

**“DESIGN, FABRICATION AND COMPUTATIONAL STUDY ON
FLOW CHARACTERISTICS OF
S-DUCT AND Y-DUCT”**

A Main project report submitted to the
SATHYABAMA INSTITUTE OF SCIENCE AND TECHNOLOGY, CHENNAI
In partial fulfillment of the requirements for the award of the Degree of
BACHELOR OF ENGINEERING
IN
AERONAUTICAL ENGINEERING
Submitted by

VASANTH K

(Register. No: 37260047)

GIRISH KUMAR GS

(Register. No: 37260014)

Under the Guidance of

Mr. VENKATESH

Assistant Professor



DEPARTMENT OF AERONAUTICAL ENGINEERING
SATHYABAMA INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
(Accredited with Grade “A” by NAAC/ 12B Status by UGC/Approved by
AICTE)
Jeppiaar Nagar, Rajiv Gandhi Salai, Chennai – 600 119



SATHYABAMA

INSTITUTE OF SCIENCE AND TECHNOLOGY

(DEEMED TO BE UNIVERSITY)

Accredited with “A” grade by NAAC

Jeppiaar Nagar, Rajiv Gandhi Salai, Chennai – 600 119

www.sathyabama.ac.in



DEPARTMENT OF AERONAUTICAL ENGINEERING

BONAFIDE CERTIFICATE

This is to certify that this Project Report is the bonafide work of **VASANTH K (37260047)** and **GIRISH KUMAR GS (37260014)** who carried out the project entitled “**DESIGN, FABRICATION AND COMPUTATIONAL STUDY ON FLOW CHARACTERISTICS AND OF S-DUCT AND Y-DUCT**” under our supervision from SEPTEMBER 2020 to MARCH 2021

Internal Guide: Mr. VENKATESH S

Head of the Department: Dr. P BOOMA DEVI M.E, PhD

Submitted for Viva voce Examination held on_____

Internal Examiner

External Examiner

DECLARATION

I **VASANTH K (37260047)** hereby declare that the Project Report entitled **“DESIGN, FABRICATION AND COMPUTATIONAL STUDY ON FLOW CHARACTERISTICS AND OF S-DUCT AND Y-DUCT”** done by me under the guidance of Dr. /Prof./ **Mr. VENKATESH S and Dr. VALARMATHI** at is submitted in partial fulfillment of the requirements for the award of Bachelor of Engineering / Technology degree in **Sathyabama institute of science and technology**.

DATE:4/3/2021

PLACE: Chennai

SIGNATURE OF CANDIDATE

DECLARATION

I **GIRISH KUMAR GS (37260014)** hereby declare that the Project Report entitled “**DESIGN, FABRICATION AND COMPUTATIONAL STUDY ON FLOW CHARACTERISTICS AND OF S-DUCT AND Y-DUCT**” done by me under the guidance of Dr. /Prof./ **Mr. VENATESH S** and **Dr. VALARMATHI** is submitted in partial fulfillment of the requirements for the award of Bachelor of Engineering / Technology degree in **Sathyabama Institute of Science And Technology**.

DATE: 04/03/2021

PLACE: Chennai

SIGNATURE OF THE CANDIDATE

ACKNOWLEDGEMENT

I am pleased to acknowledge my sincere thanks to Board of Management of **SATHYABAMA** for their kind encouragement in doing this project and for completing it successfully. I am grateful to them.

I convey my thanks to **Dr. S Prakash M.E., Ph.D.**, Dean, School of ----- Mechanical engineering and **Dr.---Mrs. Booma Devi M.E., Ph.D.** Head of the Department, Dept. of Aeronautical engineering for providing me necessary support and details at the right time during the progressive reviews.

I would like to express my sincere and deep sense of gratitude to my Project Guide **Mr. Venkatesh** for his valuable guidance, suggestions and constant encouragement paved way for the successful completion of my project work.

I wish to express my thanks to all Teaching and Non-teaching staff members of the **Department of Aeronautical engineering** who were helpful in many ways for the completion of the project.

ABSTRACT

S-duct and Y-duct diffusers are used in air-intake system of many types of jet aircrafts to get the drag lesser and lowering the position of engine compared to straight through design. In Aircraft propulsion systems they often use Y-shaped subsonic diffusing ducts as air-intakes to supply the ambient air into the engine compressor for generation of thrust . In this study the distorted flow condition parameter like total pressure loss is investigated. CATIA V5 is used for design the required diffusers and ANSYS fluent is employed for the investigation of both Y-duct and S-duct for the different Mach numbers. The design objective is to improve the static pressure recovery and reduce the total pressure loss and also drag in both the ducts and comparing them with respective to each other.

CONTENTS

Chapter No 1

1. Introduction	1
1.1. Motivation and Objective	
1.1.1. Motivation	1
1.1.2. Objective	2
1.2. Introduction to Intakes	2
1.2.1. Subsonic Inlets	3
1.2.2. Supersonic Inlets	4
1.2.3. Hypersonic Inlets	5
1.3. S-Duct	5
1.4. Y-Duct	6

Chapter No 2

2. Literature Survey	8
2.1. Reviews on Literature	8

Chapter No 3

3. Experimental Setup and Procedure	12
3.1. Air Jet Experimental Setup	12
3.1.1. Compressed Air Supply System	12
3.1.2. Free and wall jet test setup	13
3.1.3. Traverse Mechanism	15
3.1.4. Pitot Probe	15
3.2. Calculations	16
3.2.1. Calculation of Head Mach Number	16

3.3.	Design of Models	17
3.3.1.	Geometry Design Of S- Duct	18
3.3.2.	Geometry Design Of Y-Duct	19
3.4.	Fabrication of Model	23
3.4.1.	3D-Design machine	23
3.4.2.	Modelling	24
3.4.3	Printing	25
3.4.4	Finishing	26
3.5.	Experimental Procedure	28
Chapter No 4		
4.	Computational Analysis	
4.1.	Introduction to Ansys Fluent	29
4.1.1.	Ansys Workbench Applications	29
4.1.2.	Ansys Cfd	29
4.1.3.	Ansys Icem Cfd	29
4.1.4.	Ansys Ls-Dyna	29
4.2.	Procedure to Solution by Ansys	29
4.2.1.	Geometry	29
4.2.2.	Meshing	30
4.2.3.	Solver Setup	30
4.2.4.	Solution	34
4.2.5.	Result	36
4.3.	Computational Procedure	36
4.3.1.	Geometry	36
4.3.2.	Meshing	36
4.3.3.	Solver Setup	37

Chapter No 5

5. Results and Discussion	38
5.1 Flow Analysis through Y-Duct	38
5.1.1 Variation pressure and temperature along the duct centerline	38
5.1.2 Variation Mach number along the duct centerline	39
5.2 Flow Analysis through S-Duct	42
5.2.1 Variation of pressure and temperature along wall and centerline of duct	42
5.2.2 Variation Mach number along the duct	44
5.3 Centerline Comparison of Ducts for different Mach numbers	47
5.3.1 Centerline comparison of pressure	47
5.3.2 Centerline comparison of temperature	48

Chapter No 6

6. Summary and Conclusion	50
6.1 Conclusion	50
6.2 Future Work	50
7. References	51

List of Figures

S. No	Figure. No	Name of the Figure	Page No
1	1.1	Subsonic Inlet	3
2	1.2	Supersonic Inlet	4
3	1.3	Hypersonic Inlet	5
4	1.4	S-Duct	6
5	1.5	Y-Duct	7
6	3.1	Layout of the open jet facility laboratory	12
7	3.2	Schematic of free and wall jet setup	13
8	3.4	Photograph of traverse mechanism	15
9	3.5	Schematic diagram of pitot probe	16
10	3.6	Design of s duct	18
11	3.7	3D view of s duct	19
12	3.8	2D view of s duct	19
13	3.9	Part design of y duct	20
14	3.10	Mirroring of y duct	21
15	3.11	3D of y duct	21
16	3.12	Design of y duct	22
17	3.13	2D of y duct	22
18	3.14	3D Design of S Duct	25
19	3.15	3D Design of Y Duct	26
20	3.16	Filament Material	27
21	4.1	Process of Pressure Based Solver	31
22	4.2	Process of Density Based Solver	32

23	5.1	Centerline Pressure variation through Y-duct for different Mach numbers	38
24	5.2	Centerline temperature variation through Y-duct for different Mach numbers	39
25	5.3	Pressure variation through S-duct for different Mach numbers	42
26	5.4	Pressure variation through S-duct for different Mach numbers	43
27	5.5	Temperature variation through S-duct for different Mach numbers	43
28	5.6	Mach number variation through the Y-Duct at Different Mach numbers	47
29	5.7	Centerline comparison of pressure	48
30	5.8	Centerline comparison of temperature	49

List of Tables

S. NO	Table No	Name of the Table	Page No
1	4.1	Units of Mesh for FFF of Y-duct & S-duct	36
2	4.2	Mesh Information for FFF of Y-Duct	36
3	4.3	Mesh Information for FFF of S-Duct	36
4	4.4	Sizing Information for FFF of Y-duct & S-duct	37
5	4.5	Setup conditions and Boundary conditions of Y-duct & S-duct	37

Nomenclature

P_o	=	Settling chamber pressure or stagnation Pressure
P_a	=	Atmospheric pressure
P_e	=	Exit pressure
H	=	Pressure head rise
V	=	Flow Velocity
a	=	Speed of sound
M	=	Mach number
X	=	Co-ordinate along jet axis
Y	=	Co-ordinate along lateral axis
Z	=	Co-ordinate along transverse axis
X/L	=	Axial distance to the exit distance ratio
CPD	=	Centre line pressure distribution

CHAPTER 1

INTRODUCTION

An Aircraft is a vehicle that flies in the space by gaining the support from air. Each aircraft must contain primary parts like nose cone, body, landing gear, power controls, wings. Engines are the major source for power controls. Engines are of many types. Some of them are having turbines and some of them have no turbines. Aircrafts having turbines have parts like intake, compressor, combustion chamber, turbine and finally nozzle. This are simply called as gas turbine engines. Similarly, for non-turbine engine it contains only intake, combustion chamber, and nozzle. This are simply called as gas RAMJETs. In any type of engine intakes play a crucial part. These intakes are located just a head of the compressor. This Intakes are used to give uniform flow to the compressor so that the blades of compressor are not affected by the flow direction. Due to this the pressure is distributed evenly to the compressor. The overall efficiency of the engine also depends on the inlet efficiency of the engine. Good design of inlet lead to the good efficiency.

1.1 Motivation & Objective

1.1.1 Motivation

An inlet prepares the operating conditions for the engine as the first part of a propulsion system. It is highly important for the engine's performance and safety during the flight. An inlet's job is to slow down incoming airflow by translating as much of the kinetic energy into static pressure at the compressor face as possible. Since an inlet is where the air comes in, it's essential to have one.

. On their flights, missiles and tactical aircraft are not supposed to be identifiable. The most sensitive components of radar signals are spinning blades. As a result, placing the engine within the plane, and thereby out of direct radar range, is the most basic solution to achieving good camouflage properties. Serpentine inlets are needed where the engine is situated within the fuselage. As a result, S-duct inlets became an important part of aircraft design

because they enable the engine to be submerged in the fuselage and reduce radar visibility by shielding the engine face from direct radar signals. Furthermore, as opposed to a traditional pylon-mounted engine, submergence of the engine inlet decreases inlet ram drag. The displacement of the pylon also has a detrimental effect.

The use of serpentine inlets in the aerospace industry has been increasingly growing recently due to the above factors. The defence sector prefers them for missile and combat aircraft projects. The success of serpentine inlets stems from their ability to limit aircraft visibility and mass while still increasing the amount of fuel tanks.

1.1.2 Objective

The aim of this study is to use experimental and theoretical approaches to examine the aerodynamic properties of an aircraft's S duct diffuser.

The cause of pressure loss at the aerodynamics interface plane is studied using flow characteristics such as separation regions. The instrumented S-duct diffuser is used in the experiments. Experiments are used to test the simulations.

