AIJREAS VOLUME 2, ISSUE 8 (2017, August) (ISSN-2455-6300) ONLINE
ANVESHANA'S INTERNATIONAL JOURNAL OF RESEARCH IN ENGINEERING AND APPLIED SCIENCES

DESIGN AND STEADY STATE ANALYSIS OF A TRANSITION DUCT

CH. VEERU NAIDU
M.Tech.(THERMAL)
Department of Mechanical Engineering,
kakinada Institute of technological sciences.
A.Agraharam- Ramachandrapuram East Godavari District.
Email id: veerunaidu308@gmail.com

MR.D.VIJAYA PRASAD
Associate Professor,
Mechanical Engineering Department,
kakinada inistitute of technological sciences.
A.agraharam- Ramachandrapuram East Godavari District.
Email id:vijayaprasad35@gmail.com

ABSTRACT

An experiential and numerical investigation of three duct geometries was performed to examine their relative performance and suitability for application as gas turbine exhaust ducts. A11 the ducts had the same length and area change and ali featured a transition from circular to oblong with axial offset. The aspect ratio and offset of the base line duct were increased to produce two additional geometries for investigation. The models were cold flow tested and nurnerically simulated using a commercial 3-D Navies Stokes solver using the Realizable k-E turbulence model. Three different conditions of zero, twenty and forty degrees of swirl were tested to examine the sensitivity of the ducts to the retag flow typical of varied flight conditions. The results show that static pressure losses through the ducts are larger for the increased offset and outlet aspect ratio ducts. Pressure recovery also decreased as the magnitude of the swirl increased. The total diffusion of 27% was not sufficient in al1 cases to prevent an increase in the inlet static pressure. The losses were shown to be a combination of inefficient diffusion resulting Come viscous losses and insufficient division resulting from significant velocity non-uniformity. The relative importance of these two factors varied with the combination of the severity of the swirl and duct geometry. The CFD solutions under predicted the flow losses but correctly identified the trends indicating the potential for CFD to provide useful information towards the design of such ducts.

1.0 INTRODUCTION

Transition to turbulence in ducts is still a challenging issue despite the 125 years since the seminal paper by Osborne Reynolds (1883) which led to the definition of a dimensionless number, that came some time afterwards to bear his name, capable of broadly discriminating cases of 'streamlined' flow from cases where 'sinuous motion' prevailed. Osborne Reynolds recognized also that there was no unique value of this dimensionless parameter, representing the ratio of convective to viscous forces, separating the two classes of motion, and that the end state was influenced by the background perturbations present. The problem of laminar-turbulent transition in shear flows was posed, to fascinate and attract the attention of thousands of researchers in the years to come. Incidentally, the story of how the paper was reviewed by two referees of the caliber of Lord Rayleigh and Sir George Stokes makes for instructive reading (Jackson & Launder 2007). In a subsequent paper, Reynolds (1895) goes beyond transition and presents, for the first time, the decomposition of the turbulent field into a mean and a fluctuating part, arriving at the equations now known as the 'Reynolds-averaged Navies-Stokes equations', with unknown turbulent stress terms in the mean flow equations

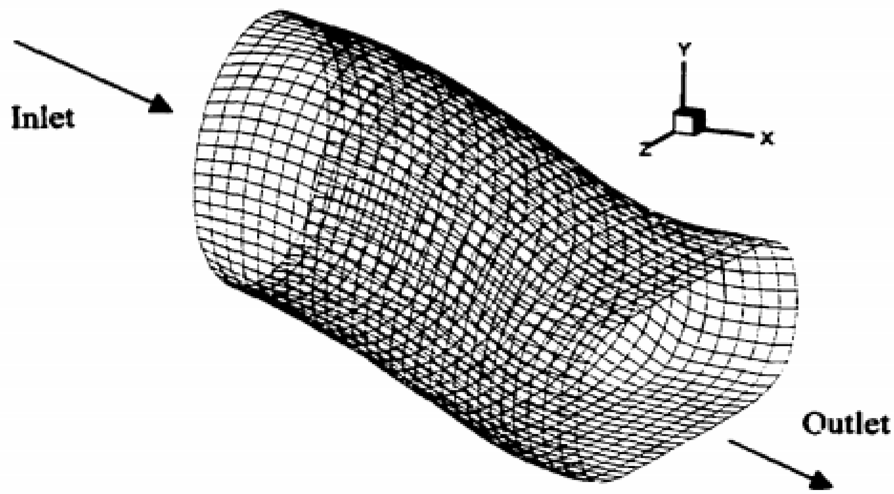


Figure 1.1 shows an example of this type of circular to oblong duct with offset.

