remark</th>
</tr>
</thead>
<tbody>
<tr>
<td>$0.94T_f$</td>
<td>0.707</td>
<td>4.03</td>
<td>No switching</td>
</tr>
<tr>
<td>$0.96T_f$</td>
<td>0.704</td>
<td>4.43</td>
<td>Stable switching</td>
</tr>
</tbody>
</table>

Table 7.5: Results of intermediate flow simulations obtained from simulation with $2T_f$ flow time for $C = 2$

<table>
<thead>
<tr>
<th>Flow time</th>
<th>$p_c/P_{in}$</th>
<th>$\sqrt{J}$</th>
<th>Remark</th>
</tr>
</thead>
<tbody>
<tr>
<td>$1.90T_f$</td>
<td>0.707</td>
<td>4.14</td>
<td>No switching</td>
</tr>
<tr>
<td>$1.91T_f$</td>
<td>0.706</td>
<td>4.59</td>
<td>Stable switching</td>
</tr>
</tbody>
</table>

7.4.2 Switching from low to high flow state

This section presents the results of numerical simulations conducted on flow switching from low to high flow state for pressure ratios of 1.54 and 2.48. In this study, the stable low flow state was considered as the initial flow conditions in the flow domain. In the simulations the flow was switched from low to high flow state by reducing the static pressure at high flow control port 70% of inlet total pressure as per equation 7.2. The numerical study reported in section 7.4.1 showed that the effect of different rates of change of static pressure at the control port has very little effect on the overall switching mechanism, hence in this study the only a single flow through time was used to reduce the control pressure. For both test cases with pressure ratio 1.54 and 2.48, the flow was switched well before high flow control port static pressure reached to 70% of inlet pressure. In this study, the variation of $\sqrt{J}$ was computed with respect to flow time.
7.4.2a Test case with pressure ratio 1.54

Figure 7.7 (a) shows the contours of velocity magnitude at mid-plane of the device for test case of pressure ratio 1.54 (with $C = 1$) for a stable low flow state. Figure 7.7 (b) shows the switched flow state from low to high flow state within a single flow through time. The simulation successfully predicted the flow switching after $T_f$. However it was observed that the flow switched to the high flow state well before low flow control port pressure reached to 70% of inlet pressure.

![Figure 7.7: Contours of velocity magnitude (m/s) at mid-plane of the device for test case with pressure ratio 1.54 after (with $C = 1$) (a) $t = 0$ (b) $t = T_f$](image)

Figure 7.8 shows the contours of velocity magnitude at different flow time obtained during switching procedure. Figure 7.8 (d) shows that the flow is switched to the high flow state well above 70% of inlet pressure. At this flow time ($t = 0.4T_f$) the ratio of control port pressure ($p_c$) to inlet total pressure ($P_{in}$) was 0.84.

Further numerical simulations were conducted in order to determine the critical pressure drop required at the control port for reliable switching. Figure 7.9 (a) shows the flow field within the device after $0.265T_f$ simulation time obtained from
the stable low flow state shown in figure 7.7 (a). This shows a deflection of the nozzle jet towards the high flow passage, indicating the flow is starting to switch. At this simulation time, the boundary condition of the high flow control port was switched to no-slip from transient static pressure. As explained previously, this procedure mimics the instantaneous closure of solenoid valve. The simulation then continued for a single flow through time. At $t = 0.265T_f$ the ratio $p_c/P_m$ was $\approx 0.88$ and the magnitude of $\sqrt{J}$ was $\approx 3.3$. Figure 7.9 (b) shows the predicted flow field at $t = 1.265T_f$ with no-slip boundary condition at the high flow control port.
With the sudden shut-off of control flow the device returned to low flow state. This signifies that though the nozzle jet showed deflection in the device, the nozzle jet could not switch to the other passage due to insufficient inertia. This simulation suggests that with the control port pressure more than or equal to 88% of inlet pressure sudden closing of solenoid valve would not result in flow switching.

Figure 7.9 (c) shows the flow solution obtained at $t = 0.272T_f$ during the switching process. As expected the nozzle jet shows slightly more deflection than in figure 7.9(a). The simulation was continued from this point, changing the boundary condition at the control port to no-slip. At this flow time the ratio $p_c/P_{in}$ was $\approx 0.87$ and the magnitude of $\sqrt{J}$ was $\approx 4$. Figure 7.9 (d) shows the predicted flow field after a further flow through time. The figure shows the flow successfully switched to the high flow state without any control flow. From these simulations it can be concluded that for control port pressure less than 87% the device can be switched to high flow state without any further control flow.

