ASEE's 123rd Annual • Conference & Exposition • New Orleans, LA • June 26-29, 2016

New Orleans JAZZED about Engineering Education

Paper ID #15281

Space Air Diffusion Laboratory

Dr. John E Matsson, Oral Roberts University

John Matsson is a Professor of Mechanical Engineering at Oral Roberts University in Tulsa, OK. He earned M.S. and Ph.D. degrees from the Royal Institute of Technology in Stockholm, Sweden in 1988 and 1994 respectively.

Mr. Matthew Raman Fulton, Oral Roberts University

Matthew Fulton is a senior Mechanical Engineering Student at Oral Roberts University

Mr. Brennan T. Harrup, Oral Roberts University

Brennan Harrup is a senior Mechanical Engineering Student at Oral Roberts University with a Biochemistry minor.

Ms. Meagan N Case, Oral Roberts University

A senior student in Mechanical Engineering at Oral Roberts University, about to graduate. Interested in manufacturing/design.

Space Air Diffusion Laboratory

Abstract

Space air diffusion systems are an integral part of many HVAC systems[1-2]. The design of the system includes finding the best location for introduction of air into the room. This paper expounds on the designing, building, and testing of a space air diffusion laboratory setup for undergraduate engineering students. The laboratory will enable students to conduct hands-on experiments that involve visualization and measurements of laminar, transitional, and turbulent ceiling wall air-jets and the resulting room air motion.

Students designed the space air diffusion experimental test setup for use in the Fluid Mechanics course’s laboratory and for the ASHRAE Senior Undergraduate Project Grant program, which funded the project. Three groups of students in the Manufacturing Processes course designed the experimental setup – one group designed the mechanism to position the Pitot-tube and hot-wire anemometry measurement devices in the flow field, another group designed the layout of the channel the air would pass through to become steady flow, and the remaining group designed the section which would streamline the air supplied by the fan. The overall objective was to engage the students in a design project. This paper will provide details of the evaluations and outcomes of the project.

For this project, the students designed the apparatus using SolidWorks® CAD software, and ANSYS Fluent software was used for CFD simulations of the flow field. The components used in the design were: a blower, a settling chamber, a perforated plate, honeycomb, mesh screens, and a 3D printed contraction in the shape of a fifth order polynomial to minimize the turbulence level of the flow entering the 2,440 mm long plane channel made with a cross section of 10 mm high and 232 mm wide. The purpose of the channel was to get a fully developed flow entering the model room for calibration of the hot-wire anemometer. The dimensions of the model room designed by the students are 232 mm in height, 232 mm in width, and 464 mm in length.

The ceiling wall jet was discharged parallel and adjacent to the straight horizontal ceiling of the model room. The jet developed along the ceiling surface of the room and entrained air from the room as its velocity deceased when it moved into the room. The maximum velocity remained close to the ceiling surface with the distance that the ceiling jet adhered to the surface depending on the relative influence of inertia and gravity.

Introduction

This project involved the design, building, and testing of a space air diffusion laboratory by undergraduate engineering students. The laboratory will enable students to conduct hands-on experiments including visualization and measurement of laminar, transitional and turbulent isothermal and non-isothermal wall air-jet and room air motion. Space air diffusion is the distribution of conditioned air by an outlet that is discharging the air into the room to provide a healthy environment.

The wall jet has four distinct zones. The first zone is the core zone in which the maximum velocity of the jet is not changing with streamwise location. The second zone is called the transition zone where the flow changes from laminar to turbulent flow. The length of this zone depends on parameters such as the geometry of the outlet section and the turbulence level of the incoming flow. In the third zone, the flow is turbulent and it is this zone that has the largest streamwise extent. Finally, in the fourth zone the velocity of the air decreases very rapidly to almost zero level.

In an isothermal wall or ceiling jet, the centerline velocity in the second and third zones can be shown to decay as

$$\frac{U_{x,2}}{U_0} = \sqrt{\frac{kh}{x}}, \quad \frac{U_{x,3}}{U_0} = k\frac{\sqrt{hw}}{x} \tag{1}$$

Where *x* is the streamwise coordinate, $U_{x,2}, U_{x,3}$ are the centerline velocities in zones 2 and 3 respectively, U_0 is the initial velocity of the wall or ceiling air jet, *h* and *w* are the height and width of the initial incoming air jet cross section, and *k* is a dimensionless constant of proportionality.

The laminar wall jet was first studied by Glauert[3], who found a self-similar velocity profile solution to the momentum equation using the boundary layer approximation. For the plane wall jet the streamwise velocity profile $u = 5U_0 f'(\eta)(\bar{x})^{-\frac{1}{2}}/4$ where $f'(\eta)$ is determined by the following ordinary differential equation:

$$f''(\eta) + f(\eta)f'(\eta) + \frac{4f'(\eta)^2}{10} = 0 \tag{2}$$

with boundary conditions $f(0) = f'(0) = f'(\infty) = 0$ where the similarity variable $\eta = \frac{5\bar{y}}{4(\bar{x})^{3/4}}$ and $\bar{x} = \frac{xU_0}{\nu}, \bar{y} = \frac{yU_0}{\nu}$ are the non-dimensional streamwise and wall normal coordinates based on dimensional coordinates *x*, *y* and the kinematic viscosity ν. Many studies have since then been conducted on wall jets.

