

Effect of Cone Angle and Reynold's number on the efficiency of a Diffuser (Reg:190250984)

Introduction

Diffusers are devices that are used to decrease the velocity of a fluid flow and increase its static pressure. As such, they have a wide variety of applications. In ventilation systems, diffusers are used to slow and evenly spread the flow of air into a room. A turbine engine diffuser slows the fast flowing air coming from the compressor into the combustion section to prevent the otherwise fast flowing air from extinguishing the flame. An F1 car diffuser creates down-force by creating a low pressure region of fast flowing air on the underside of the car. This creates a pressure differential with the slower moving air flowing over the car which pushes down on the vehicle.

The diffuser does this by exploiting the mass conservation law. Expressed mathematically as $\rho_1 v_1 A_1 = \rho_2 v_2 A_2$, for constant density, if the area increases from point 1 to 2, then velocity must decrease. Bernoulli's principle says that for two points along a streamline, the total pressure P_1 at point 1 is equal to the total pressure P_2 at point 2, i.e

$$\frac{1}{2}\rho v_1^2 + p_1 = \frac{1}{2}\rho v_2^2 + p_2.$$

Then if $v_1 > v_2$, $p_2 > p_1$, so an increase in static pressure is achieved.

Bernoulli's principle relies on steady flow, incompressible, and inviscid assumptions. However, in reality, viscous and turbulent effects can have a significant influence on the efficiency of a diffuser. A perfect diffuser will decrease the velocity of a flow uniformly and convert all the kinetic energy into potential energy, but in reality skin friction at the wall causes the flow to slow down non-uniformly and gives rise to turbulent conditions. Turbulence steals kinetic energy from the main flow and converts it to thermal energy through friction which decreases the conversion of kinetic energy to pressure energy. This manifests as a total pressure drop across the diffuser which is a sign of diffuser inefficiency. The goal is then to minimize turbulent effects in a diffuser.

This report looks at the effect of changing the diffuser cone angle and the Reynold's number on the pressure recovery coefficient $PRC = (p_2 - p_1)/(\rho V_0^2/2)$. The PRC measures the proportion of dynamic pressure which is converted to static pressure and is the measure of diffuser efficiency. The report will also use, and give a critique of, three analysis methods: theoretical; numerical; and experimental; and will discuss the merits and limitations of each.

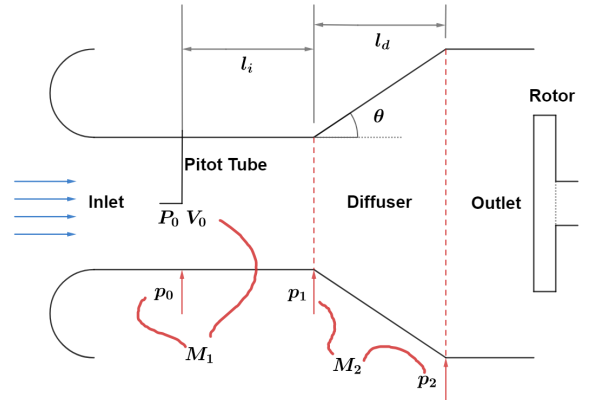


Figure 1: Experiment Setup.

Experimental Procedure

Figure 1 shows a cylindrical wind tunnel consisting of a bell-mouth inlet section with a diameter $D_1 = 90\text{mm}$, a diffuser section with a cone angle 2θ , and an outlet section with a diameter $D_2 = 180\text{mm}$ which houses a speed controlled rotor.

A pitot tube is installed at a distance l_i upstream from the diffuser where the total pressure P_0 is measured at the central axis of the wind tunnel. Several tubes are installed at equal spacings around the circumference of the wind tunnel at positions p_0, p_1 , and p_2 to measure the static pressure at those points.

The manometer M_1 and measures the dynamic pressure at p_0 by calculating $P_0 - p_0$. The manometer M_2 measures the static pressure increase across the diffuser by calculating $p_2 - p_1$.

