UNIVERSITY OF TECHNOLOGY SYDNEY

Study of Ventilation Flow in a Room Induced by Solar Chimneys

By

Kashif Nazir

Supervisor :**Dr. B. Phuoc Huynh**
Co-Supervisor :**Dr. Tien Thanh Nguyen**

A Thesis Submitted to the Graduate Research School, University of Technology Sydney for the Degree of Master of Science (Research)

School of Electrical, Mechanical & Mechatronics System
Faculty of Engineering & IT, University of Technology Sydney

January, 2018

CERTIFICATE OF ORIGINAL AUTHORSHIP

I certify that the work in this thesis has not previously been submitted for a degree nor has it been submitted as part of requirements for a degree except as fully acknowledged within the text.

I also certify that the thesis has been written by me. Any help that I have received in my research work and the preparation of the thesis itself has been acknowledged. In addition, I certify that all information sources and literature used are indicated in the thesis.

Production Note:
Signature removed prior to publication.

Signature of Student:

Dated: 16-01-2018

Acknowledgements

I am extremely grateful to Dr. Phuoc Huynh for his invaluable technical guidance and attentive, friendly support throughout this degree. He guided and supported me in every aspect to achieve this target. I was also very lucky to learn from Dr. Thanh Nguyen, who helped me in every phase, whenever required, during my study. I would like to thank Management / Staff of Faculty of Engineering & Information Technology and Graduate Research School for their support and help.

I would also like to thank my wife who supported me all the time especially during this study period. I would like to dedicate this study to my wife and parents, who always prayed for me to accomplish this task.

Table of Contents

		Page No.
Contents		4
List of Figures		8
List of Tables		23
Abstract		26
Nomenclature and List of Symbols		28
Chapter – 1	**Introduction**	
1.1	History of Solar Chimney Based Research and Factors for Motivation	31
1.2	Alternate Energy Resources	35
1.3	Solar Energy	39
1.4	Chimney and Natural Ventilation	42
1.5	Solar Chimney	44
1.6	Principle and Performance of a Solar Chimney	48
1.7	Research Objectives	48
Chapter – 2	**Basic Concepts of Fluid Mechanics and Solar Chimney Literature Review**	
2.1	*Objectives of Literature Review*	53
2.2	*Theoretical Concepts*	53
2.3	*The Chimney and Natural Ventilation*	77

2.4	*Computational Fluid Dynamics (CFD)*	80
2.5	*CFD and Solar Chimney Systems*	83
2.6	*Solar Chimney Systems Modeling and Analysis*	85
2.7	*Effect of Geometry on Ventilation*	88
2.8	*Research Based on Controlled Environment*	92
2.9	*Studies Based on Experiments*	93
2.10	*Effect of Solar Chimney Angle on Performance*	94
2.11	*Outcome Of Literature Review*	96
Chapter 3	**Two-Dimensional Modeling and Simulation**	
3.0	Abstract	98
3.1	2D Modeling and Simulation of a Small Size Room	98
3.2	2D Modeling and Simulation of a Real Size Room	100
3.3	Designing of a Real Sized 2D Room in CFD GEOM	104
3.4	Simulation Steps of a Real Sized 2D Room in CFD ACE+ GUI (Software Solver)	105
3.5	Physical Conditions for the Model and Simulation in the CFD Solver	106
3.6	Simulation and Analysis of First (Basic) Model	112
3.7	Grid Convergence Study	118
3.8	Effect of Room Air Inlet Location	137

3.9	Analysis and Discussion on Simulation Results	154
Chapter 4	**Three-Dimensional Modeling and Simulation**	
4.0	Abstract	162
4.1	Designing / Modeling of a Three-Dimensional Room for Simulation	163
4.2	3D Modeling and Simulation of a Real Size Room	163
4.3	Designing of a Real Sized 3D Room in CFD GEOM	169
4.4	Physical Conditions for the Model and Simulation in the CFD Solver	171
4.5	Simulation of a Real Sized 3D Room in CFD ACE+ GUI (Software Solver)	175
4.6	Designing and Simulation of Different Room Models	177
4.7	Simulation Results and Analysis of Different Models	186
4.8	Simulation and Validation of Results	203
4.9	Grid Convergence Study	204
4.10	Effect of Room Air Inlet Location	215
4.11	Analysis and Discussion on Simulation Results	228
4.12	Study of Air Behavior and Pressure at Exit of the Solar Chimney	235
4.13	Analysis of Air Flow Rate in Terms of "Air Changes per Hour (ACH)	252