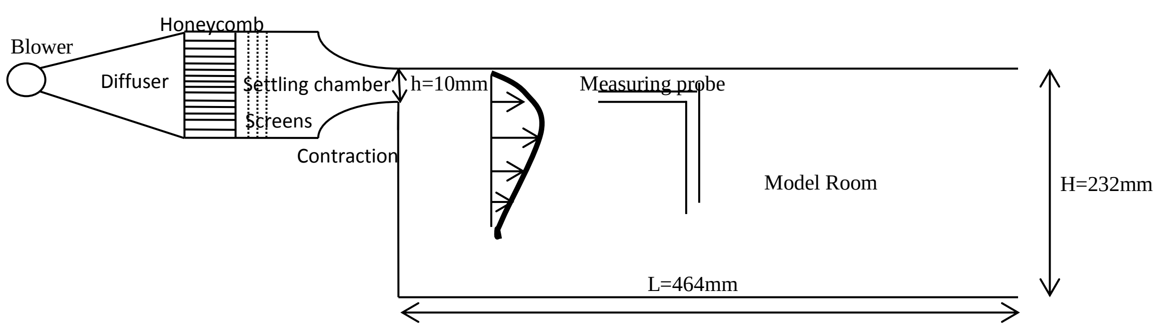


Figure 1 Schematic of experimental setup for space air diffusion laboratory (note: plane channel section not shown)

In this laboratory setup, the ceiling jet will be discharged parallel and adjacent to the straight ceiling as seen in Figure 1. The maximum velocity will remain close to the ceiling surface as shown by Tuve[4]. The distance that a non-isothermal ceiling jet will adhere to the surface depends on the relative influence of inertia and buoyancy as described by the Archimedes number

$$Ar = \frac{gD(T_0 - T_S)}{U_0^2 T_S} \quad (3)$$

where $D = 2hw/(h + w)$ is the hydraulic diameter for the rectangular cross section at the outlet, g is acceleration due to gravity, T_0 is the initial temperature of the ceiling air jet, and T_s is the surrounding air temperature. Wilson et al.[5] derived the velocity and temperature profiles for the non-isothermal wall jet. The trajectory described by non-isothermal wall jet has been studied by Etheridge and Sandberg[6] who derived an expression for the trajectory and Grimitlyn and Pozin[7] who proposed a different model for the trajectory.

The students designed the space air diffusion flow apparatus with diffuser, honeycomb, settling chamber, screens, and a contraction in order to minimize the disturbance level of the flow. A ceiling jet will develop along the ceiling surface of the room model. Measurements of velocity profiles will be completed using a Pitot tube and hot-wire anemometry. The ceiling jet will entrain air from the room and its velocity will decrease as it moves into the room. The throw is defined as the distance from the inlet where the velocity has decreased to a certain percentage of the inlet velocity. The drop is the vertical distance that the ceiling jet falls from outlet to the throw.

Experimental Set-Up

The experimental setup consisted of a settling chamber with a honeycomb to even out the air pressure, mesh screens to help settle the flow, and a contraction followed by a long, straight rectangular section to fully develop the flow before it enters the model room at the end.

The contraction was designed with a curve based on the following 5th dimensional polynomial function in accordance with the paper by Bell and Mehta[8]. In their paper they determined that the contraction wall shape satisfying most of the requirements (low Reynolds Number and flow uniformity) is the one given by a 5th order polynomial.

$$\mathrm{y(x)} = H_i - (H_i - H_o)\left(6\left(\frac{\mathrm{x}}{L}\right)^5 - 15\left(\frac{\mathrm{x}}{L}\right)^4 + 10\left(\frac{\mathrm{x}}{L}\right)^3\right) \quad (4)$$

In which case L is the length of the contraction, H_i is half the inlet height of the contraction, and H_o is half the outlet height of the contraction. Through evaluation of the dimensions of the resultant flow area the following equation was obtained.

$$\mathrm{y(x)} = 45.75 - (45.75 - 5)\left(6\left(\frac{\mathrm{x}}{109.8}\right)^5 - 15\left(\frac{\mathrm{x}}{109.8}\right)^4 + 10\left(\frac{\mathrm{x}}{109.8}\right)^3\right) \quad (5)$$

Parameters for the equation included an initial *x* value of 0 and a final *x* value of 109.8 and an area ratio of 9. This 5th order polynomial was designed to avoid separation of the flow in the contraction and to ensure that the airflow through it didn't introduce separated flow.

The contraction was modeled in SolidWorks® (Fig. 2) and 3D printed from an STL file by the university with the assistance of the lab technician to retain dimensional accuracy of the necessary curve (Fig. 3). Below is the SolidWorks® model compared with the actual printed part. The inside surfaces were sanded and painted to add a smooth finish.