Three diffusers with half-cone angles 4.1° , 19.8° , and 68.2° , were tested for four unique wind velocities. The manometer readings were tracked for 1 minute (after an initial 2 minutes to allow the flow to fully develop). The maximum and minimum readings over that time were recorded to estimate the range of values.

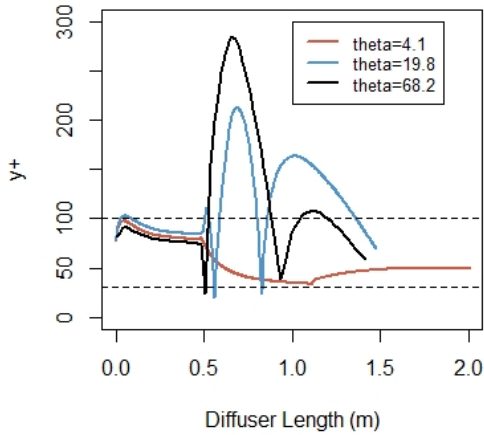


Figure 3: Shows an example of $y+$ for each diffuser geometry. For the short and medium diffuser, $y+$ covers a wide range of values. This causes divergence in the short diffuser residuals and a potentially unreliable solution for the medium diffuser despite achieving convergence.

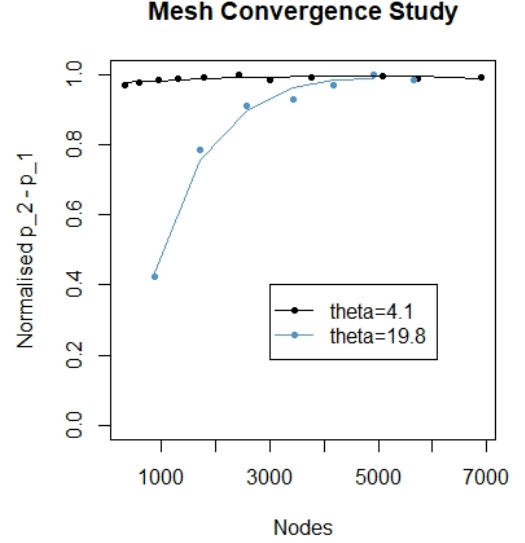


Figure 2: Shows the mesh convergence study for half-cone angles $\theta = 4.1$ and $\theta = 19.8$.

The air temperature was assumed to be 20°C which implies a density $\rho = 1.2$ and kinematic viscosity $\nu = 1.5111E - 05$. The quantities $Re_0 = \frac{V_0 D_1}{\nu}$, and $PRC = \frac{p_2 - p_1}{\rho V_0 / 2}$ were calculated.

Numerical Procedure

The diffuser simulation uses the Reynold's Averaged Navier-Stokes Equations (RANS) to solve for the dimensionless mean velocity U^* . The standard $k - \epsilon$ model was selected to model the Reynold's stresses, which assumes the Reynold's stresses are linearly proportional to the mean strain rates, i.e.

$$\rho u_i u_j = \nu_T \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right).$$

Here, ν_T models the kinetic energy dissipation from the main flow due to turbulent effects by acting as an additional viscosity. The standard wall function was selected to model the velocity profile at the wall of the diffuser.