1.2 Introduction to Intakes

Most of the modern aircrafts of both military and passenger are mostly powered by gas turbine engines. Intakes are the most visible part of the aircraft propulsion system and are also the opening for the aircraft engine. This are used to give optimal functioning to flight at different conditions. This are used to increase mass flow rate to the engine, which leads to the increase of net thrust of the aircraft. This are also helpful in reducing the aircraft sound acoustics. Intake performance depends mainly on the its geometry, pressure difference and mass flow rate. By the advancement in aircrafts in aviation field the ability of aircrafts is increased to supersonic and even to hypersonic speeds. It has become compulsory to design inlets to provide required flow while

operating in higher mach numbers to overcome the problem of high-pressure recovery and low distortion. Intakes which are operating in transonic mach numbers are having variable geometry. Inlets happens to in verity sizes and shapes. These intakes are displayed in the shape of divergent ducts. This are mainly systematized based on their engine speed or speed of aircraft.

1.2.1 Subsonic Inlets:

Aircraft that go less than speed of sound i.e. $M < 1$ are usually subsonic intakes. Inlets with simple and straight and short works good. The front portion of the inlet is called the higher or the inlet lip. A subsonic aircraft has an inlet relatively thick lip shown in Fig 1.1 shown below. Generally subsonic inlet flights are used for the civil aircrafts.



Fig. 1.1 Subsonic Inlet

This are typically classified into two types They are mainly used in transport aircrafts They are:

- Internal Compression Subsonic Intakes
- External Compression Subsonic Intakes

1.2.1.1 Internal Compression Subsonic Intakes:

1. Pressure gradient is very Low enough to avoid large stagnation pressure
2. Act as a divergent duct for these types.
3. This are having low angles to increase the length of the portion which deals with friction losses.

1.2.1.2 External Compression Subsonic Intakes:

1. Due to large length in the duct the boundary layer in them leads to losses like skin friction losses, if the compression of the gas is made to occur before it enters the passage the losses can be reduced. This type is called external compression.
2. This type of intakes is useful for high subsonic intakes.

1.2.2 Supersonic Inlets:

Aircraft that go more than speed of sound i.e. $M > 1$ are usually supersonic intakes. These inlets have relatively sharp lip. This lip is made sharp to reduce the shock wave losses that occur during flight. This type of inlets is designed in such way that maximum pressure is increased only at inlet. As its flight condition is greater than 1 in terms of mach numbers there will be the formation of shock. This is designed in such a way that normal shock is formed in the inlet before reaching to compressor. This is mainly used in RAMJET. Figure 1.2 shows supersonic inlet.

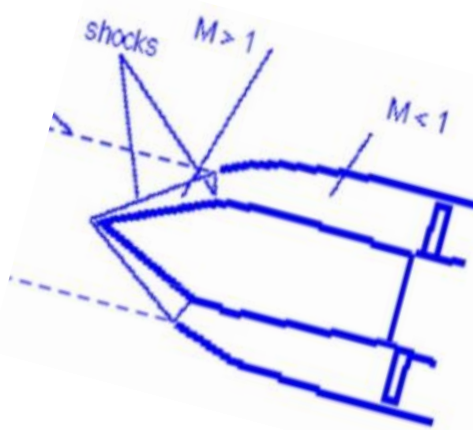


Fig. 1.2 Supersonic Inlet

1.2.3 Hypersonic Inlets:

Aircraft that go more than speed of sound i.e. $M > 5$ are usually hypersonic intakes. The air flows with very high speed almost greater than 5 in terms of

mach numbers there will definitely be formation of shock. This is designed in such a way that no normal shock is formed in the inlet before reaching to compressor. This is mainly used in SCRAMJET. Due to these high pressures the temperature is of high. Due to this they cause drastic effect on the boundary layer. As there is no formation of normal shock the flow is supersonic at the exit. Figure 1. 3 shows hypersonic inlet.

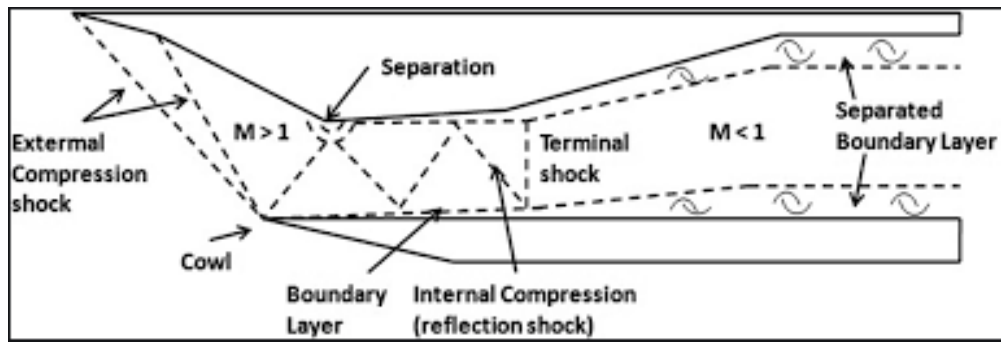


Fig. 1. 3 Hypersonic Inlet

1.3 S-Duct:

S-duct diffusers are often used for aircraft propulsion systems that convey the intake air to the engine compressor. S-ducts are used in engine intakes of civil and military aircraft. One of the design objectives is to minimize the loss of pressure mainly the one arising from separation of the flow (turbulence). So, being able to correctly predict the occurrence of such flow separation in the several conditions of pressure, temperature, speed and mass flow that an aircraft S-duct is submitted to is really difficult and of great interest in aerodynamics.

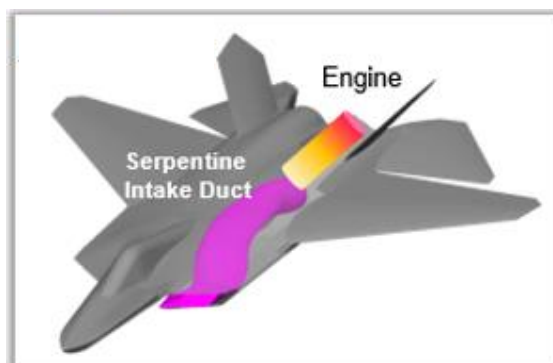


Fig. 1.4 S-Duct

S-duct design is more prominent in aircraft industry. This design is approached by some aircraft manufacturers for their trijet aircrafts to get less drag and lowering the engine position compared to straight through design. This design also allows a shorter fin and a rudder closer to longitudinal axis. But due to its curved shape engine performance is compromised. Figure 1.4 shows S-Duct.

1.4 Y-Duct:

The expulsion of boundary-layer fluid by solid longitudinal vortices embedded in the boundary layer near the outside of the second bend has been due to broad deviations from axisymmetric in the escape plane of a Y-shaped duct (dorsal or wing-root intake) (the bottom of a dorsal intake). The process that causes these vortices to form is a mixture of cross-flow and convergence.. The presence of vortices formed by a similar mechanism in a two-dimensional wind tunnel contraction has been deduced from the measurements of an unnaturally thick and distorted boundary layer near the center line of the working section wall. It is suggested that the vortex-producing mechanism may occur in other flows. Provided that the engine is not too sensitive to departures from axisymmetry, useful decreases in overall total-pressure loss may be obtained by using shorter intakes than those typical of modern commercial aircraft practice. The "built-in vortex generator" effect probably helps to suppress separation, at the expense of a more widely-distributed region of low total pressure. Present boundary-layer estimation techniques are unlikely to have useful results in such a dynamic wave. Unfortunately, the rig was too small to test boundary layer fences for minimizing cross-flow and vortex formation. It may be unwise to suppress the vortices if they help to avoid separation, but it might be possible to reduce the non-uniformities of total pressure generated.

Figure 1.5 shows Y-Duct.



Fig. 1.5 Y-Duct

Chapter 2

LITERATURE SURVEY

2.1 Reviews on Literature:

The properties of fluid motion within S-duct diffusers have been determined by previous experimental and analytical studies. Mathematically specified duct shapes have been built to provide typical geometries for studying this flow. This section reviews experiments that are close to the current research in terms of targets from the available literature. Both laboratory investigations and computational research will be highlighted. Boundary layer separation and secondary flows are two major phenomena that cause loss of output and distortion at the compressor face in S-duct air inlets. Several experiments have been conducted to further explain the causes of these phenomena, both theoretically and experimentally.

At $M=0.6$ and $Re=2.6 \times 10^6$, Wellborn et al [1] studied compressible flow through a serpentine inlet experimentally. The flow has symmetric actions in the plane with geometrical symmetry. After the duct bends, regions of streamwise flow separation can be observed. As a consequence of the duct curvature, secondary flows appear, which transform into a pair of counterrotating vortices.

Harloff et al [2] discovered that using the same turbulence models, transverse flow patterns were estimated to be greater than experimental transverse velocities in the area of the counter-rotating vortices and less than in the central flow region.

Fiola and Agarwal [3] conducted the numerical simulations of compressible flow inside an S-duct using Fluent commercial code. The three-dimensional RANS equations were solved on a structured mesh with the Spalart–Allmaras (SA), $k-\epsilon$, $k-\omega$ SST, and Transition SST turbulence models. A second-order accurate, steady, density-based solver was employed. The computed results of the flow field and pressure recovery were agreed with the experimental data. However, some discrepancies were observed and further improvements in the computational model were needed. It was found that $k-\omega$ SST turbulence model

gives best agreement with the experimental data in predicting the pressure distribution along the duct at various angular locations, the separated flow region (both the separation and reattachment points), and the secondary flow.

At different flow speeds, Aslan et al... examined the aerodynamic properties of an S-duct inlet designed for a micro turbojet engine driven aircraft. Flow isolation happens stream by stream within the conduit, according to dimensions. As a function of the flow structure, there is a complete pressure loss.

B. J. Prior and C. N...[5] Hall studied the effects of various types of air intake on the longitudinal characteristics of a modern fighter aircraft at Mach numbers up to 0.93 by evaluating a series of full models of nose, hand, or wing-root intakes and comparing their behavior to that of a datum model with no intake. The type of intake installed is found to have only minor effects on the lift and pitching moment characteristics but considerable variation in external drag occurs between the different models. A single intake placed in the nose gives less drag over the whole of the speed than either form of divided intake the installation of intake having been achieved without detriment to the design. In spite of increased frontal area in fact the combinations of the intake lay-out with shaping of the wing-body junction results greatly reduced transonic drag. The wing-body junction's shape, when paired with the nose and side intake installations, resulted in a significant reduction in drag at transonic speeds, which the wing-root layout did not provide.

W. J. G. and J. Seddon... [6] Trebble looked at twin air intake systems (i.e., a pair of intakes discharging into a typical vent or chamber) in which external boundary layers impact the losses. Below a certain critical value of the flow coefficient, asymmetry of flow between the two ducts occurs. The impact of this asymmetry on intake performance, and particularly flow distribution at the compressor, may be significant. If the flow oscillates between the two sides, as appears to be the case, the aircraft can vibrate. A pair of wing-root leading-edge intakes, as well as different arrangements of body-side submerged intakes, have been tested in the wind tunnel. A area of flow asymmetry was found in both situations. The required flow coefficients are those that may be

found in a dive or while abruptly throttling back in level flight, but are beyond the key working range of the intakes. The key factors that influence the degree of asymmetry are briefly discussed. A theory of intake loss has been modified to provide a method for estimating critical flow coefficient.

.

Kristina, et.al....[8], studies experimentally and numerically passive scaled-up micro mixer based on fluid rotation over Reynolds number ranging from 0.5 to 100 Micro mixers are broadly categorized as either passive or active. Active micro mixers make use of external fields and forces, including moving membranes and artificial cilia and electromagnetic and acoustic fields to promote mixing. Passive micro mixers rely on periodic variations in channel geometry to promote mixing. Although active micromixers are typically more effective mixers than their passive counter parts, they are more costly, bulky, and difficult to integrate into analysis systems. These drawbacks render passive micro mixers preferable Flow visualisation is used to determine flow patterns as they are being generated along the channel length. A Y-inlet supplies two different fluids to measure the section... Numerical simulations show that rotation occurs at all Reynolds numbers studied, and that the strength of rotation increases as the Reynolds number increases. Dean Vortices are formed as a result of the serpentine channel formation, which uses grooves to facilitate helical movement.