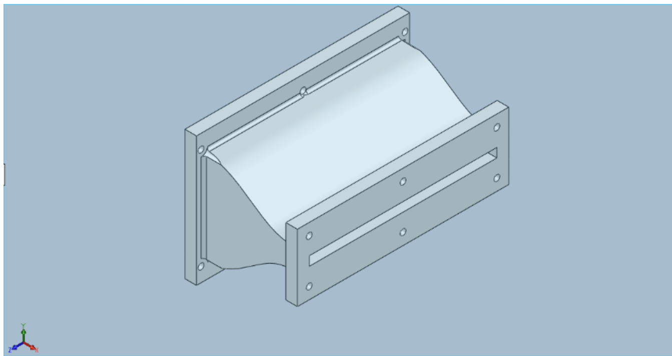


Figure 2 SolidWorks® model of the contraction

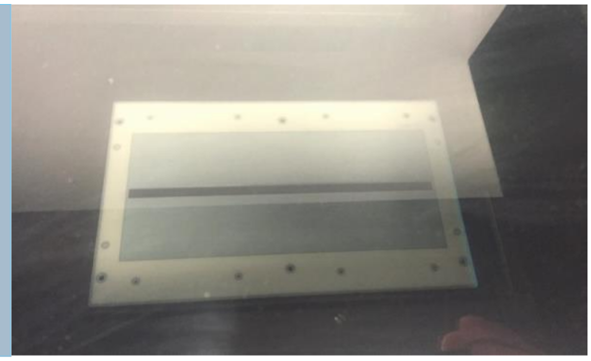


Figure 3 3D printed contraction

The settling chamber is intended to produce a spatially uniform fluid flow and reduce turbulence. The objective is to obtain laminar flow by eliminating as much transverse flow as possible. The chamber was constructed out of four sections of Plexiglas plate: two 38.1 mm thick sections and two 50.8 mm thick sections mounted together by six 3.175 mm diameter all-thread steel rods. The 50.8 mm thick sections were attached directly to the blower, and housed a perforated plate and the honeycomb respectively. The perforated plate was intended to induce an initial pressure drop, and the clear polycarbonate section of honeycomb was introduced to reduce transverse flow. These were followed by a rough and fine section of stainless steel mesh fixed to the back of the two remaining 38.1 mm thick sections of Plexiglas. The last 38.1 mm section was attached to the contraction via threaded holes in its printed back-plate as seen in Figure 4 below.

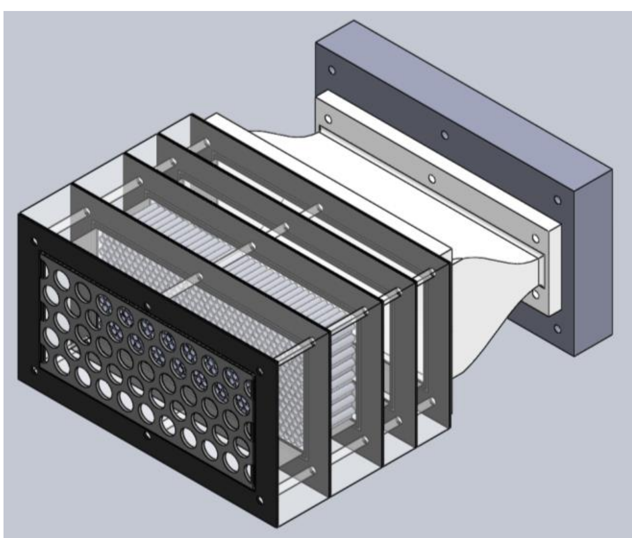


Figure 4 Final SolidWorks® model of the settling chamber and contraction

The contraction was then bolted to a 2440 mm long plane channel with an inside cross-section 10 mm high and 232 mm wide designed to allow the flow to fully develop. This flow then exited into the prepared model room 232 mm high, 232 mm wide, and 464 mm long. The far end of the room was left open for access, and to allow the flow to exit the system. A traversing system for mounting a Pitot tube was installed outside of the box to measure the velocity profile of the resulting flow inside the box as seen later in Figure 9. Figure 5 shows the finished arrangement of the settling chamber.

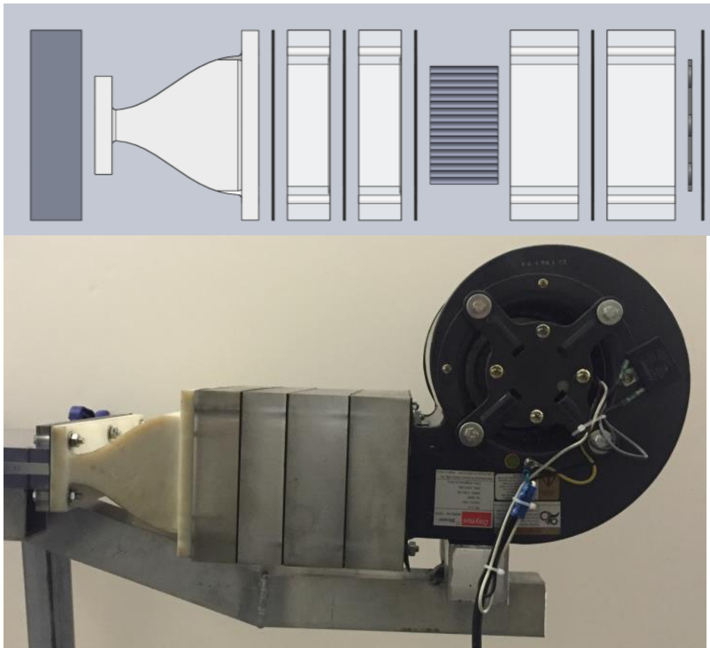


Figure 5 Settling chamber in SolidWorks® exploded assembly compared to constructed experimental arrangement

Dividing the sections within the settling chamber from one another, a rubber gasket (seen in Figure 6) was introduced to reduce the amount of vibration translated by the attached blower and to keep the assembly airtight. The blower purchased for this application was the Dayton 1TDR9 Blower attached at the inlet of the settling chamber. The blower was capable of producing a maximum flowrate of 463 cfm on 115 V and 1.28 A at 1600 rpm. A circuit had to be introduced to allow the flow to be modulated to a desired value.