**Figure 7.9:** Contours of velocity magnitude (m/s) at mid-plane of the device for test case with pressure ratio 1.54 (with $C = 1$) (a) flow state at $t = 0.265T_f$ with $p_c/P_{in}$ limited to 0.88 (b) flow state at $t = 1.265T_f$ with no-slip high flow control port boundary (c) flow state at $t = 0.272T_f$ with $p_c/P_{in}$ limited to 0.87 (d) flow state at $t = 1.272T_f$ with no-slip high flow control port boundary
Figure 7.10 shows the variation of $\sqrt{J}$ with respect to the flow time. With the help of numerical results shown in figure 7.9, the variation of $\sqrt{J}$ can be broadly divided into two parts, namely ‘no switching’ and ‘stable switching’. The region ‘no switching’ signifies that the device fails to switch the flow if control flow is reduced to zero whereas, ‘stable switching’ represents the state of device where no control flow is required for flow switching.

In this test case, $\sqrt{J}$ showed several peaks in its variation. These peaks were not found in the previous test case reported in figure 7.5. The peak occurring at $t \approx 0.5T_f$ is particularly strong. Figure 7.11 shows the contours of velocity magnitude at $t = 0.5T_f$. At this flow time, the device has switched to the high flow state. As shown in the figure, the attachment of nozzle jet with the high flow passage walls results in a separation bubble. As this separation bubble forms the flow through the high flow control port nearly dropped to zero. Since, $\sqrt{J}$ is a ratio, this results in a peak in its variation with time. The other peaks can also be explained by intermittent attachment and detachment of the jet from the high flow passage wall.

![Figure 7.10: Variation of $\sqrt{J}$ with simulation time for test case pressure ratio 1.54](image-url)
7.4.2b Test case with pressure ratio 2.48

The numerical study reported in the section 7.4.2a was further extended to a pressure ratio of 2.48. In this study, the high flow control port pressure was reduced to 70% of inlet pressure in a single flow through time. Figure 7.12 shows the flow switching of the device after $T_f$ flow time.

Several further simulations were conducted to understand the switching ability of the device. The control flow was switched-off at different flow times and simulations were continued with a no-slip boundary condition at high flow control port. Table 7.6 shows the results of simulations conducted corresponding to the test case shown in figure 7.12. For this test case, the simulations predicted stable switching at lower $p_c/P_in$ compared to the low to high flow switching test case with pressure ratio 1.54. The higher values of $\sqrt{J}$ at which switching occurs compared to the values for a pressure ratio of 1.54, suggest that at higher pressure ratio the device needs less pressure reduction at the control port. This observation is supported by the fact that the nozzle jet is biased (shown in section 6.2.1) towards the high flow state. Due to the biased design, the nozzle jet experiences less resistance to
switching the flow from low to high flow state compared to high to low flow state, with the increase in the pressure ratio.

**Table 7.6:** Results of intermediate flow simulations obtained from simulation with $T_f$ flow time

<table>
<thead>
<tr>
<th>Flow time</th>
<th>$p_c/P_{in}$</th>
<th>$\sqrt{J}$</th>
<th>Remark</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.11$T_f$</td>
<td>0.91</td>
<td>6.4</td>
<td>No switching</td>
</tr>
<tr>
<td>0.12$T_f$</td>
<td>0.90</td>
<td>6.9</td>
<td>Stable switching</td>
</tr>
</tbody>
</table>

Figure 7.13 shows the variation of $\sqrt{J}$ measured for this test case. The numerical results shown in table 7.6 can be used to divide this variation into ‘no switching’ and ‘stable switching’ regions. In this test case, as in the previous test case (7.4.2a) a sharp peak $\sqrt{J}$ was observed occurring at $\approx t = 0.4T_f$. This sharp peak can be again attributed to the reduction of control flow due to attachment and detachment of the jet to the high flow passage wall.
Figure 7.13: Variation of $\sqrt{J}$ with simulation time for test case pressure ratio 2.48

Figure 7.14 shows the contours of velocity magnitude at $t = 0.4T_f$ obtained from the simulation reported in figure 7.12. The switching of the flow is clearly evident. The attachment of the nozzle jet to the lower high flow passage wall results in a separation bubble near the high flow control port. This reduced the flow through the control port and hence results in the sudden peak as shown in the figure 7.13.
7.5 Conclusion