1.2 DUCT COMPONENTS

Starting with the basics, let's start at the most elementary level by identifying components of a duct system. A duct system is a network of round or rectangular tubes—generally constructed of sheet metal, fiberglass board, or a flexible plastic and-wire composite—located within the walls, floors, and ceilings. Usually, you can see only the outlet, which is a register covered with grillwork. The purpose of a duct system is to transmit air from the central air source to the air diffusers located in the building control zones. Figure below shows a central heating furnace connected to supply and return air ductwork. The furnace is connected to the air plenum at the starting point. Furnace fan/s draw air in through grilles called returns and force air through the plenum and into the conditioned space through supply registers.

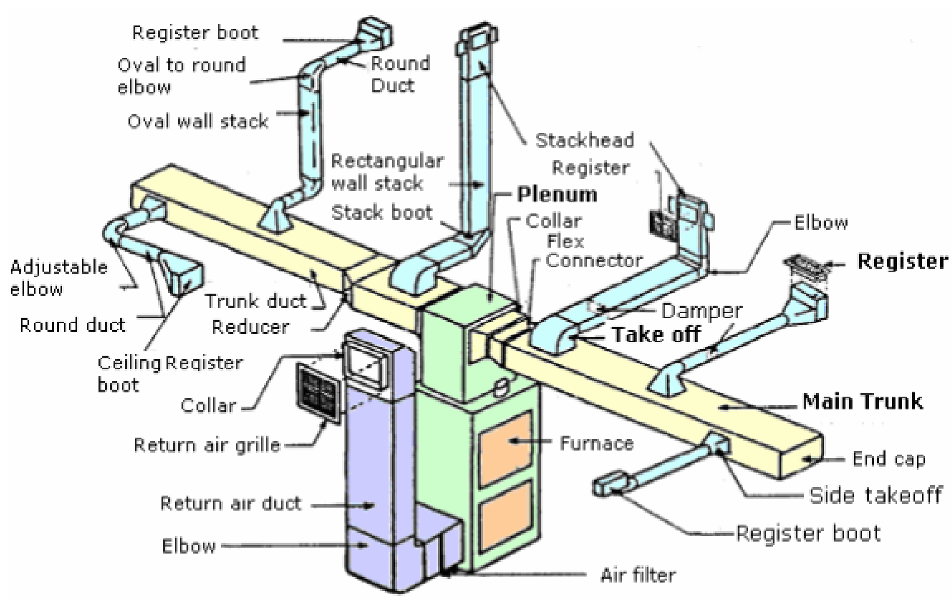


Fig 1.1 composite duct system

The Plenum The plenum is the main part of the supply and return duct system that goes directly from the air handler or furnace to the Main Trunk. What is the Main Trunk? The Main Trunk(s) are the part of the duct system that all the air from the system is going to travel in before we "take off" the main plenum to the "diffusers" or grilles A "take off "is that part of the system where we take the air off the trunk to supply air to the living area of the house. Then from the "take off" we will go directly to the Grilles, diffuser or registers. What is a Grille, diffuser or register? A system of fixed or adjustable vanes covering an opening through which air is discharged; Diffuser is an outlet devise discharging supply air in a direction radially to the axis of entry. Register is a grille equipped with a damper control valve.

Oval Ducts

Flat oval ducts have smaller height requirements than round ducts and retain most of advantages of the round ducts. However, fittings for flat oval ducts are difficult to fabricate or modify in the field. Other disadvantages of flat oval ducts include: difficulty of handling and shipping larger sizes; a tendency of these ducts to become more round under pressure; and, in large aspect ratios, difficulty of assembling oval slip joints.