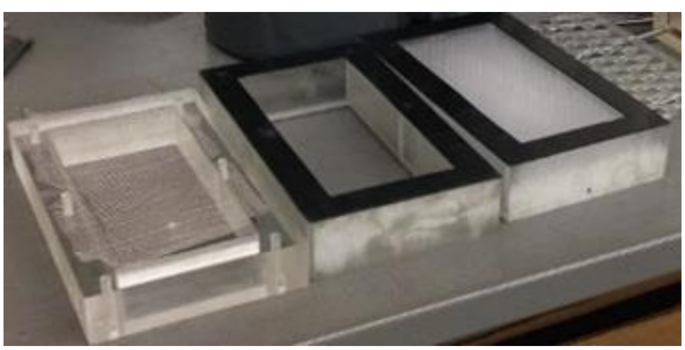


Figure 6 Components of the disassembled settling chamber being fitted with gaskets

In order to move the probe within the model room, two separate traversing systems were made available. One of these systems can be manually operated, while the other is computer-controlled. Both systems are mounted for two degrees of freedom movement: streamwise and vertical. The probes for each of these systems enter through the floor of the model room via a slit which is cut streamwise along the center of the base. Opposing brushes are used to seal the slit.

The manual system utilizes a Shars digital height gauge (shown in Figure 7) to measure the vertical displacement to 1/100 mm. This height gauge is supported on a rail (shown in Figure 8) that provides displacement in the streamwise direction. A Pitot tube is mounted on the height gage (shown in Figure 9) and directed into the airstream to take measurements.

Figure 7 Manual traversing system

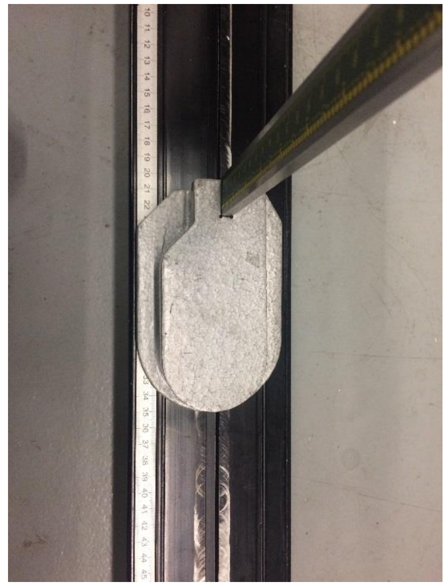


Figure 8 Height gage mounted on rail

The computer-controlled system gives the option of faster and automatic movement of the measuring probe. Two Airpax Series L92400 digital linear actuators are used to create movement in the wall normal and streamwise directions. The actuators are controlled using National Instruments' Labview© software. Each actuator may be controlled separately using the software and hardware. A custom-built control box (shown in Figure 10) with a stepper motor driver board links the actuators to a Labview© digital I/O board. The flow is measured using a Pitot tube and a Dantec Dynamics miniature boundary layer hot wire anemometer probe. The anemometer feeds data back into the Labview© program which writes the data into a spreadsheet.

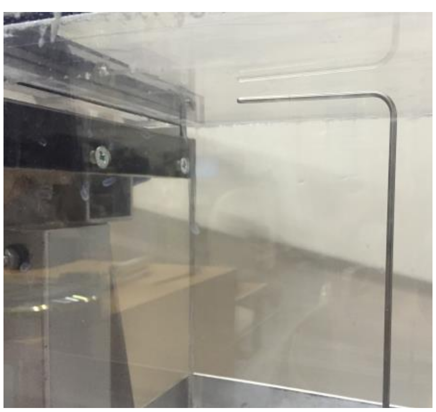


Figure 9 The Pitot tube in the model room

Figure 10 Stepper motor drive board box

ANSYS Fluent Flow Simulations

Simulations of the Space Air Diffusion laboratory were performed using ANSYS Fluent software. These simulations also covered different flow rates through the model room. The contours function in the graphics and animations windows were used to provide x-velocity visualization while the solver was running, providing a video of the simulation.
The general settings chosen in ANSYS Fluent were for both steady state and transient study in the 2D plane, and the viscous laminar model was also used. Gravitational acceleration was set in the general settings with a y-component of -9.81 m/s^2. The rectangular computational domain was the same as the projected area in the experiments with dimensions of 464 mm in width and 232 mm in height, which was used in all of the trials. The inlet velocities along the ceiling wall were set to be 1, 2, 4, and 8 m/s, and the computational domain was meshed using a biased mesh in the vertical direction. The simulations were run with a time step of 0.01 s during 500 time steps for a total of 5 s.

The x-velocity plot in Figure 11 shows the distribution of the air velocity at different distances from the inlet using a User Defined Function (UDF) with a maximum velocity value of 1 m/s. As can be seen from the diagram, the velocity has a parabolic shape that tends to flatten out as distance from the inlet increases.