Chapter 3

EXPERIMENTAL SETUP AND PROCEDURE

3.1 Air Jet Experimental Set up

The experiment was conducted in the free and wall jet test setup at the Aerodynamics laboratory, Sathyabama institute of science and technology. The test facility consists of compressed air supply system and jet test setup. The layout of the experimental jet facility laboratory is as shown in Fig.3.1.

3.1.1 Compressed air supply system:

An air source compressor with a single stage of action. In this laboratory, an air compressor capable of delivering 100 ft³/min is being used. A 20 hp three-phase induction motor powers the compressor. An induction motor powers a cooling fan.

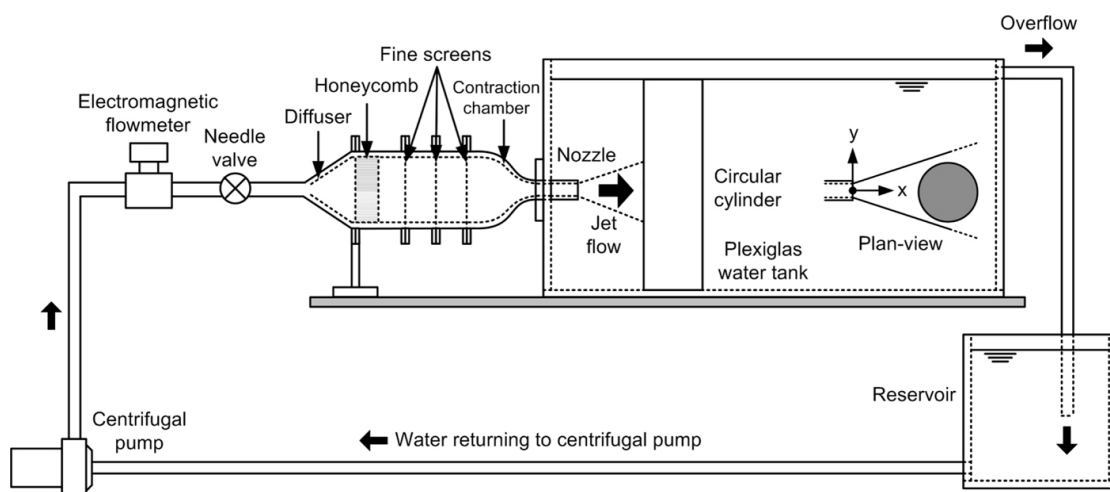


Fig. 3.1 Layout of free and wall jet setup

To extract solid pollutants such as rust particles and oil droplets, the air is passed through a pre-filter made up of porous stone candles. For finer filtering, an activated carbon filter is used.

The dryer will run at 250 psi thanks to a diaphragm style back pressure valve controlled by a pressure relief pilot, as the pressure in the storage tanks rises from atmospheric to the necessary storage pressure. The compressed air is held in two tanks with a combined volume of 600 pounds per square inch. The tunnel control portion contains a gate valve that is attached to a 3-inch-diameter mixing tube. above meter length and then to a settling chamber, the safety operation purpose of tanks the pressure relief valves placed at top of the tanks sets 200 psi.

3.1.2 Free and Wall Jet test setup

The experiments were conducted using free and wall jet test setup which consists of a cylindrical settling chamber connected to high pressure storage tanks. Figure 3.2 shows a schematic sketch of the free and wall jet test setup.

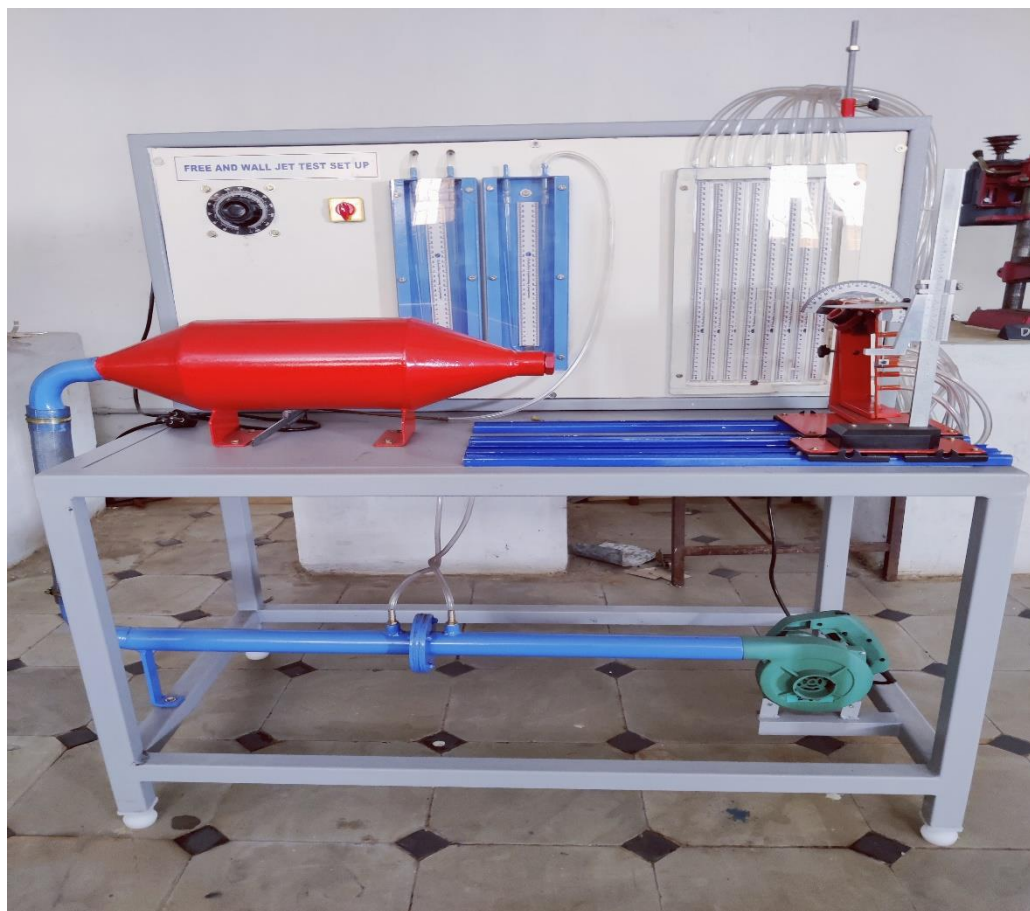


Fig. 3.2 Schematic of free and wall jet setup

These screens are inserted in the settling chamber. The settling chamber is a constant area circular section of 300 mm inside diameter and 600 mm length. The settling chamber has tapings for stagnation pressure and temperature measurements. The test models are fixed at the end of the settling chamber by a slot holder arrangement, which is a short by pipe like protrusion with embedded O-ring, over which an annular retaining sleeve with internal threads is screwed tightly. The settling chamber total pressure (P_0), which was the controlling parameter in this investigation, was maintained constant during a run by controlling the pressure regulating valve. The stagnation pressure (P_0) level in the settling chamber gives the different nozzle pressure ratios (NPR), defined as the ratio of stagnation pressure to the back pressure (P_0/P_b), required for any study. The settling chamber is the same as the ambient temperature and the back pressure is the ambient pressure into which the jets were discharged. The ambient temperature of the room was almost constant within + or - 5°C during one experimental run. The stagnation pressure was maintained with an accuracy of + or - 0.1%. During experimental runs, the settling chamber pressure was measured by a U-tube manometer. The room temperature was measured by a thermometer. The day-to-day changes in ambient pressure (Pa), were measured by a mercury barometer placed in the laboratory and averaged over the duration of experiment.

3.1.3 Traverse Mechanism

This traverse mechanism is made of cast iron having three adjustable beds which are constructed perpendicular to each other. The rotation of the screw is used to traverse the pitot probe in a particular direction. The pitot probe is also provided to rotate about one of the radial directions as well shown in Fig. 3.3

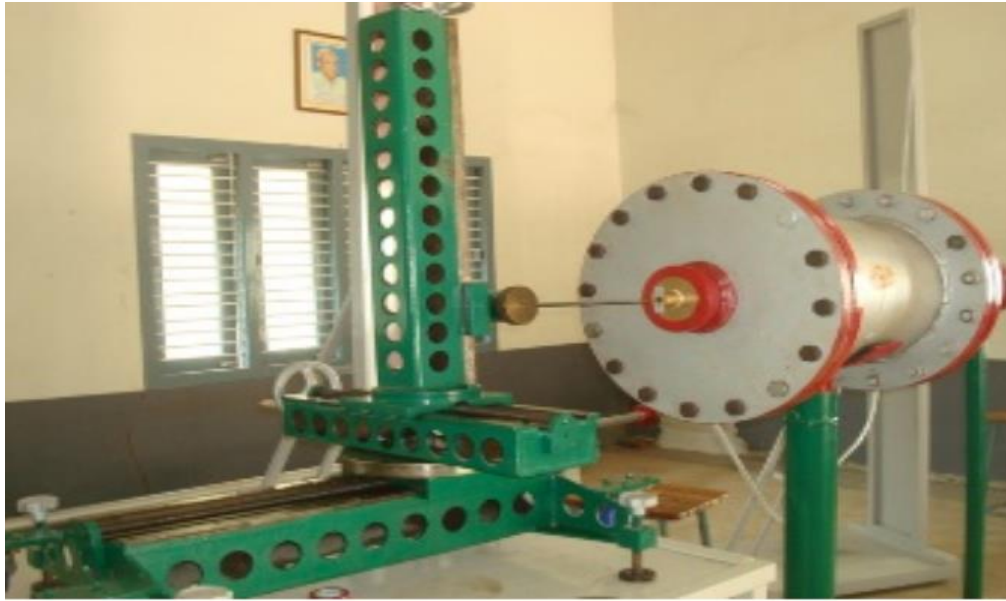


Fig. 3.4 Photographic view of traverse mechanism

3.1.4 Pitot Probe

A pitot tube is a tube that is inserted into the gas stream and has a blunt end. The tube's inside-to-outside diameter ratio should be 12 to 34, with a length that fits the gas stream's diameter of 15 to 22. The pressure opening is created by the tube's inner diameter at the blunt end. The stagnation pressure is often measured by the open ended tube faced into the river. Initially, the flowing fluid is taken to a standstill (stagnation), and some of the fluid runs into the probe, causing the pressure head gap in the U-tube manometer. Following the calculation of strain, the resulting velocity can be measured. Figure 3.4 depicts a schematic diagram of the pitot tube

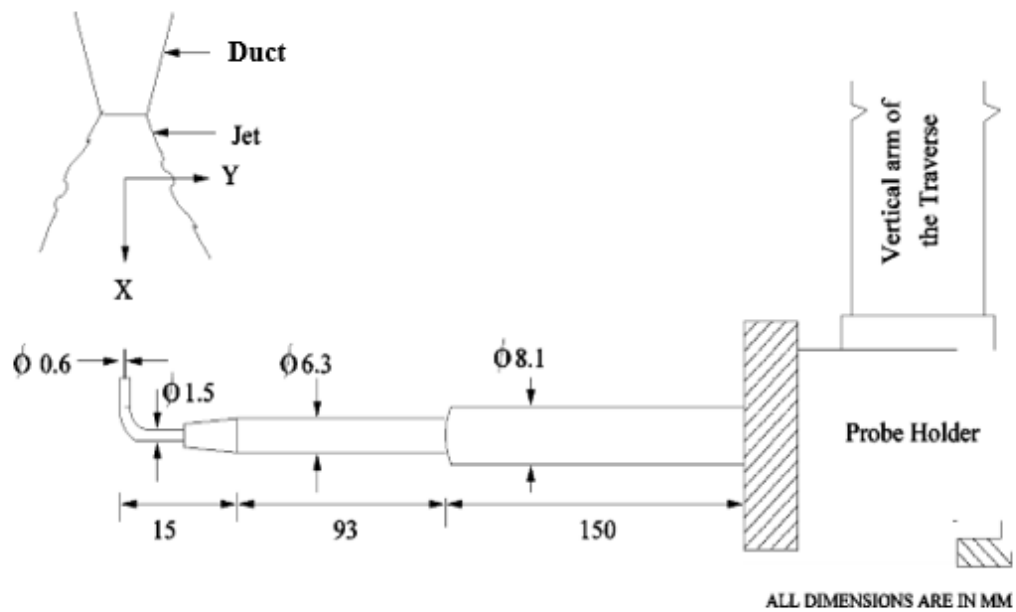


Fig. 3.5 Schematic diagrams of pitot probe

The stagnated air in the stagnation chamber is sent through the nozzle where the expansion process can be taken place and this expansion process can be done for different operating conditions by varying the Mach numbers. The pitot probe was moved for a distance of 2 mm upto the 10D of nozzle exit diameter and in order to take centerline pressure readings. The profile readings were taken at different X/D ratios for 1mm movement of pitot probe in y directions. The various axial location chosen are $X/D=0,1,2,3,4,5,6$ and 7 for the below mentioned Mach numbers. The Mach numbers chosen for the study are 0.6 and 1. The pressure measurement is done by using U-tube Manometers. U-tube manometer with Mercury as manometric fluid is used for primary jet(inner nozzle) and water as manometric fluid is used for secondary jet(co-flow nozzle).