In this study the dynamics of flow switching associated with the SVV was numerically simulated. Two different flow switching procedures were simulated in this work which are switching of high flow to low flow state and vice versa, for pressure ratio 1.54 and 2.48. In experiments the flow was switched by venting the air through the respective control port. This procedure was simulated with using transient pressure boundary condition at the control port. The control pressure was transiently dropped to 70% of inlet pressure at different rates. The response time of the nozzle jet for different rates of pressure reduction was investigated. The following conclusions can be derived from the study

- A qualitative comparison of $\sqrt{J}$ obtained in this study with work of Feikema & Culley (2008), suggest that the switching of flow in fluidic device by venting the air through control port is more efficient than injecting the air.
• In the case of switching from high to low flow state the device showed little effect of the rate of pressure reduction at the low flow control port.

• For any given pressure ratio the magnitude of $\sqrt{J}$ at which switching occurred was found be higher for low to high flow state switching than high to low state. This suggests that due to the biased nozzle design, the nozzle jet finds less resistance to switch the flow from low to high.
Chapter 8

CFD study of the effect of vibration on the performance of the device

During the flight cycle, aero-engine components are subjected to several dynamic forces resulting in vibration. These dynamic forces are due to different reasons such as centrifugal force due to unbalanced rotors and aero-elastic effects on the blade rows (Wu & Imregun (2005)). According to DO-60 (Environmental Conditions and Test Procedures for Airborne Equipment) norms the aircraft manufacturer has to check the effects of vibration and the ability of systems to operate during all vibration scenarios.

According to Boeing (2008), the different vibration scenarios can be broadly divided into normal and abnormal vibrations. The normal vibration of an aero-engine is due to the mass distribution and structural stiffness that results in vibration modes at different frequencies. Whereas, the abnormal vibration is due to malfunction of mechanical equipment where the equipment loses the structural integrity due to incidents such as bird strike. In industry, both these scenarios are
simulated experimentally to verify the ability of different systems to operate in such different vibration conditions. However, it is also possible to simulate these conditions using CFD methods. A detailed review of CFD application for aero-engine flows carried by Chew & Hills (2009), presented the current advancement of CFD methods for understanding aero-elastic problems. The review pointed out that, the aero-elastic methods which consist of CFD and FEA are used to study the effects of vibration on flow and vice versa. These method are of interest where the structure changes its dimension due to the flow and vibration effects. However, in some engineering problems only the effects on flow due to vibration are of interest and effects of flow on structure are neglected. For example, Elkholy (1997) performed numerical simulations to understand the flow behavior in a pipe under sinusoidal oscillations. In his simulations, he imposed the sinusoidal motion on the walls of a pipe imitating the vibrational motion. A similar concept was employed by Deshpande & Barigou (2001) and Essa (2009). Deshpande & Barigou, extended the studies of Elkholy to non-Newtonian fluids and Essa studied the heat transfer in Newtonian and non-Newtonian fluids in a vibrating pipe. Essa also found good agreement between the numerical results with the experimental data by Deshpande & Barigou. The above publications, suggest that the CFD methods can be extended to the complex problems in which vibration is coupled with flow physics.

8.1 Objective of the study

The SVV device studied in this research is envisaged to be part of secondary air system of aero-engine. The operating environment of secondary air system is dynamic in nature due high speed rotating components. Hence, it is necessary to examine the ability of the device under different operating conditions, before it is included in the final end product. In order to verify the device design under vibration effect, Rolls-Royce conduct a vibration test bed known as a ‘Shaker study’.
In this study any proposed device is experimented under forced vibration with maximum acceleration of $\approx 0.35 \, m/s^2$ in all three directions at a time. During the test the device performance is monitored, if the device performance substantially changes due to vibration then the device goes through redesign procedures otherwise it is certified as safe for engine vibrations.

In this study, an attempt was made to simulate the test conditions of the shaker study. The main objective of the study was to examine the performance of the device under different vibration conditions. These conditions include forced vibration. The primary emphasis was on understanding the changes in air mass flow rate through the device and stability of flow state under different vibration scenarios.