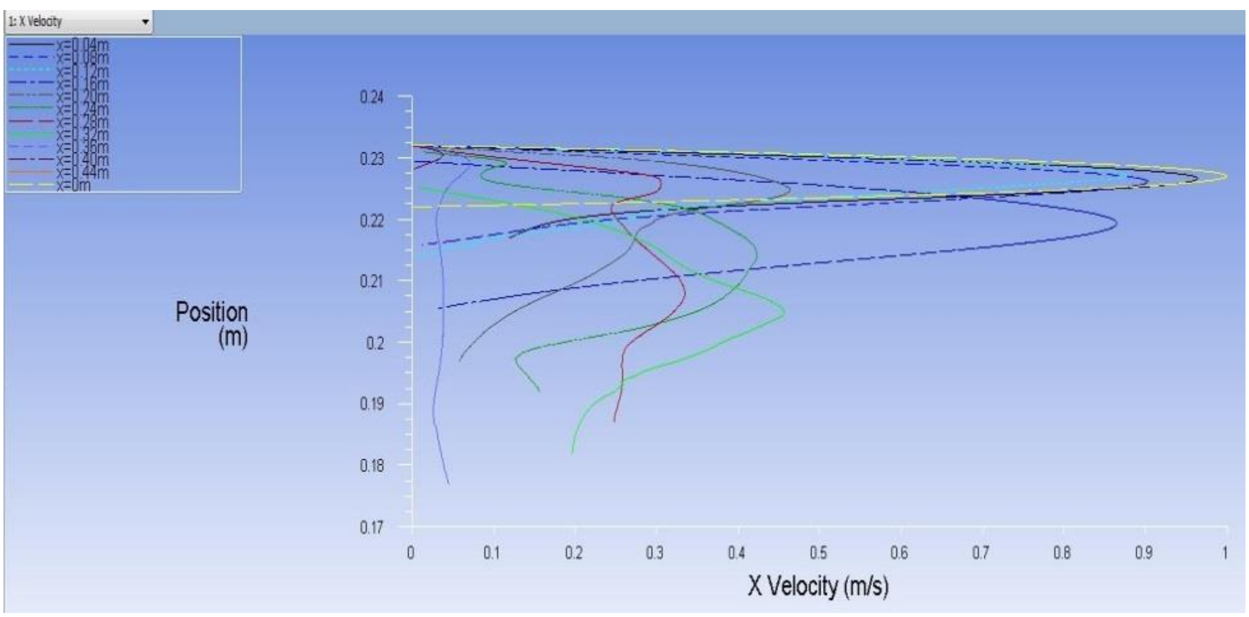
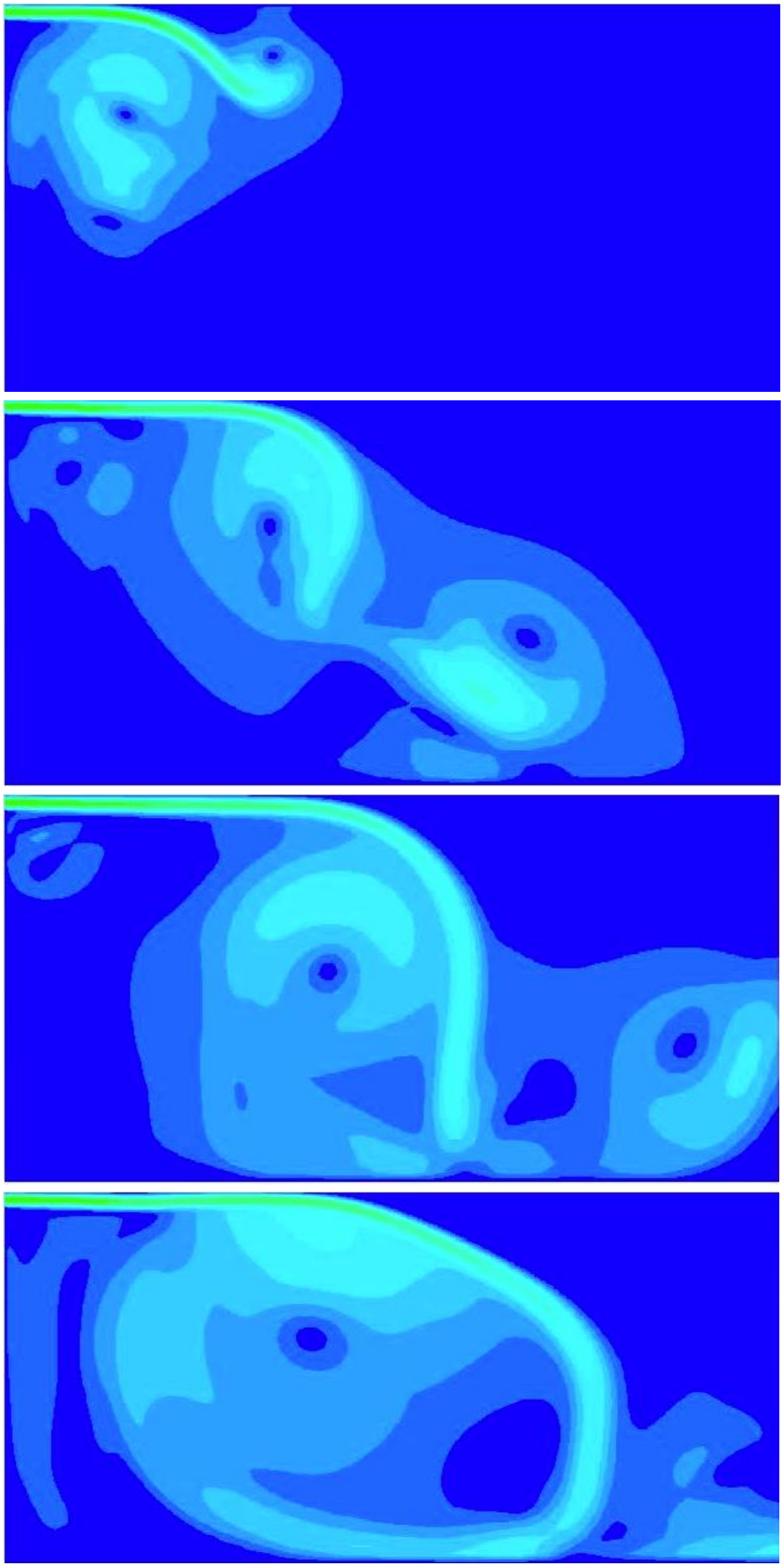