Chapter 5	**Conclusion and Discussion**	
5.1	Simulation and Validation of Results	259
5.2	Importance of Grid Convergence	260
5.3	Analysis of Simulation Results	261
5.4	Modeling and Simulation of 2D Real Sized Room	262
5.5	Modeling and Simulation of 3D Real Sized Room	263
5.6	Analysis of Air Flow Rate in Terms of “Air Changes per Hour (ACH)	265
5.7	Contribution and Achievements	266
5.8	Recommendations	268
Bibliography		269

List of Figures

Figure No.	Description	Page
Figure 1.1	Global Energy Consumption	31
Figure 1.2	LNG storage tanks and a membrane-type tanker	33
Figure 1.3	Graph showing depletion model of conventional energy resources.	34
Figure 1.4	Pollution Caused by Power Plants using Oil and Gas.	34
Figure 1.5	Example of Disasters / Accidents Caused during Exploration / Storage of Oil and Gas	35
Figure 1.6	Example of Solar Energy Utilization – Solar Panels	36
Figure 1.7	Example of Wind Energy Utilization – Wind Turbines	36
Figure 1.8	Example of Energy / Electricity production from Sea.	37
Figure 1.9	Example of Electricity Production from Water – Hydro.	37
Figure 1.10	Example of Geo Thermal Energy Plant - Nesjavellir Geothermal Power Station.	38
Figure 1.11	An example of different Biomass Sources from which Bio Energy can be produced.	38
Figure 1.12	An example of Electricity Production from Solar Panels – Bright Source Solar Power Plant in California's Mojave Dessert.	39
Figure 1.13	An example of Solar Chimney power Plant - Solar Updraft Tower System	40

Figure 1.14	An example of Water Heating based on Solar Energy – Domestic Solar Water Heater	40
Figure 1.14a	An example of Solar Chimney Installed at a Building for Natural Ventilation.	41
Figure 1.15	Chimneys, Hampton Court Palace, Surrey, UK.	43
Figure 1.16	Chimneys built for natural ventilation at The Energy and Resources Institute (TERI), Delhi, India.	43
Figure 1.17	Typical Sketch of a Room Fitted with Side Wall Solar Chimney	44
Figure 1.18	Solar Chimneys installed at the roof of Sidwell Friends School, Washington DC.	45
Figure 1.19	Solar Chimney at Urban Ecology Center, Menomonee Valley, USA.	45
Figure 1.20	Showing parts of a typical Solar Chimney	46
Figure 1.21	Different solar chimney designs, from a simple black-painted pipe to integrated Trombe roof structure	47
Figure 1.22	Showing the stack effect: hot air rises due to buoyancy, and its low pressure sucks in fresh air from outside.	47
Figure 2.1	Laminar Flow in a Pipe	56
Figure 2.2	Turbulent Flow in a Pipe	57
Figure 2.3	Laminar to Turbulent Flow Transition in a Pipe	59
Figure 2.4	Natural Chimney functionality in a multistory building	77
Figure 2.5	Standard Solar Chimney	79
Figure 3.1	Air Flow through small size room fitted with	99

	straight solar chimney	
Figure:3.2	Air Flow through a room (3 × 5 meter) fitted with straight solar chimney of 2 meter height.	100
Figure 3.3	Air Flow through Real Size room fitted with straight solar chimney	101
Figure: 3.4	Temperature pattern in a Real Size room Fitted with straight solar chimney	102
Figure 3.5	Model of a real sized room fitted with inclined solar chimney	103
Figure 3.6	Finalized Two Dimensional Model with Grids and Faces	104
Figure 3.7	Exported DTF model (Figure: 3.6) in CFD-ACE GUI Module for Simulation	105
Figure 3.8	Simulation in CFD-GUI Module of the CFD-ACE+ Solver	112
Figure 3.9	Model of a real sized room in CFD GEOM with Edges	113
Figure 3.10	Model of a real sized room in CFD GEOM with Edges and Meshes	113
Figure 3.11	Flow Pattern through 2D Real Sized Room	114
Figure 3.11a	Graph showing change in velocity at three different locations of the Model	115
Figure 3.12	Showing number of Iterations and Grid Convergence Criteria for Simulation in the Solver	116
Figure 3.13	Normalized residual plot after convergence of model at desired level	117