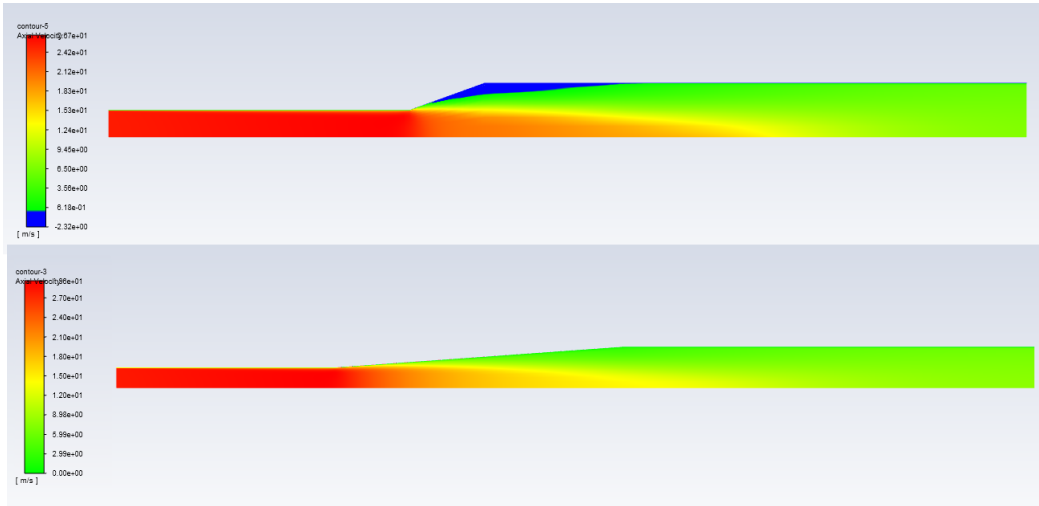


Figure 4: CFD resolved velocity field for half-cone angle $\theta = 19.8^\circ$ and $\theta = 4.1^\circ$, top and bottom respectively. Blue colour signifies areas of negative x -direction velocity, i.e. flow separation.

The diffuser is modelled with a 2-D axi-symmetric geometry. The "double precision" solver, and "second order upwind" option for the momentum, k , and ϵ were chosen to ensure solution accuracy since the computational time required for this simulation is low. The turbulence intensity I for each simulation was calculated using $I = 0.16(Re_{D_h})^{-18/8}$, where D_h is the hydraulic diameter taken as $D_h = 90\text{mm}$. The air temperature was taken as 20°C . Point surface monitors were specified on the wall 20mm from the diffuser inlet and outlet to track p_1 and p_2 respectively. Standard initialization was used, k and ϵ were computed from the inlet. The residual convergence threshold for continuity, momentum, k , and ϵ was specified as $1E - 06$.

Model Validation

A mesh convergence study for the medium and long diffuser is shown in Figure 2 (validated for $p_2 - p_1$). A 3000 and 4000 node mesh was used to model the long and medium diffuser respectively. The geometry of the short and medium diffuser prohibited $y+$ from staying in the correct range for the standard wall function (see Figure 3). The solution for the medium diffuser did converge, however, the solution diverged for the short diffuser (see residuals in Figure 6). A more sophisticated model is necessary to accurately model the larger cone angles.

Results

The numerical and experimental results are shown in Figure 5. The cone angle is the dominating factor for a reduction in the PRC. The CFD models do not predict any PRC change due the Re number, but the experimental results do show some changes. This is because the CFD model is not fully capturing the complexity of the turbulent behaviour. Both the experimental and numerical results fall short of the theoretical prediction since the ideal case ignores turbulence effects completely. The large reduction in PRC between $\theta = 4.1$ and $\theta = 19.8$ is due to boundary layer separation in the diffuser (see Figure 4). The discrepancy between experimental and numerical results for the medium diffuser is due to using an insufficient model. The $k - \epsilon$ model is not designed to model flow separation. The $k - \omega$ SST model is a better model for flow separation.