SUPPLY DUCT CONFIGURATIONS

The configuration of a duct system is often like a tree with branches connected to the terminal units and a fan located at the root. In reality ductwork forms a double tree because the fan is in the middle of supply and return/outside air parts of the system. The two most common supply duct systems are the 'extended plenum' system and the 'radial' system because of their versatility, performance, and economy. The spider and perimeter loop systems are other options depending on space type and other design considerations.

2.0 LITERATURE SURVEY

Reichert et al. (1994) determined experimentally that for a non-swirling flow entering a circular to rectangular duct with constant area, a distribution of static wall pressures was produced. in the early part of the duct, maximum values were observed on the top and bottom surfaces where a reduced cross sectional distance forced the flow to converge. The side walls where the increased width allowed the flow to diverge saw the minimum static pressure values. Further downStearns the reverse trend was observed where streamline curvature was reversed and the flow forced to move downstrearn. Accompanying this static pressure distribution across the flow path is a developing cross flow. This flow can be visualized by velocity vectors directed from regions of higher static pressure to lower regions and creates counter rotating vortices. They observed in experimental traverses and with oil film visualization that, due to the symmetry of the duct, four identical counter rotating vortices were developed in each corner of the rectangular shape when the inlet flow had no swirl. Accompanying these four larger vortices are two more pairs of smaller vortices which exist along the two axes of symmetry. See Figure 2-2 **(a).**

Davis and Gesner (1992) reported that for a similar transition duct, they observed a similar pressure driven cross flow that resulted from the curvature of the duct walls and created contra rotating vortex pairs near the diverging side walls.The work of Sobota and Marble (1989) experimentally examined flow through an annular to rectangular transition duct with inlet swirl resulting fiom both 15 and 30 degree straight blades. They reported that along the centerline of the rectangular outlet there was a single vortex for the lower swirl angle and two CO-rotating vortices for the higher swirl angle. They suggested thot with the cross sectional change from annular to rectangular, a separation was induced along the outer wall.

Ban sod and Bradshaw (1972) observed that at the outlet plane of several s-shaped ducts they tested there was a large region of low total pressure centered at the bottom of the outlet. See Figure 2-5. They attributed the presence of this low velocity region to a pair of contra-rotator vortices in the boundary layer at the bottom center of the outlet which pushed low velocity fluid towards the center. The contra-rotating stream wise vortices are suggested to have formed from the pressure gradients that exist to balance out the centrifuga1 force of the fluid as it is timed. The vortex pairs evident in the outlet plume are formed in a similar manner to the vortices that develop in the transition ducts discussed above. Karloff et al. (1993) reported experimental and computational results for a diffusing s-duct which showed the vortex pairs resulting from secondary flows riven by pressure gradients.

Nallasamy (1986) reviewed the application of the currently available models for in teal turbulent flow. He concluded that despite it's widespread use for industrial flows, the performance of the standard k-E model becomes poor (including poor prediction of the size and location of recirculation zones) as the flow increases in complexity from attached to recirculating to swirling.

While offering improved performance for flows with streamline curvature and secondary flow, the Reynolds stress models required much more processor time In al1 cases, the quality of the inlet conditions provided to the model were seen to influence results. Armfield and Fletcher (1989) compared numerical results from two types of k-s and two types of RSM with experimental results for a swirling flow in a diffuser with 20° walls. They also reported improved performance with the RSM over both the standard k-E model and one with a swirl modification, although the swirl modification did result in some improvement to the prediction of the flow field. The fault of the k-E models is that the eddy viscosity prediction is too high.

3.0 METHODOLAGY

The design of diffusers requires the use of parameters that allow cornparisons between different designs or operating conditions. The following section presents some theoretical background. This discussion is limited to theory applicable to subsonic, incompressible flows.