3.2 Calculations

The head difference in U-Tube manometer for corresponding inlet Mach number can be calculated by using the following steps.

3.2.1 Calculation of head Mach number

From the gas tables for $M=0.6$,

From the isentropic relation:

Corresponding Pressure ratio is, $(P_{stag}/P_{atm})=1.2755$

We know that, $P_{guage}=P_{absolute}-P_{atmospheric}$

$$P_{guage}= P_{atmospheric} *1.2755- P_{atmospheric}$$

[$P_{atmospheric}=100862.496 \text{ N/M}^2$]

$$P_{guage}=27787.99 \text{ N/M}^2$$

We know that, $P_{guage}=pgh$

For water: $h=283.26\text{cm}$ and for Mercury= 20.82 cm

3.3 Design of Models

Introduction to Catia V5

This is defined as COMPUTER AIDED 3D INTERACTIVE APPLICATION. This is a multiplatform software for the CAD, CAM (Computer Aided Manufacturing), CAE (Computer Aided Engineering) etc. developed by French company Dassault Systems released in the year 1977. From 3D sketches, sheet metal, composites, moulded, forged, or tooling components, to the concept of mechanical assemblies, CATIA allows the construction of 3D parts. The programme includes sophisticated mechanical surfacing technology. It includes methods for completing product specification, such as practical tolerances and kinematics.. With the ICEM surfacing technology, CATIA provides a solution for shape design, styling, surfacing workflow, and visualisation to create, alter, and validate complex creative shapes from industrial design to Class-A surfacing. Despite rapid adoption of 3D modelling, 90% of production firms still use 2D sketches as their primary guide for proportions, tolerances, and annotations. The central concept of the 3D Master approach is that the 3D product model is the most appropriate vehicle for delivering the detailed product information needed by downstream processes. By converting to the 3D Master method, companies can:

In this we use part design in order to design the part. In order to open we must go through sketcher workbench. This is the set of tools that helps us to create our own design. It has feature like either surface can be created or any solid or hallow body can be created. Tis can be open by 2 methods. The

first one is by using the pull-down menu (Start-Mechanical Design- Sketcher) and the other way is by selecting sketcher icon

3.3.1 Geometry Design of S-Duct

The s-duct is designed in the following procedure. Open the part design and select the axis that has to be needed to sketch. Considering the total all length of 150 mm s duct is designed. At first make the rectangle of length 40*25 placed 25 mm above the origin. Make another rectangle of length 90mm*50 mm starting its position 60 mm away from the vertical axis. Now connect the both rectangles by a line and deleted the inner lines of the rectangle that are facing each other. Select the inclined line and right upper line of rectangle and give fillet radius of 93 mm and make it tangent to each other and now again similarly on the other side make fillet radius of 76.5 mm. similarly the process is repeated on the bottom side of the rectangle as shown in the figure 3.6.

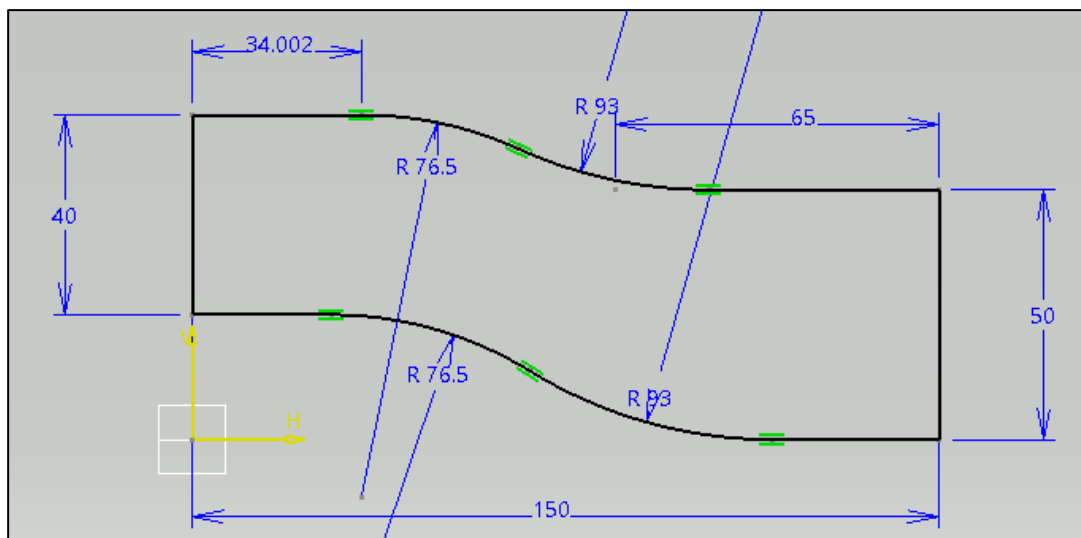


Fig 3.6 Design of S-duct

Make sure that the curve is in tangent to the lines, the lines are either horizontal or vertical and there should be no open surfaces. Now exit from the sketcher and select pad option and give pad of 30 mm and click enter. A solid body of the designed shaped is formed. Shell option is used in order to get hallow duct with thickness of 2mm. The 3D diagram is shown in the below figure 3.7

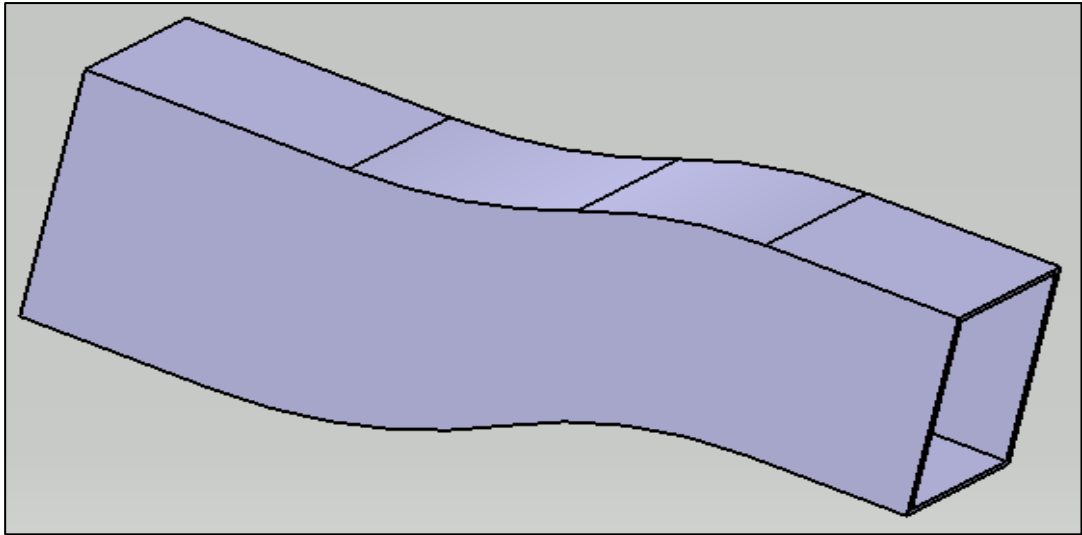


Fig. 3.7 3D view of S-duct

Now suppress the pad and shell option and go to generative shape design. Select the sketch of drawn before and select fill option so that surface is created on it. The 2D diagram is as shown in the below diagram (fig 3.8).

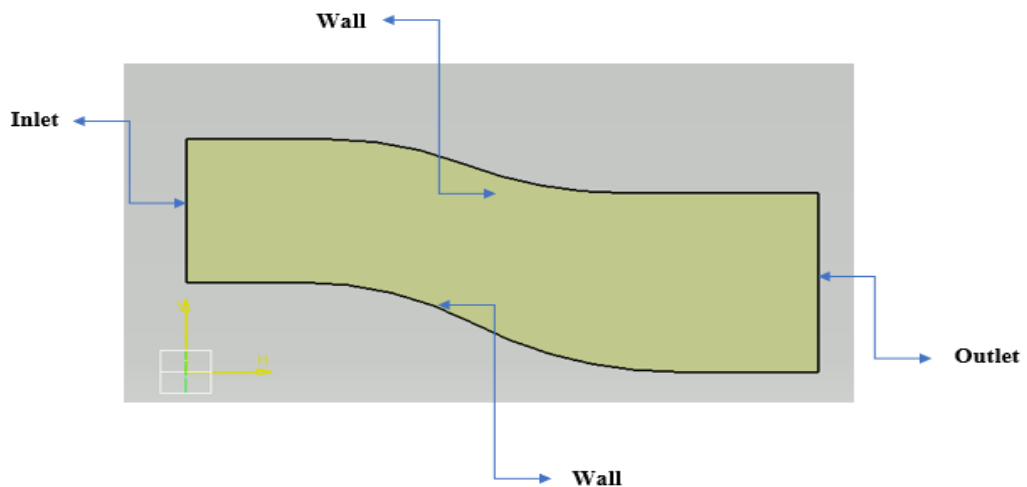


Fig. 3.8 2D view of S-duct

3.3.2 Geometry Design of Y-Duct

The s-duct is designed in the following procedure. Open the part design and select the axis that has to be needed to sketch.

Considering the total all length of 150 mm s duct is designed. Select an axis and draw a construction horizontally of 60 mm and draw an inclined line having

10° coinciding the horizontal line distance. Now draw a horizontal line of 90mm backwards and draw a vertical line having a length of 25mm from the ending of that horizontal line as shown in the below figure 5.4. Even from the angled construction line draw a vertical line of same 25mm upwards. Now connect both the open ends by a line and having a fillet radius of 95mm on right and 76.5mm on left side. Adjust the arcs and lines with tangency. Select 3-point arc and draw an arc having a radius with 98.5 and 76.9 as shown in the figure 3.9 below.

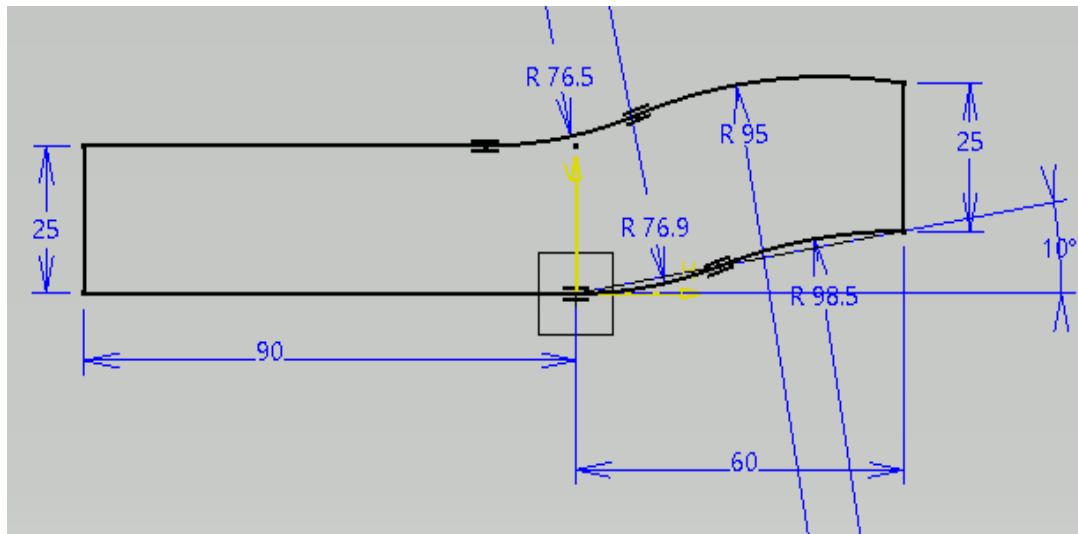


Fig. 3.9 Part design of Y-duct

Now do padding of 30mm and shell the diagram after coming out of sketcher as done same for s-duct. Select the total body and do mirror by selecting the 90mm surface as reference axis as shown in the figure 3.10 below.