Discussion & Conclusion

The results show that the cone angle has the most significant effect on the PRC. Increasing the static pressure of a 1D flow creates an adverse pressure gradient ($dp/dx > 0$).

$$\rho \left(u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) = -\frac{\partial p}{\partial x} + \mu \left(u \frac{\partial^2 u}{\partial x^2} + v \frac{\partial^2 u}{\partial y^2} \right)$$

Inspecting the 2D N-S equations above, if the static pressure is increased too quickly, i.e. dp/dx is large, it dominates the viscous effects and the RHS becomes negative, implying a reversal in velocity. This is when flow separation occurs. Flow separation greatly increases k due to a rapid expansion of the boundary layer into a turbulent region. This leads to a large drop in the PRC since $PRC = PRC_{ideal} - k$. Notice that increasing the velocity of the flow will delay flow separation and increase efficiency unless θ is large, in which case flow separation will occur and there is more kinetic energy to lose to turbulence, thus reducing efficiency. This matches the experimental observations.

The trade-off is then between length and efficiency. Shorter diffusers are more prone to flow separation and efficiency losses. Design should focus on avoiding flow separation.

Each of the three analysis techniques have their own merits and flaws. Theory can provide a limiting case, however, the ideal assumptions show that accurate predictions cannot be made if turbulent effects are neglected. The numerical model is more successful in matching experimental results since turbulent effects are considered, however, using the wrong model for the problem will also produce highly inaccurate results. This highlights the importance of understanding the limitations and applications of a model, and understanding the problem we are trying to model. The experimental approach is useful for validating theoretical and numerical models and understanding the physical behaviour of the problem. The main drawbacks of experiments are cost and also sample consistency. Experimental data usually has a high variance and so small datasets can be difficult to draw accurate conclusions from.

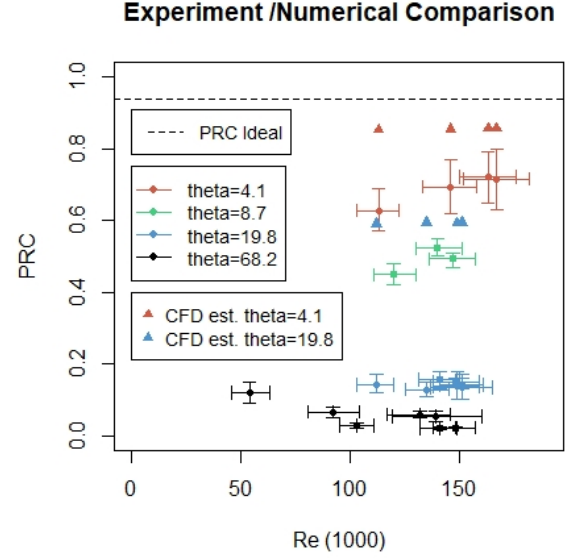


Figure 5: Shows the PRC as a function of Re for 3 unique cone angles and compares experimental data with numerical results.

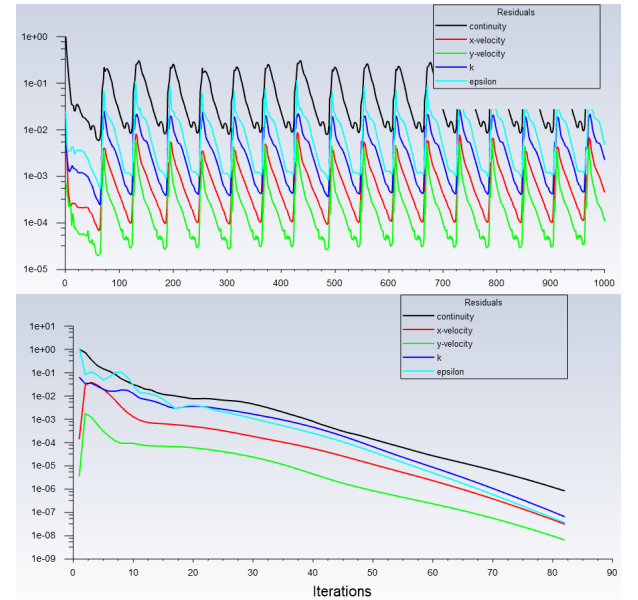


Figure 6: Shows the residuals for the short diffuser (top) and the medium diffuser (bottom). The solution diverges for the short diffuser and converges for the medium diffuser.