3.1 Dimensional Analysis

For the analysis of the ducts in this study, murderous variables impact the performance of the ducts. A dimensional analysis reveals the following expression for the factors that influence the performance of the duct. Performance = f (Geometry, Fluid Properties, inlet Flow Conditions) Where, Geometry includes: L, D, S, H, W, A2 Fluid properties includes: Speed of sound, p, p Inlet conditions includes: Velocity, S whirl A standard dimensional analysis will reveal the following dimensionless groups: Re, Ma, Swirl, Lm, Ms., W/H, A2/Ai, In this study, the following parameters were not varied: Lm, A2/Al The following parameters could not be independently controlled due to the nature of the experimental apparatus: Re, and Ma

However, since the Mach number is below 0.3, compressibility factors will not be important and thus the Ma number is not critical. The tests were conducted at as high a Reynolds number as possible however the test Reynolds number does not match the Reynolds number of the actual full scale application. A Reynolds number independence test was performed to try and determine if the tests were conducted within the Reynolds number independence range. The following parameters were varied during the pararnetric study portion of this investigation: Swirl, WM, S/D

3.2 Diffusers

The performance of a diffuser can be quantified by several different performance parameters. The following includes information from the work of Some and Clomp (1 967). One of the important characteristics of a diffuser is the outlet profile it produces. Depending on the application, a certain outlet flow profile may be desired. However, if the diffuser has complicated geometry and includes a swirling flow at the inlet, the possibility of a non-uniform flow at the outlet increases. One method of quantifying the magnitude of this non unifonnity is the effective area. The effective area is a measure of the area which could pass the identical volume flow rate if it had a union velocity profile of a magnitude equal to that of the peak velocity in the cross section. The greater the non-unifonnity of the flow, the smaller the effective area is. The effective area is defined below.

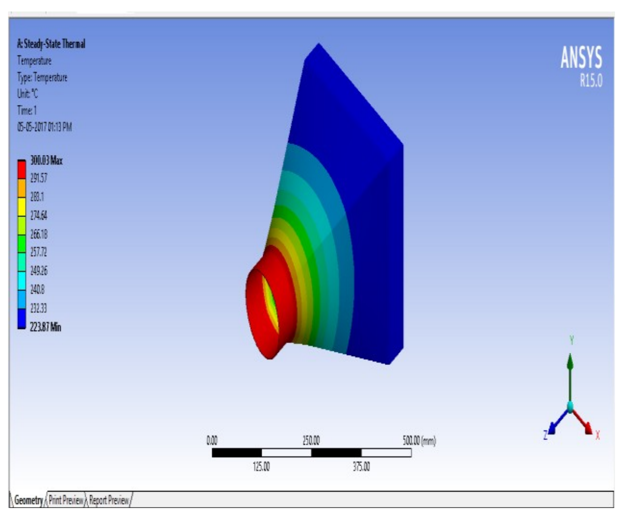
It must be noted that the effective area ratio is an averaged parameter and thus, two entirely different lowing flow profiles could have the some effective area. The second purpose of a diffuser is to increase the static pressure of the flow. The pressure recovery coefficient relates the actual rise in static pressure to the maximum possible pressure rise equal to the inlet dynamic head.Note that due to the significant tangential flow present at the inlet for the swirling cases, the inlet dynamic pressure is calculated using the square of the mass averaged velocity magnitude rather than the axial velocity. The ideal pressure nose coefficient is simply a function of the geometric area ratio of the diffuser. The difference between the ideal and experimental is a result of viscous losses which reduce the potential for full pressure recovery from the velocity decreasethat would accompany the diffusion, and from the effects of the non in front velocity and flow separation at the outlet which mates a skewed velocity profile.

Table 3.2: Duct outkt measuremeats.