Figure 11 x-velocity profiles at different streamwise positions, *Re* = 427

The figures below (Figures 12a-d) show how contours of x-velocity develop over time. As can be seen from the Figures the air stream enters through the inlet and flows parallel to the ceiling for a short time. However, as the distance from the inlet increase, the air stream starts falling towards the floor of the model room due to the gravitational force. Finally, the air stream reaches the floor of the room and splits with a greater portion of it returning towards the inlet and thereby causing a recirculation zone.



a)

b)

c)

d)

Figure 12 a), b), c), and d) x-velocity distribution over 5 seconds, *Re* = 427

Experimental Results

Experimental results were obtained by the students using Pitot tube measurements to evaluate the flow. They determined the velocity profile at a number of streamwise locations to help visualize the streamwise development of the flow. LabVIEW™ software was used for data acquisition in order to determine the time averaged streamwise velocity at each data point. The flow results determined were then compared to the ANSYS Fluent simulation results.

First, a velocity profile was measured at the end of the straight channel section to establish that the flow was fully developed with a parabolic profile. The plot shown in Figure 13 was obtained.

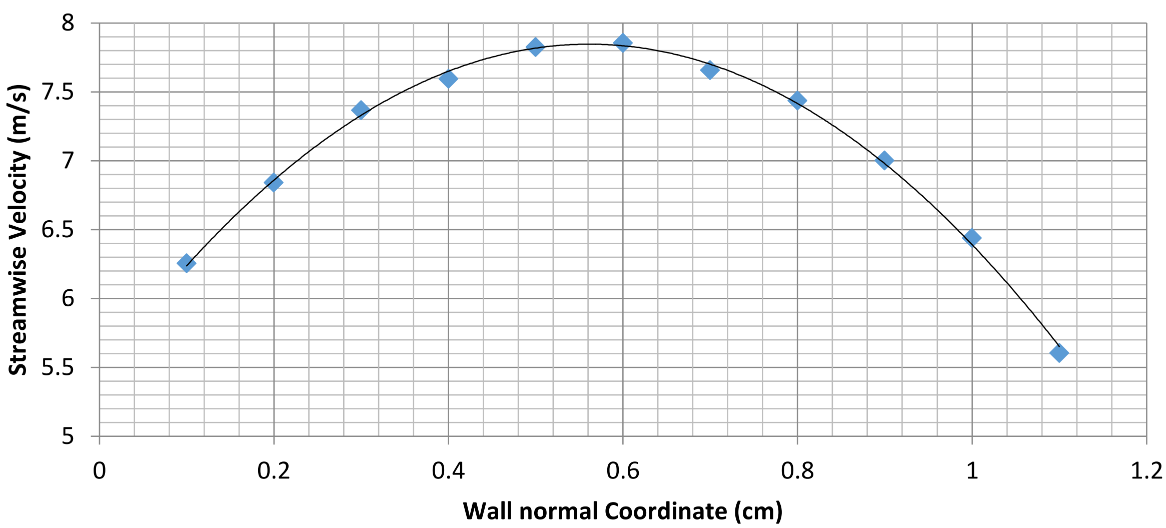


Figure 13 Velocity profile measured at the exit of the straight channel, *Re* = 3418

The parabolic curve fit corresponded to a R^2 value of 0.9975, establishing that the flow was indeed fully developed. The corresponding Reynold's number was obtained using the standard equation:

$$Re = \frac{\rho U_0 L_c}{\mu} \quad (6)$$

Where ρ is the density of the air, L_c corresponds to the channel height for the straight channel section, and μ refers to the dynamic viscosity of the air.

Further results were obtained detailing the profiles within the model room. Measurements were taken at distances of 2, 4, 6, 8, 13, 18, and 21 centimeters from the inlet of the room. The points were taken at increments of 1mm in the wall normal direction at a minimum of eleven points per profile. The sampling time was varied from 5 to 15 seconds as needed to obtain a smooth profile.