Figure 3.14	Un-normalized residual plot after convergence of model at desired level	117
Figure 3.14a	Graph showing change in flow rate at inlet and outlet of the model with increase in number of cells	132
Figure 3.14b	Graph showing change in temperature with increase in number of cells	132
Figure 3.14c	Graph showing change in velocity with increase in number of cells	133
Figure 3.15	Model for different inlet opening locations A) Inlet at bottom of the wall, B) Inlet at middle of the wall, C) Inlet at top of the room wall.	138
Figure 3.16	Air Flow Pattern after Simulation of Basic Model with different inlet opening locations analyzed in CFD View	139
Figure 3.17	Model with inlet opening at bottom of the side wall – Showing location of data measurement points for analysis.	142
Figure 3.18	Model with inlet opening at bottom of the side wall – Showing Velocity pattern after simulation.	143
Figure 3.19	Model with inlet opening at bottom of the side wall – Showing Temperature profile after simulation.	143
Figure 3.20	Model with inlet opening at bottom of the side wall – Showing Pressure profile after simulation.	1144
Figure 3.21	Model with inlet opening at middle of the side wall – Showing location of data measurement	146

	points for analysis.	
Figure 3.22	Model with inlet opening at middle of the side wall – Showing Velocity pattern after simulation.	147
Figure 3.23	Model with inlet opening at middle of the side wall – Showing Temperature profile after simulation.	147
Figure 3.24	Model with inlet opening at middle of the side wall – Showing Pressure profile after simulation.	148
Figure 3.25	Model with inlet opening at top of the side wall – Showing location of data measurement points for analysis.	150
Figure 3.26	Model with inlet opening at top of the side wall – Showing Velocity pattern after simulation.	151
Figure 3.27	Model with inlet opening at top of the side wall – Showing Temperature profile after simulation.	151
Figure 3.28	Model with inlet opening at top of the side wall – Showing Pressure profile after simulation.	152
Figure 3.28a	Graph showing comparison of Total Mass Air Flow Rate at Room inlet and outlet.	153
Figure 3.28b	Graph showing comparison of Air Flow Rate and Velocity with change in inlet location	155
Figure 3.29	Model with inlet opening at bottom of the side wall – Showing air flow pattern after simulation.	156
Figure 3.30	Model with inlet opening at top of the side wall	156

	– Showing air flow pattern after simulation.	
Figure 3.31	2D Room with Inlet at Bottom of the Wall : Curve showing Velocity Magnitude Pattern along the Room.	158
Figure 3.32	2D Room with Inlet at Middle of the Wall : Curve showing Velocity Magnitude Pattern along the Room.	159
Figure 3.33	2D Room with Inlet at Top of the Wall : Curve showing Velocity Magnitude Pattern along the Room.	160
Figure 4.1	Real Sized 3D Room created in CFD-GEOM	164
Figure4.2a	3D Room created in CFD-GEOM with Inlet Box	165
Figure4.2b	Defining different sections / faces of a 3D Room created in CFD-GEOM with Inlet Box	166
Figure 4.3	3D Room showing Edges and Faces created in CFD-GEOM	168
Figure 4.4	Basic 3D Room model created in CFD-GEOM using points and lines	169
Figure 4.5	Creation of box around inlet opening of the room	170
Figure 4.6	Finalized Three Dimensional Model with Grids and Faces	170
Figure 4.7	Exported DTF model (Figure: 4.6) in CFD-ACE GUI Module for Simulation	176
Figure 4.8	Room model with solar chimney on left side of the roof top	178
Figure 4.9	Room model with solar chimney at the center	178

	of the roof top	
Figure 4.10	Room model with solar chimney on right side of the roof top – in front of room air inlet opening	179
Figure 4.11	Room model with room air inlet opening at bottom right side of the wall – adjacent to room floor	179
Figure 4.12	Room model with room air inlet opening at middle right side of the wall – with bigger inlet opening	180
Figure 4.13	Change in Size of Inlet Box	180
Figure 4.14	Change in Size of Inlet Box	181
Figure 4.15	Change in Size of Inlet Box and location of inlet opening	181
Figure 4.16	Change in Size of Inlet Box	182
Figure 4.17	Change in location of inlet opening	182
Figure 4.18	Change in location of inlet opening	183
Figure 4.19	Showing number of Iterations and Grid Convergence Criteria for Simulation in the Solver	184
Figure 4.20	Graph showing Convergence of the one Model up to desired level (1.0×10^{-04}) with Normalized values	185
Figure 4.21	Graph showing Convergence of the one Model up to desired level (1.0×10^{-04}) with Actual values	185
Figure 4.22	Velocity Profile in CFD-View after Simulation of a 3D Room	186