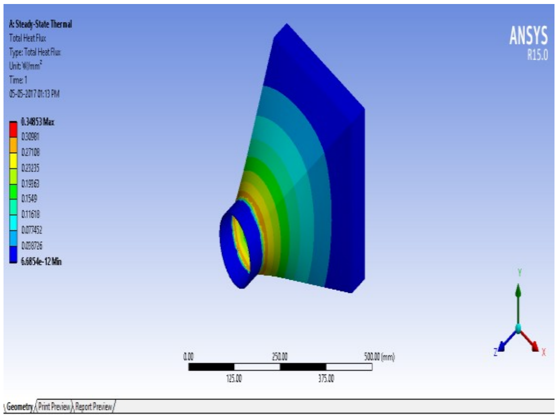
	Duct1	**Duct2**	**Duct3**
Mesured Aspect ratio	2.92	4.64	2.86
Integrated area of outlet traverse	0.01641	0.01635	0.01661
Tracted out let area	0.01637	0.0162	0.01634
Integrated area of in let traverse	0.01297	0.01297	0.01297
Actual diffusion	26.5%26.2%	26.5% 26.2%	28.1%26.0%

4.0 RESULTS

Copper alloy:

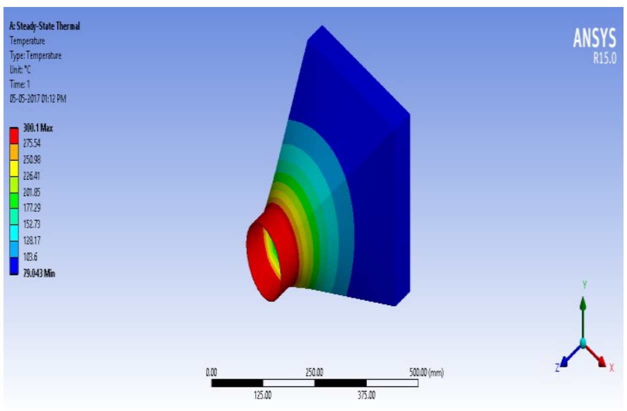


The fig 4.1 shows that stedy state thermal maxmum bending moment 300.03and 223.187minimum values.