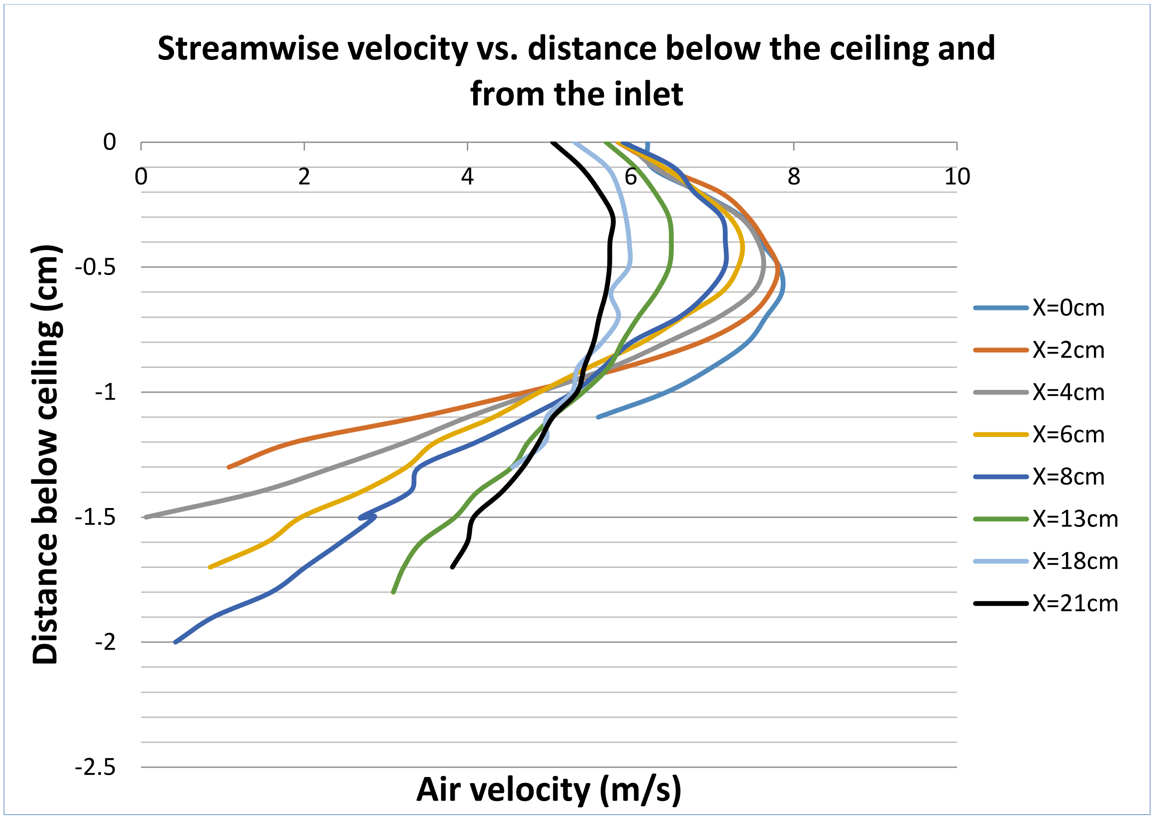


Figure 14 Streamwise velocity profiles at different distances from the inlet

The distances below the ceiling (the vertical axis) on the chart for each stream profile represent the distance of the top of the Pitot tube from the ceiling. This means that the true distance is offset downwards by the thickness of the Pitot tube wall, which accounts for the fact that the no-slip condition (zero velocity at any wall) is not demonstrated by the chart above (Figure 14).

These velocity profiles do demonstrate, on the other hand, the decreasing overall velocity based on distance and the flattening out of the profile. While the profile starts in a parabolic shape (a fully developed flow at the inlet to the room), as the distance increases the velocity profile (at x = 21 cm) approaches the shape shown in Figure 1 in the Introduction to this paper. Overall, this experiment demonstrates not only that the settling chamber, contraction, and channel do make a fully developed, laminar flow into the chamber, but also that the flow (as it gets further away from the inlet) does approach the estimated profile at the beginning of the experiment (Figure 1).

Student Involvement

Students were responsible for or involved in every phase of this project. The primary focus of student effort was on design and material selection. The Fall 2015 Manufacturing Processes class of 18 students was split into three teams which each worked on a section of the experimental setup design. Team one focused on the settling chamber and contraction; team two designed the model room and plane channel section, and team three manufactured two traversing systems: one mechanical, one electrical. This required communication between the three groups in order for designs to blend together.

Each student worked on the course project a minimum of one hour per week outside of scheduled class hours. Weekly progress reports accounted for 33% of the project grade while the completion of the project and final report contributed to the rest of the grade. This apparatus has been included as a course project in the spring semester to obtain measurements of the flow field.

The majority of the design of the contraction was accomplished using SolidWorks® software. Models and drawings were presented to the instructor and lab technician for feedback. Some minor details and adjustments were communicated to the lab technician verbally during the building process. Students ran fluid flow simulations using ANSYS Fluent software. The data from those simulations was used to create a model of a part which was 3D printed using an Objet 24 printer.

The building of the remainder of the system was done in the machine shop in the engineering department under the supervision of the technician. Students performed online research and consulted with the instructor and the technician in order to select materials for purchase. Examples of such supplies and materials include linear actuators for the electrical traversing system, a height gage for the mechanical traversing system, and a 3D printed contraction used to reduce turbulence level of the air before entry into the plane channel section. The majority of materials were in stock in the machine shop, although some specialized parts were acquired and purchased.