### 8.2 Numerical set up

In forced vibration the body is forced with external periodic acceleration. The simplest form of forced acceleration is, described by following equation

$$\ddot{y} = A\omega^2 \cos \omega t \quad (8.1)$$

where $y$ is displacement, $A$ is amplitude of vibration, $\omega$ is frequency of vibration in rad/s ($\omega = 2\pi f$) and $f$ is frequency of vibration in Hz. Integrating the above equation results in an equation describing sinusoidal variation of velocity with respect to time as follows

$$\dot{y} = A\omega \sin \omega t \quad (8.2)$$

Integrating equation 8.2 gives an equation describing periodic displacement (shown in figure 8.1) as follows
\[ y = -A \cos \omega t \] (8.3)

In FLUENT, dynamic mesh facility enables to simulate the flows where the shape of the domain changes or the domain shifts its spatial location with time. Here, the dynamic mesh was used to simulate the vibration motion by applying periodically varying velocity to the domain walls. The spatial displacement of the wall nodes was computed from the the assigned velocity vector and the motion of the all remaining nodes is determined by a spring based smoothing method. In this method, the edges between mesh nodes are idealized as a network of interconnected springs where the spring stiffness can be adjusted by the value of the Spring constant factor between 0 and 1. In this problem, since the whole domain is displacing with time, the Spring constant was taken as 0 which signifies that the there is no damping on the spring or infinite stiffness.

Figure 8.1 shows the imposed sinusoidal motion on the wall of the domain in ‘y’ direction. In this study, in all simulations, only vibration in ‘y’ direction was considered. In chapter 7, it was found that the flow state of the device is depends upon the exchange of momentum in ‘y’ direction. Hence, it is interesting to examine the flow state stability of the device under the influence of momentum created in ‘y’ direction due to forced vibration.

Table 8.1 shows the CFD model set up. The test case with pressure ratio 2.54 in the high flow state was considered. The comparison of numerical results predicted by the RSM with the experimental data reported in Chapter 6 was very encouraging. Hence, in this study only the RSM was used.

The vibration phenomenon is time dependent, hence unsteady flow simulations were required. For all simulations, the spatial discretization was 2nd order except for Reynolds stresses and dissipation rate equation due to solution convergence issue. The solutions were advanced in time using a 1st order implicit method. The
Chapter 8. CFD study of effect of vibration on the performance of the device 159

Figure 8.1: The direction of vibration imposed on the SVV

<table>
<thead>
<tr>
<th>Item</th>
<th>Descriptions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solver</td>
<td>FLUENT 6.3, Pressure based unsteady solver</td>
</tr>
<tr>
<td>Fluid</td>
<td>Air, compressible, Operating pressure=101325 Pa</td>
</tr>
<tr>
<td>Pressure-velocity coupling</td>
<td>Coupled</td>
</tr>
<tr>
<td>Discretization</td>
<td>Second order for all equations (except Reynolds stresses and dissipation rate)</td>
</tr>
<tr>
<td>Time step</td>
<td>5e-06 s</td>
</tr>
<tr>
<td>Turbulence modeling</td>
<td>RSM (Linear pressure strain without wall effect)</td>
</tr>
<tr>
<td>Inlet boundary condition</td>
<td>Total inlet absolute pressure (251314 Pa) Turbulence intensity = 5%</td>
</tr>
<tr>
<td>Outlet boundary condition</td>
<td>Pressure outlet. A gauge pressure of 0 Pa for all simulations</td>
</tr>
<tr>
<td>$y^+$</td>
<td>&lt; 1 (Resolved near wall)</td>
</tr>
</tbody>
</table>

Table 8.1: CFD model set up

coupled implicit pressure based approach was used to solve the equations. For all simulations, 5 inner iterations was performed at each discrete time point to find the next time step flow solution. A time step of $5 \times 10^{-06}$ s was used. In order to resolve the near wall region, mesh $\Delta y^+$ values were kept near to 1 at the near wall points. In this study the flow through time was estimated to be about 0.04 s for all the simulations. The mesh dependency study carried out in chapter 6 revealed that the mesh size of 0.5 million was sufficient for flow predictions and hence in
this study for all simulations, 0.5 million mesh was used for the vibration study.

Figure 8.2 shows the sinusoidal variation of $y$ velocity with time which was imposed on the walls of the SVV domain. In this test case model was simulated for 20 harmonic cycles which complete in 4 through flow time. A stable high flow state solution was used as initial flow conditions in the fluid domain.