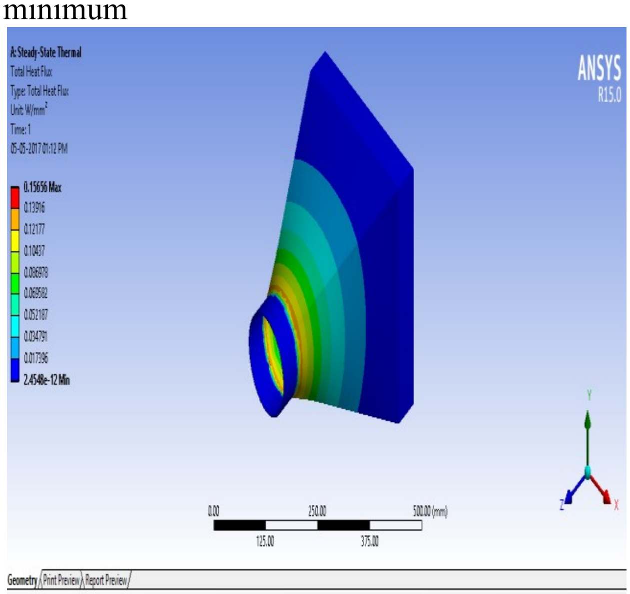


The study state thermal total heat flux maximum 0.34853&minimum 6.685maximum

Gray cast iron alloy

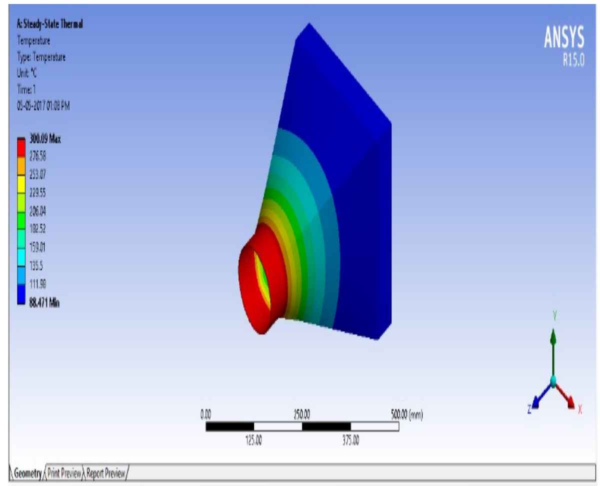


The fig4.3 shows that gray cast iron maximum bending moment 300.1&29.043 minimum



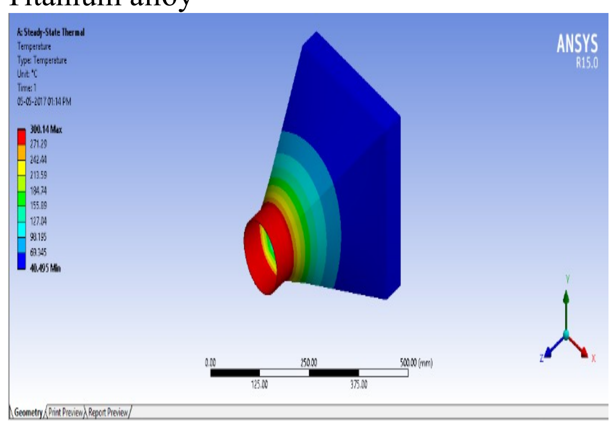
The fig 4.4 shows that gray cast iron maximum and minimum bending moments 0.015656 & 2.45e-12.

Structural steel



The fig 4.6 shows that structural steel maximum and munimum bending moments 300.09&88.471

Titanium alloy



The fig 4.8 shows that the titanium max&min values 300.14&40.495

5.0 CONCLUSION

Based on the results of thîs study, it is concluded that: The offset resulted in a redistribution of the fiow at the outlet and a cross strearn velocity fiom top to bottom. As the offset increased, flow nonunifonnity and losses increased. For an increase in outlet aspect ratio, the pressure losses increased. Flow non unifonnity and losses increased with swirl. The swirling annular flow was significantly conserved through the ducts.

The static pressure coefficients were positive or neutral for the zero and twenty degree cases of duct 1 and 2. At the higher 40 degree swirl level and for al1 of the high offset duct 3 tests, the 27% diffision was not sufficient to prevent an increase in inlet static pressure. The losses incurred were shown to be a combination of insufficient diffusion resulting fiom flow non-uniformity, and inefficient diffusion resulting f?om viscous losses.

The relative magnitude of these two factors 106 changed with the combination of increased swirl and more severe duct geometry. The combination of the high 40 degree swirl and the more severe geometries of ducts 2 and 3 combined to produce better pressure recovery than for a comparable case with the baseline duct 1. For these geometries, CFD consistently erred in the prediction of pressure recovery . CFD can reasonably predict trends in the overall performance of a family of difbing ducts with aggressive geornetry such as those examined in this snidy thus contributing to the design cycle but not eliminating the need for accurate experimental test in.

REFERENCES

- *Agrawal, Y., Talbot, L., and Gong, K., 1978, "Laser Anemometer Study of Flow Development in Curved Circular Pipes,"*
- *Journal of Fluid Mechanics, Vol. 85, part 3, pp. 497-5 18.*
- *Amfield, S.W. and Fletcher, C.A.J., 1989, "Comparison of k-s and Algebraic Reynolds Stress Models for Swirling Diffiiser Flow," International Journal for Nunrerical Methodr in Fluids, Vol.*
- *Bansod, P., and Bradshaw, P., 1972, "The Flow in S-shaped Ducts," Aeronauticd Quarterb, May 1972, pp. 13 1 - 140. Beckwi th, T.G., Marangoni, RD., and Lieahard, J.H., 1 993, Mechanical Measurements, S~ ed., Addison Wesley Publishing Company.*
- *Burley, J.R, Bangert, L.S., and Carlson, J.R., 1986, "Static investigation of Circular-toRectangular Transition Ducts for High-Aspect-Ratio Nonaxisymmetric Nozzles," NASA TP-2534. Cohen, H., Rogers, G.F.C., and Saravanarnuttoo, H.I.H., 1996, "Gas Turbine T7ieor-y ", 4h ed., Longman Group Lirnited.*
- *Chyu, W.J., and Bencze, D.P., 1992, Wavier-Stokes Simulation of Flow Through a Highly Contoured Subsonic Diffuser," International Journal for Numerical Methods in Engineering, Vol. 34, pp.473-483.*