Figure 4.23	Velocity Pattern in CFD-View after Simulation of a 3D Room	187
Figure 4.24	Velocity Profile after Simulation of a 3D Room with Solar Chimney Placed at Center of the Room	189
Figure 4.25	Velocity Profile after Simulation of a 3D Room with Solar Chimney Placed at Same Side of Room Inlet	189
Figure 4.26	Velocity Profile after Simulation of a 3D Room with Bigger Inlet Box (1.5×3×3) meter.	190
Figure 4.27	Velocity Profile after Simulation of a 3D Room with Bigger Inlet Box (1.5×3×1.5) meter.	190
Figure 4.28	Velocity Profile after Simulation of a 3D Room with Bigger Inlet Box (1.5×2.25×1.5) meter.	191
Figure 4.29	Velocity Profile after Simulation of a 3D Room with Bigger Inlet Box (3×3×3) meter and Room Inlet at the Center of the Wall.	191
Figure 4.29a	Velocity Profile after Simulation of a 3D Room with Bigger Inlet Box (3×3×3) meter and Room Inlet at Bottom of the Wall.	192
Figure 4.30	Pressure Trace in CFD-View after Simulation of a 3D Room (P=0)	193
Figure 4.31	Pressure vs Z graph just near inlet plane of Room (P=0 at Outlet)	194
Figure 4.32	Pressure vs Z graph just near inlet plane of Room against Sr. No.1 in Table : 4.2 (P=0 at Outlet)	196
Figure 4.33	Pressure vs Z graph just near inlet plane of	196

	Room against Sr. No.2 in Table : 4.2 (P= -0.35 at Outlet)	
Figure 4.34	Pressure vs Z graph just near inlet plane of Room against Sr. No.3 in Table : 4.2 (P= -0.4 at Outlet)	197
Figure 4.35	Pressure vs Z graph just near inlet plane of Room against Sr. No.4 in Table : 4.2 (P= -0.37 at Outlet)	197
Figure 4.36	Pressure vs Z graph just near inlet plane of Room against Sr. No.5 in Table : 4.2 (P= -0.32 at Outlet)	198
Figure 4.37	Pressure vs Z graph just near inlet plane of Room against Sr. No.6 in Table : 4.2 (P= -0.25 at Outlet)	198
Figure 4.38	Pressure vs Z graph just near inlet plane of Room against Sr. No.7 in Table : 4.2 (P= -0.2 at Outlet)	199
Figure 4.39	Pressure vs Z graph just near inlet plane of Room against Sr. No.8 in Table : 4.2 (P= -0.15 at Outlet)	199
Figure 4.40	Pressure vs Z graph just near inlet plane of Room against Sr. No.9 in Table : 4.2 (P= -0.1 at Outlet)	200
Figure 4.41	Pressure vs Z graph just near inlet plane of Room against Sr. No.10 in Table : 4.2 (P= -0.22 at Outlet)	200
Figure 4.42	Pressure vs Z graph just near inlet plane of Room against Sr. No.11 in Table : 4.2 (P= -	201

	0.23 at Outlet)	
Figure 4.43	Pressure vs Z graph just near inlet plane of Room against Sr. No.12 in Table : 4.2 (P= -0.21 at Outlet)	201
Figure 4.44	Pressure vs Z graph just near inlet plane of Room against Sr. No.10 in Table : 4.2 (P= -0.22 at Outlet)	202
Figure 4.46	Pressure Profile in a Room after Simulation (P= -0.22 at Outlet)	202
Figure 4.47	Velocity Profile in a Room after Simulation (P= -0.22 at Outlet)	203
Figure 4.48	Graph showing Convergence of the one Model up to desired level (1.0×10^{-04}).	205
Figure 4.48a	Model with inlet opening at bottom of the side wall – Showing location of data measurement points for analysis	208
Figure 4.48b	Graph showing change in flow through inlet interface with increase in number of cells.	210
Figure 4.48c	Graph showing change in maximum temperature with increase in number of cells.	210
Figure 4.49	Screen Shot of a 3D Room – 4 in CFD-View after Simulation	211
Figure 4.50	Screen Shot of a 3D Room – 7 in CFD-View after Simulation	212
Figure 4.51	3D Room Model in CFD GEOM with Inlet at Bottom of the Wall.	216
Figure 4.52	3D Room Model after Structured Meshing in	217