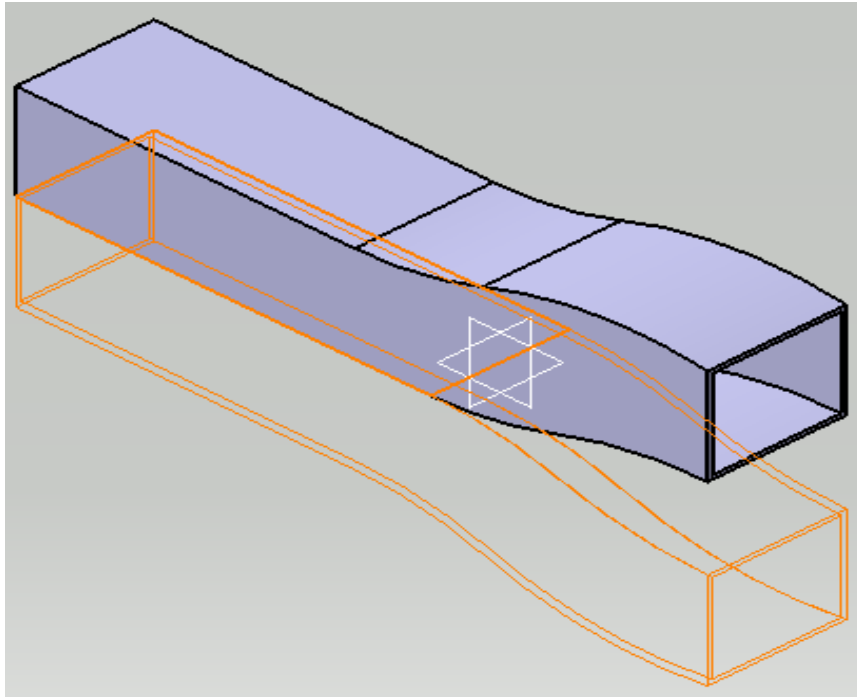


Fig. 3.10 Mirroring of Y-duct

Now go the back side of the diagram and select the face and draw a rectangle and pocket it to a distance of 90mm since the surface that is used as axis will be present and should be removed and the final 3D design of twin intake is as below figure 3.11

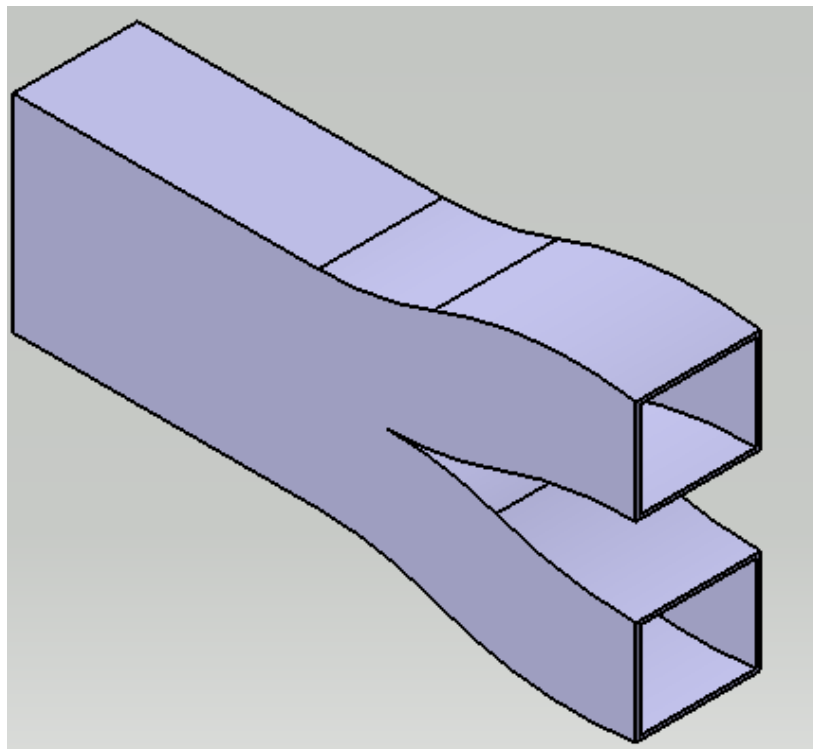


Fig. 3.11 3D of Y-duct

Suppress the design from the padding. Open the sketching and select the whole diagram. Before selection make the 90mm line into construction line and making that as the mirroring axis and select the whole diagram and mirror the top diagram and the whole design will be in the form of diagram shown below (fig 3.12).

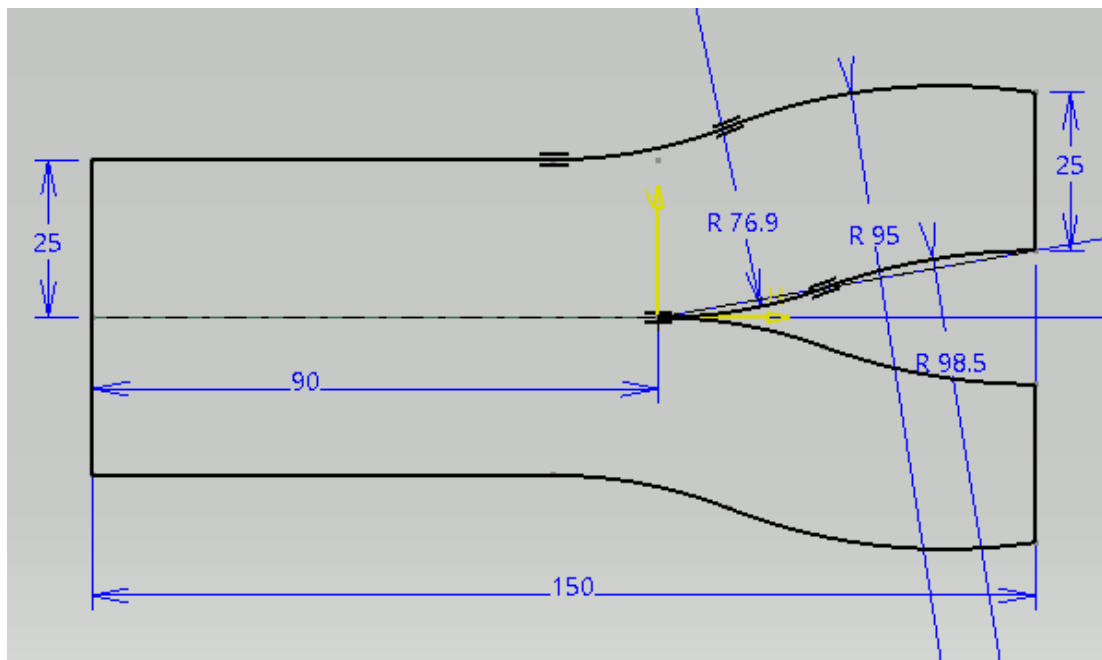


Fig. 3.12 Design of Y-duct

Now come out side of the sketcher and open the generative shape design. Select the sketch of drawn before and select fill option in order to create a surface. The 2D diagram is as shown in the below diagram (fig 3.13).

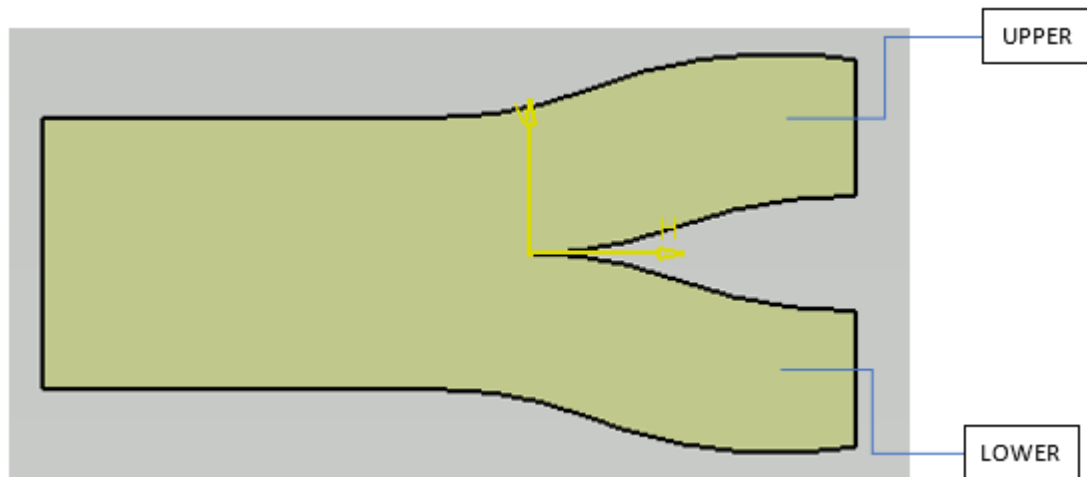


Fig. 3.13 2D of Y-duct

3.4 Fabrication of Model:

Introduction to Tools Used:

- 1) 3D Design machine
- 2) Domo Inscribe Filament

3.4.1 3D Design Machine

3D printing, or **additive manufacturing**, is the construction of a three-dimensional object from a CAD model or a digital 3D model. The term "3D printing" can refer to a variety of processes in which material is deposited, joined or solidified under computer control to create a three-dimensional object, with material being added together (such as plastics, liquids or powder grains being fused together), typically layer by layer.

Fig. 3.14 3D Design Machine

Modeling

A computer-aided design (CAD) programme, a 3D printer, or a simple digital camera and photogrammetry software can all be used to build 3D printable objects. In contrast to other approaches, 3D printed models produced with CAD have less errors. Until printing, defects in 3D printable models may be

detected and corrected. The manual modelling technique used to prepare geometric data for 3D computer graphics is close to sculpting. The method of gathering digital data on the form and appearance of a real object and making a digital image based on it is known as 3D scanning.

3.4.1 Printing

Before printing a 3D model from an STL file, it must first be examined for errors. Most CAD applications produce errors in output STL files,^{[45][46]} of the following types:

1. holes;
2. faces normal;
3. self-intersections;
4. noise shells;
5. manifold errors.

The "patch" phase in the STL generation process corrects certain flaws in the initial model. Since 3D scanning is mostly done by point-to-point acquisition/mapping, STLs created from a model acquired by 3D scanning appear to have more of these errors. Errors in 3D reconstruction are common. The STL file must then be interpreted by a "slicer," which transforms the model into a sequence of thin layers and creates a G-code file with instructions unique to a particular type of 3D printer (FDM printers). With 3D printer client programme, this G-code file can then be printed.

Layer thickness and X–Y resolution are measured in dots per inch (dpi) or micrometres (m) in printer resolution. Layer thickness is usually about 100 m (250 DPI), although some devices will print layers as thin as 16 m. (1,600 DPI). The X–Y resolution is on par with laser printers. The particles (3D dots) have a diameter of 50 to 100 m (510 to 250 DPI). Specifying a mesh resolution of 0.01–0.03 mm and a chord length of 0.016 mm for that printer resolution produces the best STL output file for a given model input file. Higher resolution settings result in bigger files with little improvement in print efficiency.

3.4.2 Finishing

Although the resolution provided by the printer is adequate for certain uses, greater precision can be accomplished by printing a slightly oversized image of the target object in regular resolution and then extracting content using a higher-resolution subtractive operation.

Both Additive Manufacturing processes have a layered structure, which causes a stair-stepping effect on component surfaces. The results are heavily influenced by the direction of a component surface during the construction process.

Chemical vapour processes based on acetone or related solvents will smooth and increase the surface finish of certain printable polymers, such as ABS.

Different components may be used in the production of parts using certain additive manufacturing techniques. These techniques can print in several colours and colour variations at the same time, and they don't involve painting.