Students will continue to use this apparatus in labs demonstrating space air diffusion studies, using hot-wire anemometry and smoke visualizations in addition to Pitot tube measurements to compare with SolidWorks® and ANSYS Fluent flow simulations. These labs will be included as part of Fluid Mechanics, Heat Transfer, and Experimental Methods courses.

Assessment and Outcomes

The purpose of the project was to have the students participate in the design and manufacture of the apparatus. The apparatus was intended to enable the observation of the time dependent laminar and turbulent air flows as they entered the model room, using a mechanically operated pitot tube and a hot wire anemometer for precise and accurate data collection.

The students were drawn from the Manufacturing Processes and Fluid Mechanics courses. The students received credit for the project as 15% of their final grade in the Manufacturing Processes course. The project counted as 10% of final grade in the Fluid Mechanics course. The project served as a means to encourage the problem solving and design creativity to successfully produce the experimental setup as well as experience with manufacturing and fabrication techniques used.

Each team composed weekly reports of the progress in their assigned division. Cooperation with the lab technician was essential for successful manufacture of the design. Also, cooperation between teams ensured the assembly would be dimensioned properly at transition points and be fully functional.

Because the three teams were able to cooperate in designing, manufacturing, and assembling an operational experimental set-up that fit the expectations they showed an ability to "apply knowledge of mathematics, science, and engineering." This is one of the qualities expected of students taking the Manufacturing Processes course. The experimental data obtained in the subsequent Fluid Mechanics course showed the students were able to "design and conduct experiments, as well as analyze and interpret data." The students, through their use of modern equipment to do processes such as NC milling and welding showed "an ability to use the techniques, skills, and modern tools necessary for engineering practice" (ABET, Criterion 3). Thanks to the cooperative quality of the assignment, all students were able to benefit creatively and interactively from designing, manufacturing, modeling, and building of the space air diffusion lab together. The student learning outcome that students prove that they are "able to apply knowledge of mathematics, science, and engineering," was the primary goal of the project, and was achieved.

Student Feedback

Through the design of the ASHRAE Space Air Diffusion Project, students were involved every step of the way. The students learned how to be given a task, develop a prototype, CAD model the system, order the needed parts, assist in the manufacturing process, and communicate with each other in different groups. In the development phase, the students communicated within and between the separate groups to ensure that each section of the project would be mated properly. CAD models were made to display the components to be built and assemblies were made to ensure they would fit with their mating components. Once the design of the system was completed, the students ordered the needed parts and assisted in the construction of the overall apparatus. This simulated how companies or engineering firms would communicate and solve problems in the development of a product or service. With the involvement in the ASHRAE Space Air Diffusion Project, students learned and experienced a variety of tasks that involved several engineering disciplines.

There is room to improve the laboratory with upgrades and additional testing methods. Suggested improvements include changing the blower to allow testing at higher flow rates and their associated Reynolds numbers, and the use of other flow visualization methods such as shadowgraphy and Particle Image Velocimetry (PIV) to compare with simulated models.

Conclusion

This paper has shown a space air diffusion project used by undergraduate students to test different variables concerned with room air motion. It was initiated as a project in the Manufacturing Processes course. Through the course of this project the undergraduate students were able to develop and construct a space air diffusion model that portrays the conditions of air being introduced into a room and the ability to analyze different areas of airflow within the room. The cost of building the experimental setup described through the paper was $5017.32 with $4997 being provided by the ASHRAE Senior Undergraduate Project Grant. The project cost is detailed in Table 1 on the following page.

Table 1 Detailed project cost

Item	**Price**
Centrifugal Blower, edm industries, D4E146-AA25-34	$185
Acrylic Sheet, 0.5 in x 48 in x 96 in, Professional Plastics	$422
3-D Printed Contraction, ORU 3D Print Lab	$0
Spur Gear	$25
Gear Rack	$25
Thompson Linear Guide Rail and Carriage Assembly	$360
Dantec Dynamics MiniCTA Hot Wire Anemometer package	$3680
Aluminum Flat Bar for Long Plane Channel	$22.32
TOTAL:	$5017.32

Bibliography

1. McQuiston F.C., Parker J.D. and Spitler J.D., Heating, Ventilating and Air Conditioning: Analysis and Design, 6th Edition, Wiley, (2004).

2. 2001 ASHRAE Fundamentals Handbook (SI).

3. Glauert, M.B., The wall jet, Journal of Fluid Mechanics 1 (6), 625-630, (1956).

4. Tuve G.L., Air Velocities in Ventilating Jets, Heating, Piping , and Air Conditioning, No. 1, 181-191, (1953).

5. Wilson J.D., Esmay M.L. and Persson S, Wall-jet velocity and temperature profiles resulting from a ventilation inlet, Transactions of the ASAE, 13(1), 77-81, (1970).

6. Ethenridge D, and Sandberg M, Building Ventilation: Theory and Measurement, John Wiley, (1996).

7. Grimitlyn M.I. and Pozin G.M., Fundamentals of optimizing air distribution in ventilated spaces. ASHRAETransactions, 93(1), 1128-1138, (1993)

8. Bell, J. H., and R. D. Mehta. Contraction Design for Small Low-speed Wind Tunnels. Moffett Field, Calif.: National Aeronautics and Space Administration, Ames Research Center, (1988).