	CFD GEOM with Inlet at Bottom of the Wall.	
Figure 4.53	3D Room Model with Inlet at Bottom of the Wall in CFD View after Simulation showing Isocurve with Velocity Value of 0.1 m/s.	217
Figure 4.54	3D Room Model with Inlet at Bottom of the Wall in CFD View after Simulation showing Velocity Profile.	218
Figure 4.55	3D Room Model with Inlet at Bottom of the Wall in CFD View after Simulation showing Velocity Vectors.	219
Figure 4.56	3D Room Model in CFD GEOM with Inlet at Top Side of the Wall.	220
Figure 4.57	3D Room Model after Structured Meshing in CFD GEOM with Inlet at Top Side of the Wall.	221
Figure 4.58	3D Room Model with Inlet at Top Side of the Wall in CFD View after Simulation showing Velocity Profile.	221
Figure 4.59	3D Room Model with Inlet at Top Side of the Wall in CFD View after Simulation showing Velocity Vectors.	222
Figure 4.60	3D Room Model with Inlet at Top Side of the Wall in CFD View after Simulation showing Isocurve with Velocity Value of 0.1 m/s.	222
Figure 4.61	3D Room Model in CFD GEOM with Inlet Opposite to Chimney Side.	224
Figure 4.62	3D Room Model after Structured Meshing in CFD GEOM with Inlet Opposite to Chimney Side.	224

Figure 4.63	3D Room Model with Inlet Opposite to Chimney Side in CFD View after Simulation showing Isocurve with Velocity Value of 0.1 m/s.	225
Figure 4.64	3D Room Model with Inlet Opposite to Chimney Side in CFD View after Simulation showing Velocity Profile.	225
Figure 4.65	3D Room Model with Inlet Opposite to Chimney Side in CFD View after Simulation showing Velocity Vectors.	226
Figure 4.66	3D Room with Inlet at Bottom of the Wall, Curve showing Velocity Magnitude Pattern along the Center of the Room.	231
Figure 4.67	3D Room with Inlet at Top Side of the Wall, Curve showing Velocity Magnitude Pattern along the Center of the Room.	232
Figure 4.68	3D Room with Inlet at Opposite Side of the Chimney, Curve showing Velocity Magnitude Pattern along the Center of the Room.	233
Figure 4.69	3D Room Model with Inlet at Top Side of the Wall in CFD View after Simulation showing Velocity Trace.	234
Figure 4.70	3D Room Model with Inlet at Top Side of the Wall in CFD View after Simulation showing Isocurve with Velocity Value of 0.1 m/s.	234
Figure 4.71	Graph showing Convergence of the Model (P=0) up to desired level (1.0×10^{-04}).	237

Figure 4.72	Screen Shot of a 3D Room Model in CFD ACE+ with P=0 N/m^2 at exit.	237
Figure 4.73	3D Room Model (P=0 N/m^2 at exit) with Inlet at Bottom of the Wall in CFD View after Simulation showing Isocurve with Velocity Value of 0.1 m/s.	238
Figure 4.74	3D Room Model (P=0 N/m^2 at exit) with Inlet at Bottom of the Wall in CFD View after Simulation showing trace of air parcel.	238
Figure 4.75	3D Room Model (P=0 N/m^2 at exit) with Inlet at Bottom of the Wall in CFD View after Simulation showing Velocity Vectors.	239
Figure 4.76	3D Room Model (P=0 N/m^2 at exit) with Inlet at Bottom of the Wall in CFD View after Simulation showing Isocurve of Temperature on heated wall of Chimney	239
Figure 4.77	Screen Shots of a 3D Room Model in CFD GEOM with Environment Box at outlet.	243
Figure 4.78	Screen Shot of a 3D Room Model in CFD Solver (ACE+) showing boundary conditions for inlet environment box.	244
Figure 4.79	Screen Shot of a 3D Room Model in CFD Solver (ACE+) showing boundary conditions for outlet environment box.	245
Figure 4.80	Graph showing Convergence of the Model (with environment box at outlet) up to desired level (1.0×10^{-04}).	245
Figure 4.81	3D Room Model (with environment box at outlet & P=0 N/m^2 at exit) in CFD View after Simulation showing trace of air parcel.	246
Figure 4.82	3D Room Model (with environment box at	247