Internal supports for overhanging features are needed for certain printing techniques during building. After the print is out, these supports must be manually withdrawn or dissolved.



3D design of a S Duct



3D Design of Y Duct

3.4.3 Materials

Owing to the ease of processing and handling polymeric materials, 3D printing has traditionally focused on polymers for printing. However, 3D printing has rapidly developed to print not only various polymers but also metals and ceramics, making it a flexible manufacturing option. Layer-by-layer fabrication of three-dimensional physical models is a new phenomenon that "stems from the ever-increasing sophistication of three-dimensional physical models. "ever-growing CAD industry, more specifically the solid modeling side of CAD. Before solid modeling was introduced in the late 1980s, three-dimensional models were created with wire frames and surfaces."

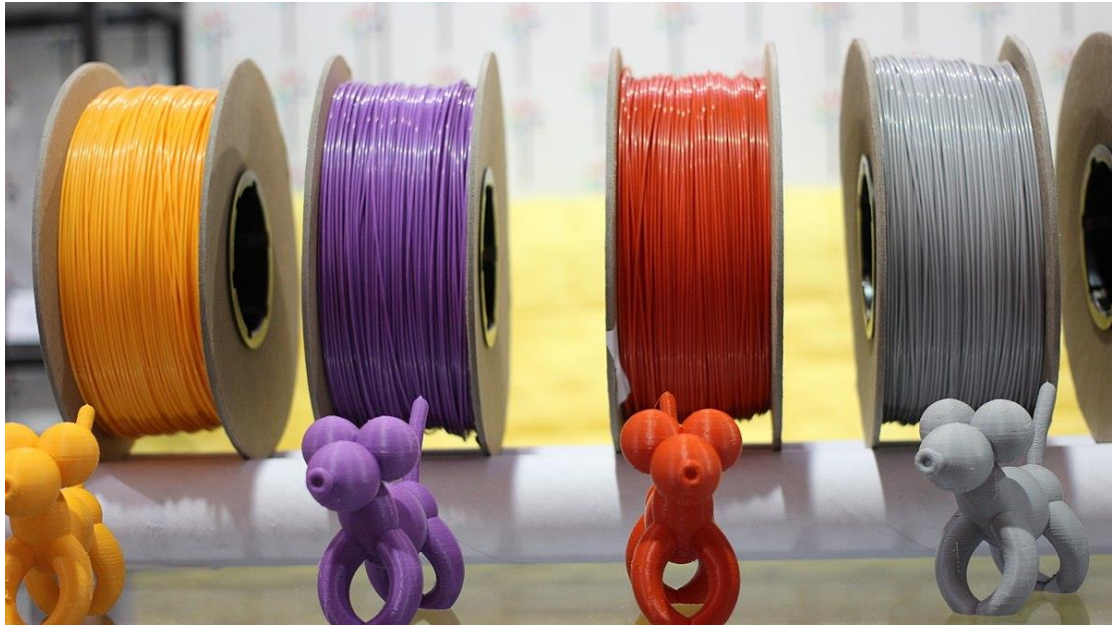


Fig. 3.16 Filament material

3.4.4 Fabrication Process and Printers:

There are many different branded additive manufacturing processes, that can be grouped into seven categories:^[72]

- Vat photopolymerization
- Material jetting
- Binder jetting
- Powder bed fusion
- Material extrusion
- Sheet lamination

Material Jetting, or particle deposition as it was originally known, was the first way of depositing three-dimensional material to form an entity. Wax inks became the first three-dimensional materials jetted, followed by CIT-jetted low-temperature alloy metal. DOD then sprayed wax and thermoplastic hot-melts. The objects were tiny and began with text characters. An object should have shape and be able to be handled. Wax characters smashed on paper papers, causing the invention of the Liquid Metal Recorder to produce metal characters for signage.

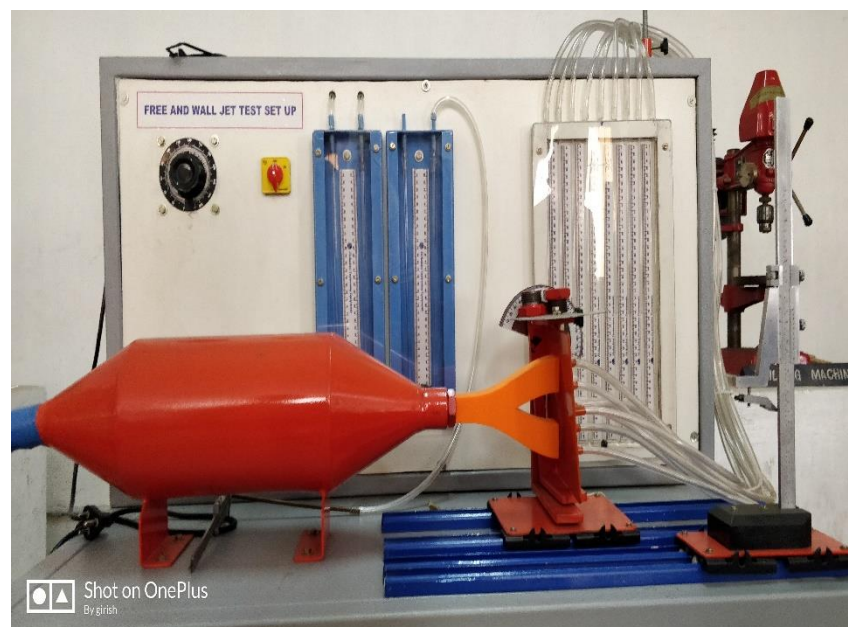
Some processes, such as stereolithography, use advanced technology to cure liquid materials.

Stereolithography uses photopolymerization to create a solid component from a liquid. Inkjet printers, such as the Objet Poly Jet, spray photopolymer materials onto a build tray in ultra-thin layers (between 16 and 30 m) before the component is finished.

3.5 Experimental Procedure

1. Fill the compressor tanks with the compressed tanks by charging them with the compressor.
2. Organize the inlets to the stagnation chamber.
3. All the ports on the surface are made attached with probes before organizing inlets to the chamber.
4. By the help of pressure regulating valve the required pressure is allowed to pass through the chamber by checking its pressure from the pressure gauge.
5. Each probe is connected to the manometer and the readings are noted
6. Graph of X/L Vs P_{abs}/P is drawn and results are compared.

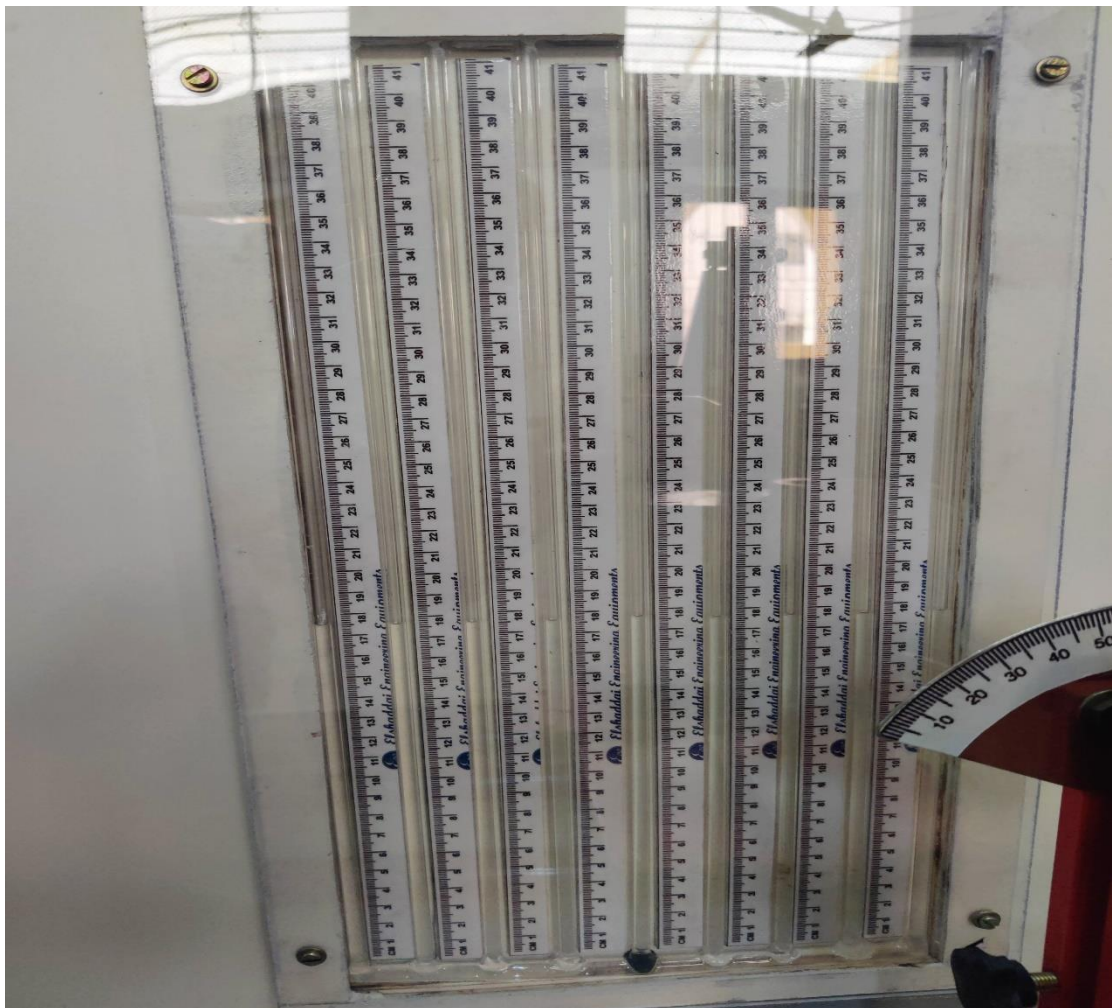
TESTING PICTURES



Testing of Y Duct



Testing of S Duct



Pressure comparison meter

COMPUTATIONAL ANALYSIS

4.1 INTRODUCTION TO ANSYS-FLUENT

ANSYS INC:

- Develops of Ansys family of products.
- Headquarters is in Canonsburg, PA-USA.
- This family of products include the following.

4.1.1 Ansys work bench application:

- Development frame work where ANSYS applications are built.
- This application is available in areas of structural, mechanics, geometry, CFD etc....

4.1.2 Ansys CFD:

- Containing CFD meshing tools with a general pre and post processing features.

4.1.3 Ansys ICEM CFD:

- State of act CFD solver includes CFX & FLUENT.

4.1.4 Ansys LS-DYNA:

- Explicit solver technology with pre/ post processor power of ANSYS software
- Used in simulation meta forging, stamping etc...
- At present we will be using ANSYS CFD tool to run our simulations.

4.2 PROCEDURE TO SOLUTION BY ANSYS:

4.2.1 Geometry:

- This is the place where we create geometry of required problem
- We can design in any other software like CATIA, SOLIDWORKS, AUTO-CAD etc.... and import them here in “igs” format.

- We generally use design modular for its user-friendly GUI.
- Here I used CATIA for designing of Y-duct and design modular for s-duct.

4.2.2 Meshing:

- The domain here is divided into several parts called elements
- Each element is joined by nodes.
- The smaller the element is the perfect the result is.
- Meshing can be either in the form of quads or triangles in 2d geometry
- Orthogonal quality, Aspect ratio, skewness are the factors that are used to check the quality of the mesh.

4.2.3 Solver Set up:

In order to start a solution for a given domain we need to give solver set up condition. Here in this solver set up everything is based on our requirement.

4.2.3.1 Flow Solvers:

- i. i. Historically, the density-based approach was used for high-speed compressible flows, while the pressure-based strategy was developed for low-speed incompressible flows. Both approaches, however, have been supplemented and redeveloped in recent years to address and work for a broader variety of flow conditions than their original purpose. The velocity field is derived from the momentum equations in both methods.
- ii. In the density-based method, the density field is calculated using the continuity equation, while the pressure field is calculated using the equation of state. The pressure field is derived in the pressure-based method by solving a pressure or pressure correction equation obtained by modifying continuity and momentum equations.
- A control-volume-based approach is used in both scenarios, which entails:
 - Dividing the area into discrete control volumes using a numerical map.