8.3 Results

8.3.1 $a_{max} = 0.5g$ test case

In this test, the device was simulated for maximum acceleration ($a_{max}$) of $0.5g$, where $g$ is gravitational constant ($9.81 \text{ m/s}^2$). The amplitude was computed from the following equation,

$$A = \frac{0.5g}{\omega^2} = 0.007 \text{ mm} \quad (8.4)$$

where $\omega$ represents the rotational speed of the intermediate pressure turbine which is $\approx 785.4 \text{ rad/s}$. Though the device will be subjected to vibration at various frequencies, considering the proposed location of the SVV installation, it is assumed that the dominating frequency of vibration would be equal to IPT speed.

Figure 8.3 shows the contours of instantaneous velocity at midplane of the device in the $xy$-plane for different times. Figure 8.3(a) shows the stable high flow state which was considered as initial flow solution for this test case and figure 8.3(b) shows the state of flow after 20 cycles of vibration or $4T_f$.

The obvious observation is that the flow state remained stable despite the imposed vibration. During the simulations non-dimensional air mass flow rate ($\dot{m}_{\text{non}}$) flowing through was monitored. The non-dimensional air mass flow rate ($\dot{m}_{\text{non}}$) is defined according to the following equation,
Chapter 8. **CFD study of effect of vibration on the performance of the device** 161

![Graph](image)

**Figure 8.2:** Variation of imposed y velocity for a single through flow time

![Contours](image)

**Figure 8.3:** Contours of instantaneous velocity (m/s) at midplane of the device in xy-plane (a) t = 0 (b) t = 4Tf

\[
\dot{m}_{non} = \frac{\dot{m}}{\dot{m}_{ref}}
\]  

(8.5)

where \(\dot{m}\) is the numerically predicted mass flow rate and \(\dot{m}_{ref}\) is mass flow rate of air measured for pressure ratio 2.48 for high flow state. For the stable high flow state RSM predicted \(\dot{m}_{non}\) approximately to 0.936 (For details see section 6.6.4).
In this test case for the high flow state with imposed vibration RSM predicted $\dot{m}_{non}$ approximately 0.934. The comparison reveals that the imposed vibration has negligible effect of the performance of the device.

8.3.2 $a_{max} = 10g$ test case

In this test, the device was simulated for the maximum acceleration ($a_{max}$) of 10$g$ which higher in magnitude than the required test conditions of a ‘shaker study’. The objective of this test case was to understand the performance of the device under high acceleration conditions. The amplitude of vibration was computed from the equation 8.4 which is approximately 0.16 mm.

Figure 8.4 shows the contours of instantaneous velocity at midplane of the device in xy-plane for different flow time. Figure 8.4(a) shows the predicted stable high flow state which was considered as initial flow solution for this test case and figure 8.4(b) shows the state of flow after 20 cycles of vibration or $4T_f$.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{figure8_4.png}
\caption{Contours of instantaneous velocity (m/s) at midplane of the device in xy-plane (a) $t = 0$ (b) $t = 4T_f$}
\end{figure}

Like the previous test case, in this test case the flow state remained stable despite the imposed vibration. The $\dot{m}_{non}$ was monitored and found approximately to 0.932
which is only 0.4% offset from the stable high flow test case without any vibration. In conclusion it was found that the imposed vibration (of $a_{\text{max}} = 10g$) also has negligible effect on the performance of the device.

### 8.4 Conclusion

In this study an attempt was made to demonstrate the applicability of CFD methods to simulate complex working conditions of an aero-engine. Vibration with defined frequency was simulated for high flow test case with pressure ratio of 2.48, by using the dynamic mesh approach. The two vibration scenarios were considered in this study namely, 0.5$g$ and 10$g$ maximum acceleration. The following conclusions can be drawn from the study:

- The device showed stability for both test cases, namely 0.5$g$ and 10$g$.
- In both test cases, the mass flow rate through the device was measured and found to be negligibly affected by the imposed displacement.
- The methodology can be extended to other operating conditions replacing the assumptions made here with actual parameters. Hence, this methodology may save valuable time from in the product design cycle.
Chapter 9

Conclusion and future work

This research focused on the application of CFD for better understanding of the flow behavior in an SVV device. The literature review revealed that based on the principle of operation, the wall attached fluidic devices and vortex amplifier are relevant to the SVV device. Hence, the research was initiated with CFD studies to understand the flow features associated with these devices, by considering appropriate test cases. Chapter 4, presents the results of CFD studies carried out for a confined vortex chamber which is similar in operation and geometry to vortex amplifiers. The novelty of this test case is that unlike previous numerical studies, the compressibility effect was also considered. In this test case three different turbulence models were considered. The agreement shown by the RSM with the experimental data agrees with the findings of other researchers. The SA model also showed encouraging comparison while the $k$-$\epsilon$ model showed least agreement.