	outlet & P=0 N/m^2 at exit) in CFD View after Simulation showing Isocurve with Velocity Value of 0.1 m/s.	
Figure 4.82a	3D Room Model (with environment box at outlet & P=0 N/m^2 at exit) in CFD View after Simulation showing Isocurve with Velocity Value of 0.05 m/s.	247
Figure 4.83	3D Room Model (with environment box at outlet & P=0 N/m^2 at exit) in CFD View after Simulation showing Velocity Vectors.	248
Figure 4.84	3D Room Model (with environment box at outlet & P=0 N/m^2 at exit) in CFD View after Simulation showing Temperature profile.	248
Figure 4.85	3D Room Model (with environment box at outlet & P=0 N/m^2 at exit) in CFD View after Simulation showing Pressure Profile.	249
Figure 4.86	Comparison of Velocity Magnitude Isocurve with velocity value of 0.05 m/s: Model with box at outlet & P=0 N/m^2 and Model without box at outlet & P= -0.22 N/m^2	251
Figure 4.87	Comparison of Velocity Magnitude Air Parcel Trace: Model with box at outlet & P=0 N/m^2 and Model without box at outlet & P= -0.22 N/m^2	251
Figure 4.88	Graph showing change in ACH value with change in Room Air Inlet Location against above data table	254
Figure 4.89	Graph showing change in ACH value with change in value of pressure at solar chimney exit against above data table.	256
Figure 5.1	Graph showing Convergence of the one Model	261

	up to desired level (1.0×10^{-04}). Above (Normalized values) and Below (Actual values)	
Figure 5.2	3D Room created in CFD-GEOM with Inlet Box	263
Figure 5.3	3D Room created in CFD-GEOM with Inlet and outlet environment Boxes	265

List of Tables

Table No.	Description	Page
Table: 3.1	Physical Parameters Considered during Numerical Simulation	109
Table: 3.2	Simulation Data of 2D Model-7 at Three Different Locations	114
Table: 3.3	Table for Comparison of Simulation Data of 2D Models. Values Taken at Three Different Locations	130
Table: 3.4	Simulation Data of 2D Model-7 at Three Different Locations	134
Table: 3.5	Simulation Data of 2D Model-9 at Three Different Locations	135
Table: 3.6	Comparison of Velocity Magnitudes and Temperatures at two selected points for two different cases	135
Table: 3.7	Estimated fractional error by using Generalization of Richardson Extrapolation for velocity magnitude and temperature at two different selected points	137
Table: 3.8	Simulation Data of 2D Model-7 with Room inlet opening at Bottom of the side wall	140
Table: 3.9	Simulation Data of 2D Model-7 with Room inlet opening at Middle of the side wall	145
Table: 3.10	Simulation Data of 2D Model-7 with Room inlet	149

	opening at Top of the side wall	
Table: 4.0	Physical Parameters Taken for Simulation of 3D Room Fitted with Inclined Solar Chimney	175
Table: 4.1	Simulation Data of 3D Model-1 at Three Different Locations	186
Table: 4.2	Mass Balance Summary of 3D Model-1	188
Table:4.3	Simulation Data of 3D Model-1 with Different Pressure Values at Outlet	195
Table: 4.4	Table for Comparison of Simulation Data of 3D Models	206
Table: 4.5	Simulation Data of 3D Model-4 at Three Different Locations	211
Table: 4.6	Simulation Data of 3D Model-7 at Three Different Locations	212
Table: 4.7	Comparison of Velocity Magnitudes and Pressure at two selected points for two different cases	213
Table: 4.8	Estimated fractional error by using Generalization of Richardson Extrapolation for velocity magnitude and pressure at two different selected points	214
Table: 4.9	Simulation Data of 3D Model-4 at Three Different Locations having Inlet at Bottom of the Wall	216
Table: 4.10	Simulation Data of 3D Model-4 at Three Different Locations having Inlet at Top Side of the Wall	220
Table: 4.11	Simulation Data of 3D Model-4 at Three Different Locations having Inlet Opposite to Chimney	223