- Integration of the governing equations on the individual control volumes to construct algebraic equations for the discrete dependent variables such as velocities, pressure, temperature, and conserved scalars.
- Linearization of the discretized equations and solution of the resultant linear equation system to yield updated values of the dependent variables.
- The two numerical methods employ a similar discretization process but the approach used to linearize and solve the discretized equations is different similar discretization process but the approach used to linearize and solve the discretized equations is different.

4.2.3.1.1 Pressure-Based Solver:

The pressure-based solver belongs to the projection method, which is a broad category of methods. The restriction of mass conservation (continuity) of the velocity field is accomplished in the projection system by solving a pressure (or pressure correction) equation. The pressure equation is derived from the continuity and momentum equations in such a way that the velocity field satisfies the continuity as corrected by the pressure. The solution process entails iterations, in which the whole series of governing equations is solved repeatedly before the solution converges, since the governing equations are nonlinear and coupled to one another.

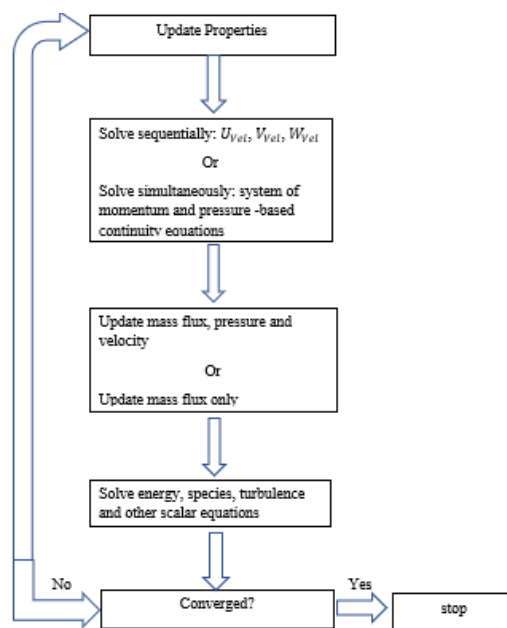


Fig 4.1: Process of pressure-based solver

4.2.3.1.2. Density-Based Solver:

The density-based solver concurrently solves the governing equations of continuity, momentum, and (if applicable) energy and species transport. The method defined in General Scalar Transport Equation: Discretization and Solution will be used to solve the governing equations for additional scalars afterward and sequentially. The non-linearity of the governing equations several iterations of the solution loop must be performed before a converged solution is obtained.

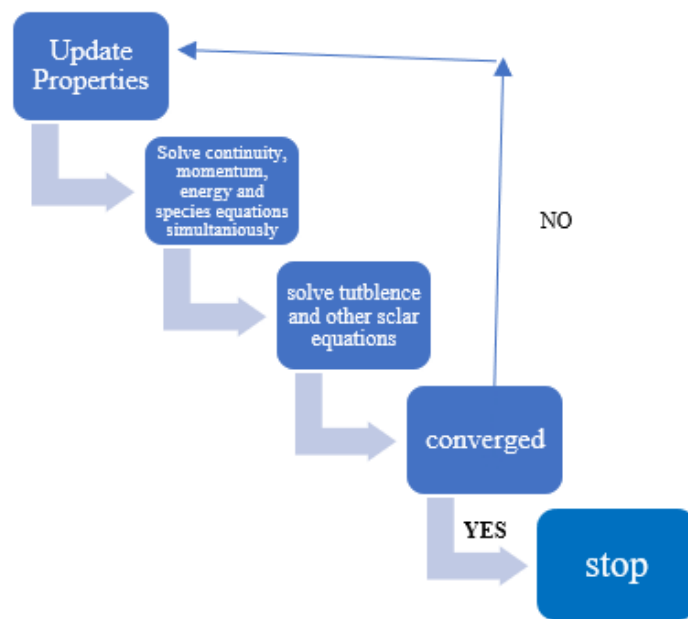


Fig 4.2: Process of density-based solver

4.2.3.2. Discretization (Interpolation Methods):

- The faces of the control volumes must be interpolated with field variables (stored at cell centres). Schemes for interpolating the convection concept include: Upwind First-Order – Easiest to converge, but only first-order precise. When Re_{cell} is 5, Power Law is more reliable than first-order for flows (typ. low Re flows) Second-Order Upwind –Uses larger stencils for 2nd order

accuracy, essential with tri/Tet mesh or when flow is not aligned with grid; convergence may be slower o Conservation Laws of Monotone Upstream-Centered Schemes (MUSCL)

- o Unstructured meshes with a locally 3rd order convection discretization scheme; more effective in forecasting secondary currents, vortices, forces, and so on.
- o Fast (Quadratic Upwind Interpolation) – Applies to quad/hex and hybrid meshes, useful for rotating/swirling flows, 3rd-order accurate on uniform mesh.

4.2.3.3 Interpolation Methods (Gradients)

- o Least-Squares Cell-Based –Default method; has the same accuracy and properties as Node-based Gradients and is less computationally intensive.

4.2.3.4 Pressure-Velocity Coupling:

When using the pressure-based solver, pressure-velocity coupling refers to the computational algorithm that uses a combination of continuity and momentum equations to obtain an equation for pressure (or pressure correction).

In FLUENT, there are five algorithms to choose from. Pressure-Linked Equations: A Semi-Implicit Method (SIMPLE) The default scheme is reliable.

SIMPLE AND CONSISTENT (SIMPLEC)

- o Facilitates the integration of simple problems (e.g., laminar flows with no physical models employed).
- Pressure-Implicit with Operator Splitting (PISO)
- o Useful for meshes with higher-than-average skewness cells or for unsteady flow problems.

- Fractional Step Method (FSM) for unsteady flows.
 - Used with the NITA scheme; similar characteristics as PISO.
- Coupled (this is how the pressure-based coupled solver, described previously, is enabled)

4.2.3.5 Hybrid Initialization:

•Hybrid Initialization is a new function in R13, and it can be picked from the Initialization menu in the GUI.

•

A series of methods is used to provide a simple approximation of the flow area.

•To evaluate the velocity and pressure fields, it solves Laplace's equation.

4.2.3.6 Convergence Difficulties:

•Sometimes running for further iterations is not the answer:

–Either the solution is diverging (aka “blowing up”) –Or the residuals are ‘stuck’ with a large imbalance still remaining.

Troubleshoot:

Convergence of both equations is affected by continuity equation convergence problems.

–Make sure the issue is clearly stated.

–

Using a first-order discretization scheme, compute an initial solution.

–Change the Courant or under-relaxation numbers.

–Make sure the mesh is of high consistency. It only takes one warped grid cell to keep the whole solution from converging [This is why you can ALWAYS verify the mesh quality].

4.2.3.4 Modifying the Courant Number:

When using coupled solvers, the Courant number is the most critical control for stability.

- Even for steady-state problems, the density-based solver has a transient concept.

The size of the time phase is determined by the Courant number.

- For a density-based explicit solver, use the following formula:

–The Courant number has an upper limit imposed by stability restrictions.

- It can't be more than two (default value is 1).

- In the case of density-base

4.2.4 Solution:

The place where all the commands given above are taken into consideration and perform the calculation till the Y^+ values becomes equal to the just above value. This condition is called “Convergence”. After the solution reaches convergence condition or the change in Y^+ is almost negligible or the residuals in the panel are showing the graph of almost straight line then we can stop the solution and go to result.

4.2.5 Result:

The place where all the data is extracted for post processing the necessary parameters.

4.3 Computational Procedure

4.3.1 Geometry:

Here I used CATIA for designing of Y-duct and design modular for s-duct.

4.3.2 Meshing:

The following conditions are used for meshing the models:

Unit System	Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

Table 4.1 Units of Mesh for FFF of Y-duct & S-duct

The number of nodes and elements of each shape for this present case a is give below in table 4.3 and table 4.4 as follows.

Domain	Nodes	Elements
solid fill	413798	205616

Table 4.2 Mesh Information for FFF of Y-Duct

Domain	Nodes	Elements
solid fill	172368	171385

Table 4.3 Mesh Information for FFF of S-Duct

Sizing	Y-duct	S-duct
Size Function	Curvature	Curvature
Relevance Centre	Fine	Fine
Span Angle Centre	Fine	Fine
Curvature Normal Angle	Default (18.0 °)	Default (18.0 °)
Min Size	5.e-002 mm	0.10 mm
Max Face Size	0.20 mm	0.20 mm
Growth Rate	1.20	1.20

Defeature Size	Default (2.5e-002 mm)	Default (5e-002 mm)
----------------	-----------------------	---------------------

Table 4.4 Sizing Information for FFF of Y-duct & S-duct

4.3.3 Solver Set Up:

The following conditions are used for meshing the models:

Type	Y-duct	S-duct
Solver	Density-based	Density-based
Time	Steady	Steady
Energy	On	On
Viscous	Realizable k-e, Standard Wall function	Realizable k-e, Standard Wall function
Boundary conditions		
Inlet	Pressure inlet	Pressure inlet
Outlet	Pressure outlet	Pressure outlet
Wall	Wall	Wall

Table 4.5 Setup conditions and Boundary conditions of Y-duct & S-duct By using the above solving conditions the models are simulated and the results are extracted after reaching the convergence criteria.

Chapter 5

Results and Discussion

5.1 Flow Analysis through Y-Duct

5.1.1 Variation Pressure and Temperature along the Duct Centerline

The centerline pressure distribution (CPD) of duct is constantly increasing along the direction of duct till the junction of the ducts and decreases till it reaches the atmosphere. Comparing the results of different Mach numbers i.e., 0.5,0.6,0.7 it is observed that as we increase the Mach number, the pressure rapidly increases at the junction of the upper and lower surfaces of the duct and they do follow the trend along the duct. As the Mach number increases the pressure also increases and the junction area is mostly affected by high pressure present there. The results for the upper and lower duct are similar as the design of duct is symmetrical with respect to x-axis. This can be seen in the below Fig. 5.1.

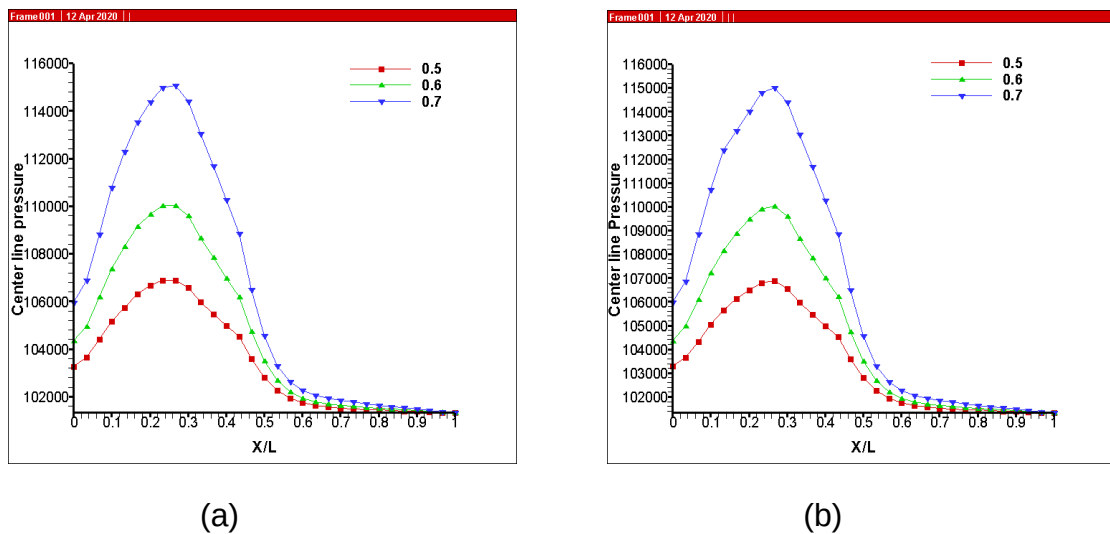


Fig. 5.1: Centerline pressure variation through Y-duct for different Mach numbers (a) upper surface (b) lower surface

Along the centerline of the duct the temperature increases gradually then decreases and then increases suddenly at junction and remains constant. Mach number is inversely proportional to temperature, as Mach number increases the temperature decreases, this can be shown by figuring at Fig. 5.2. The different Mach number used to validate are 0.5,0.6,0.7. The temperature variation is

similar when compared to upper and lower surfaces as they are symmetric about x axis.