The study was further extended to evaluate the flow features of wall attached fluidic devices. In this study the experiments conducted by Tesař (2010) were used as a test case. The motivation behind studying this test case was its operational and geometrical similarity with the SVV device. Due to the complex geometry which comprises steps in the flow, was a challenge for turbulence models to predict
the measured flow data. Along with URANS models, LES was also employed in this study. Despite the complex flow features such as flow separation, the URANS models predicted experimental data qualitatively, however they failed to some degree quantitatively. The results predicted by LES shown least agreement. However, the mesh size used for LES simulations is questionable, since it used the same mesh size which was used for URANS simulations. The overall conclusion from the turbulence model evaluations was that the RSM was recommended for use in engineering calculations and that, used with care, this could provide useful input to design.

In chapter 6, a CFD study of the SVV device was reported. The SVV device can be conceptually divided into wall attached fluidic device and vortex amplifier. The best practices learned from the chapters 4 and 5 were implemented in this study. Unsteady flow simulations were carried out using the SA model and the RSM. The performance of these two turbulence models was examined for different pressure ratios and for different flow states. The RSM showed better comparison with experimental data than the SA model. The detailed analysis of Reynolds stresses revealed that the RSM model captures the flow features of swirling flow better than the SA model. The superiority of the RSM was also seen in numerical studies reported in chapter 4.

In chapter 7, the dynamics of flow switching was numerically examined. The objective of the study was to understand the effect of control pressure on the ability of the device to switch the flow. Considering the encouraging agreement with experimental data shown by the RSM in previous chapters, in this study for all unsteady simulations only the RSM was considered for modeling turbulence. The numerical study revealed that for the present SVV device, the ability to switch the flow depends upon the magnitude of pressure at the control port and was not affected by the rate of pressure reduction. This conclusion, will be helpful to the designers for future SVV designs.
The application of CFD was further extended to understand the performance of the device under vibrations. Using the moving mesh approach the device was simulated for vibration with approximated IPT rotating frequency. This was an attempt to simulate the dynamic environment of aero-engine. The results of simulation showed that the device has negligible effect of vibration on the performance of the device.

9.1 Suggested Future work

In this research, the numerical study carried for SVV was based on the laboratory test conditions. When the device will be implemented in engine, it will be subjected to different operating conditions. Therefore it will be interesting to know the CFD predictions for the real operating conditions.

The application of CFD for prediction of performance of the device under vibration can be further extended to the ‘Bong test’. The Bong test is a experimental method to examine the performance of the device where it is imposed with vibration of several frequencies. In this research effects due vibration with a single frequency was examined. This proposed study can replace the experimental approach by CFD using the experimental parameters.
Appendix A

Dimensions of the SVV device

Figures A.1 and A.2 list the dimensions (in mm) of the SVV device numerically tested in this research. Device’s dimensions were obtained from the University of Sheffield (Romero (2014)) along with experimental data. Further details of experiment and device configuration can be found from Romero (2014).
Appendix. *Dimensions of the SVV device*

Figure A.1: *Dimensions of the SVV device experimented by Romero (2014)*

(a) top view (b) front view
Figure A.2: Dimensions of the splitter of the SVV device
Bibliography


BOUSSINESQ, J. 1877 Essai sur la theorie des eaux courantes. mem. pres. par div. savants al’Acad.


CHEN, R, HUANG, Q & LUCAS, G 1998 Theoretical and experimental study of fluidic device as fuel injector for natural gas turbine. Proceedings of the


Heo, Jun-Young, kwang-hee Yoo, Lee, Yeol & Sung, Hong-Gye 2010 dynamic characteristics of a fluidic valve for divert and attitude control system. 46th *AIAA/ASME/SAE/ASEE joint propulsion Conference and Exhibit* pp. 3334–3347.


Saqr, Khalid M, Aly, Hossam S., Wahid, Mazlan A & Sies, Mohsin M


Bibliography