Table: 4.12	Comparison of Important Parameters obtained after Simulation of Different Models	229
Table: 4.13	Simulation Data of 3D Model (P=0 N/m^2 at exit) at Three Different Locations having Inlet at Bottom of the Wall	236
Table: 4.14	Comparison of Results obtained after Simulation of 3D Models with P=0 N/m^2 & P= -0.22 N/m^2 at solar chimney exit, with all other parameters remained same	241
Table: 4.15	Simulation Results of 3D Model (with environment box at outlet & P=0 N/m^2 at exit) at Four Different Locations	246
Table: 4.16	Comparison of Results obtained after Simulation of 3D Models: Model without box at outlet & P=0 N/m^2 and Model with box at outlet & P= -0.22 N/m^2 at solar chimney exit, with all other parameters remained same	250
Table: 4.17	Data showing calculation of ACH with different room air inlet locations.	254
Table: 4.18	Data showing calculation of ACH with different value of pressure at solar chimney exit.	256

Abstract

A Solar Chimney is a structure for ventilation purpose installed on the roof or side of any building to induce the stagnant air present in the room / building naturally to the outside using solar heat and attracts the fresh air from outside into the building. Many researchers and scientists have carried out studies based on computation and experiments for different type of solar chimney systems. Despite the fact that extensive work has already been carried out in this field but still huge potential exists to make this system more effective and economical.

The focus of the study and research is on solar chimney based ventilation systems. The use of Computational Fluid Dynamics (CFD) in solar chimney based system has been investigated and CFD models have been developed for research. In this study, naturally induced flow through a Two and Three Dimensional square rooms having dimensions (3 Meter × 3 Meter) and (3 Meter × 3 Meter × 3 Meter) respectively have been developed and investigated through numerical analysis. The numerical analysis has been carried out using Computational Fluid Dynamics (CFD) software. An inclined solar chimney has been used on the roof of the above mentioned square room to induce air flow through the room.

Based on the previous research in this field, the chimney height has been taken as 1.5 meter (effective length 2.12 meter), width of the chimney (0.3 meter for 2D room and 0.4 meter for 3D room). The chimney inclination angle is 45 degree with roof top and solar heat flux rate for chimney heated wall has been fixed as 300 W/m^2. The room air inlet is square shaped having dimensions 0.4 meter × 0.4 meter. The purpose is to investigate the effect of change of room air inlet position (Bottom, Center,

Top of the room side wall) to ultimately design / suggest most effective scenario.

The graphical and numerical analysis shows that by changing the room air inlet location, the air flow rate and flow pattern significantly changes. The best pattern is the one where air is evenly distributing in the room and passes through the sitting area of the occupants i.e. around floor of the room. Keeping this important factor in view, the desired air flow pattern, in both 2D and 3D simulations, has been seen in a model with air inlet location at top of the side wall.

Nomenclature / List of Symbols Used

The nomenclatures of terminologies and symbols used in the study are given below:

T	**Temperature**
P_t	**Total Pressure**
K	**Thermal Conductivity**
k	**Turbulent Kinetic Energy**
g	**Gravity**
T_{ref}	**Reference Temperature**
ρ	**Density**
μ	**Dynamic Viscosity**
v	**Kinematic Viscosity**
ε	**Dissipation Rate**
U	**Velocity in Horizontal Direction**
V	**Velocity in Vertical Direction**
W	**Velocity in Z-direction**
ΔP	**Pressure Difference**
VM	**Velocity Magnitude**
P	**Static Pressure**
Cp	**Specific Heat**
Q	**Heat Flux**
RH	**Wall roughness Height**

H_s	**Stack Height**
S_{ij}	**Mean strain-rate tensor**
τ_{ij}	**Reynolds stress tensor**
Re	**Reynolds Number**
Ra	**Rayleigh Number**
Pr	**Prandtl's Number**
C_D	**Closure Coefficient**
f	**External forces acting on a body**
β	**Volumetric Expansion Coefficient**
ℓ	**length**
P_s	**Stack Pressure**

All length values used / given in the study are in meters, Velocity in Meter/Sec and Pressure values in N/m^2, if otherwise specified.