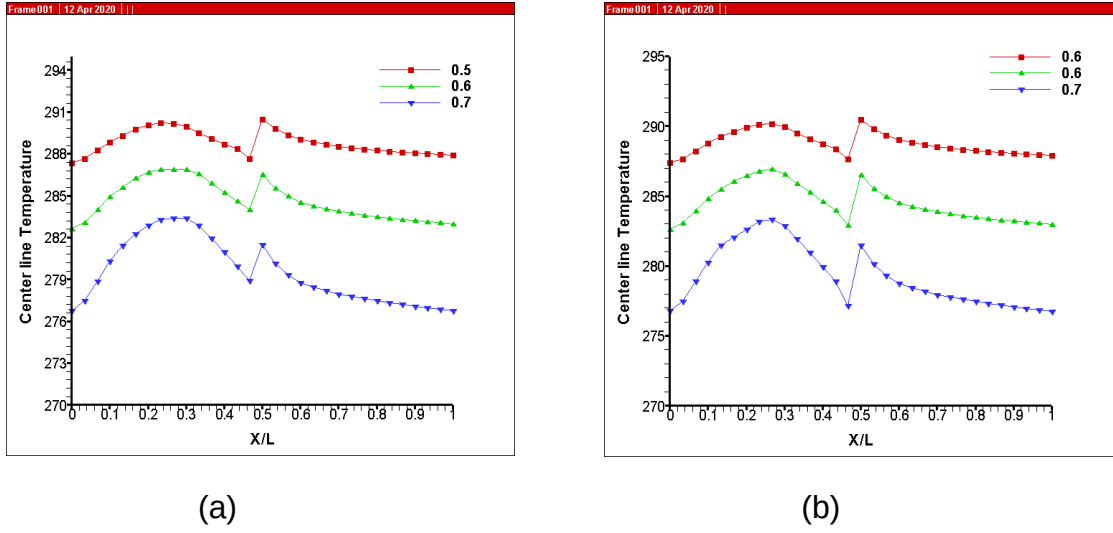


Fig. 5.2: Centerline temperature variation through Y-duct for different Mach numbers (a) upper surface (b) lower surface

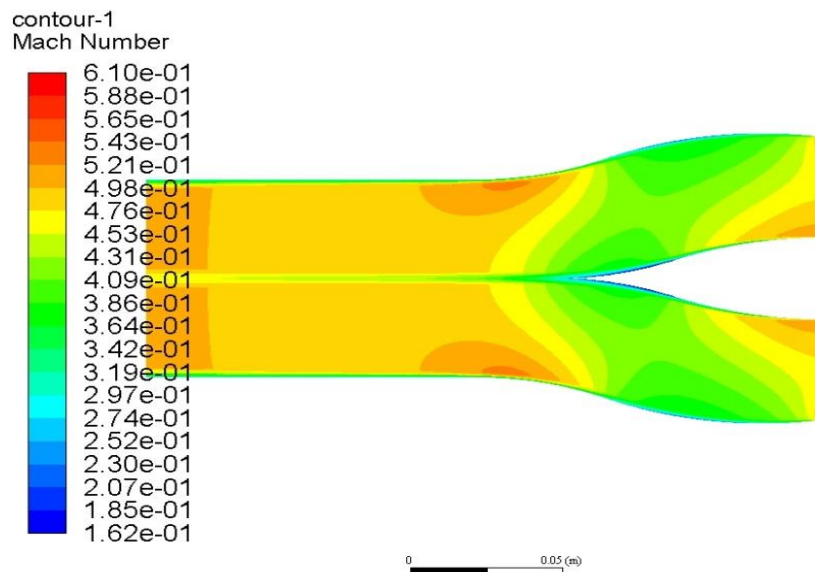
5.1.2 Variation Mach number along the Duct Centerline

Initially corresponding pressure is allowed to pass through the duct with respect to the Mach number. When we consider $M = 0.5$, the pressure inlet is $P = 120196$, the corner areas and the curved path on the wall is having high Mach number of $0.54 M$ and above, the wall region of the junction is subjected to have the least Mach number which is around $0.16M$ and gradually increasing itself along the duct's direction. The region around the junction is having the Mach number increase along its length. Over all the length of the duct the wall region is having the Mach number of constants of around $0.3 M$ from the Fig. 5.3 (a).

When we consider $M = 0.6$, the pressure inlet is $P = 129242$, only the sharp edges and the curve regions is having the highest Mach number of $0.7M$, the wall region of the junction and the divergent regions of the upper and lower surfaces are subjected to have the least mach number which is around $0.19M$ and gradually increasing itself along the duct's direction. Over all the length of the duct the wall region is having the Mach number of constants of around $0.4 M$ from the fig. 5.3 (b). Mach Number is having the almost constant till there is no junction. After the junction region it started to increase gradually and maintained constant throughout the duct with Mach number of 0.58 .

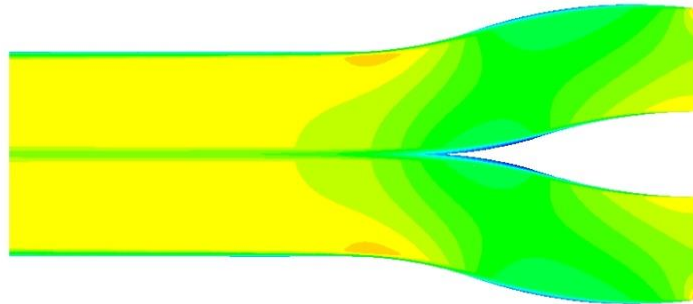
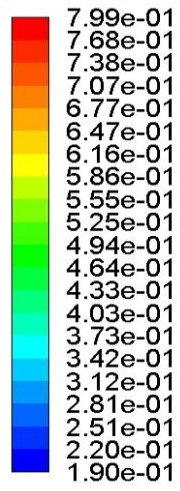
When we consider $M = 0.7$, the pressure inlet is $P = 140554$, only the sharp edges and the curve regions is having the highest Mach number of $0.8M$, the wall region of the junction and the divergent regions of the upper and lower surfaces are subjected to have the least Mach number which is around $0.21M$ and gradually increasing itself along the duct's direction. The zone of influence of the junction is reduced when compared to the $0.6M$ contour as shown in Fig. 5.3 (c). Over all the length of the duct the wall region is having the Mach number of constants of around $0.53 M$ from the (c). Mach Number is having the almost constant till there is no junction. After the junction region it started to increase gradually and maintained constant throughout the duct with Mach number of 0.67 .

When comparing the contours, the zone of influence of the junction is gradually decreasing as the Mach number is increasing constantly. The Mach number is reducing gradually for the increase in the inlet pressure or inlet Mach number for the centerline of the duct after the junction



(a) $M = 0.5$

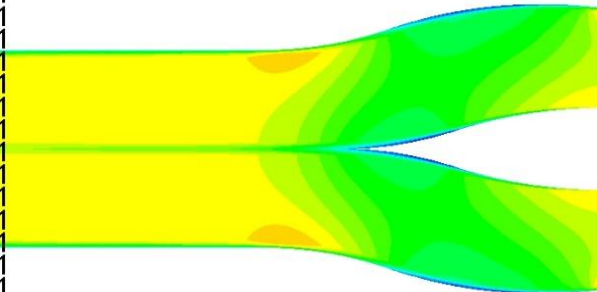
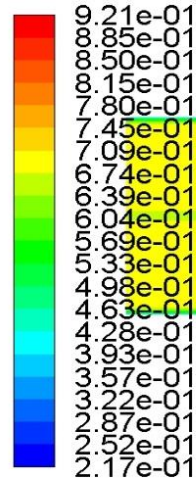
contour-1
Mach Number



0 0.05 (m)

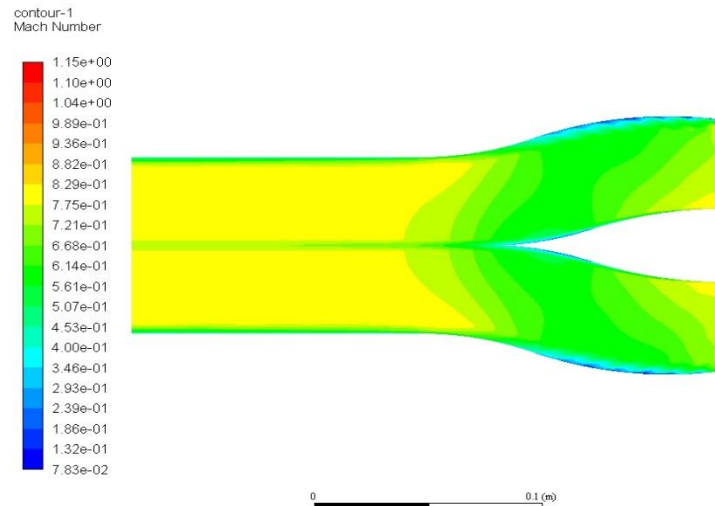
(b) $M = 0.6$

contour-1
Mach Number



0 0.1 (m)

(d) $M = 0.7$



(d) $M = 0.8$

Fig. 5.3 Pressure variation through S-duct for different Mach numbers

5.2 Flow Analysis through S-Duct:

5.2.1 Variation of Pressure and Temperature along Wall and Centerline of Duct

5.2.1.1 Variation Pressure along Wall and Centerline of Duct

As the Mach number increases the pressure gradually decreases and suddenly falls at the curvy nature of the duct and increases gradually and maintains constant along the centerline direction of duct. Same happens in the case of the wall but due to some of the frictional effects it shows fluctuations. Pressure increases at the inlet as the Mach number increases and at the bending as Mach number increases the pressure decreases. The variation of the pressure for Mach number 0.5, 0.6, .7, 0.8 are shown in Fig 5.4 below.

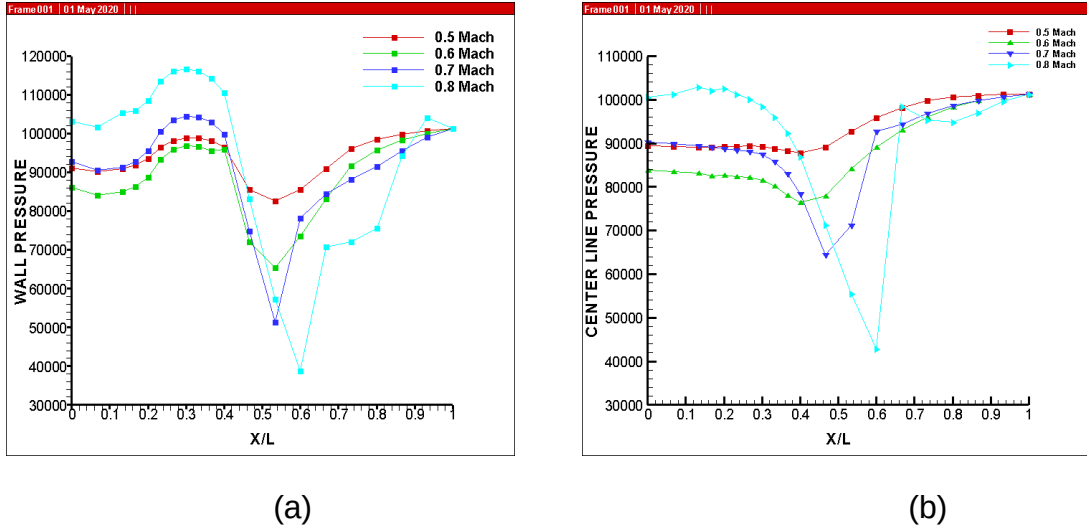


Fig. 5.4: Pressure variation through S-duct for different Mach numbers (a) wall pressure (b) centerline pressure

5.2.1.2 Variation of Temperature along Wall and Centerline of Duct

Along the centerline of the duct the temperature increases gradually then decreases and then increases suddenly at junction and remains constant. Mach number is inversely proportional to temperature, as mach number increases the temperature decreases, this can be shown by figuring at Fig. 5.5. The different mach number used to validate are 0.5,0.6,0.7. The temperature variation is similar when compared to upper and lower surfaces as they are symmetric about x axis.

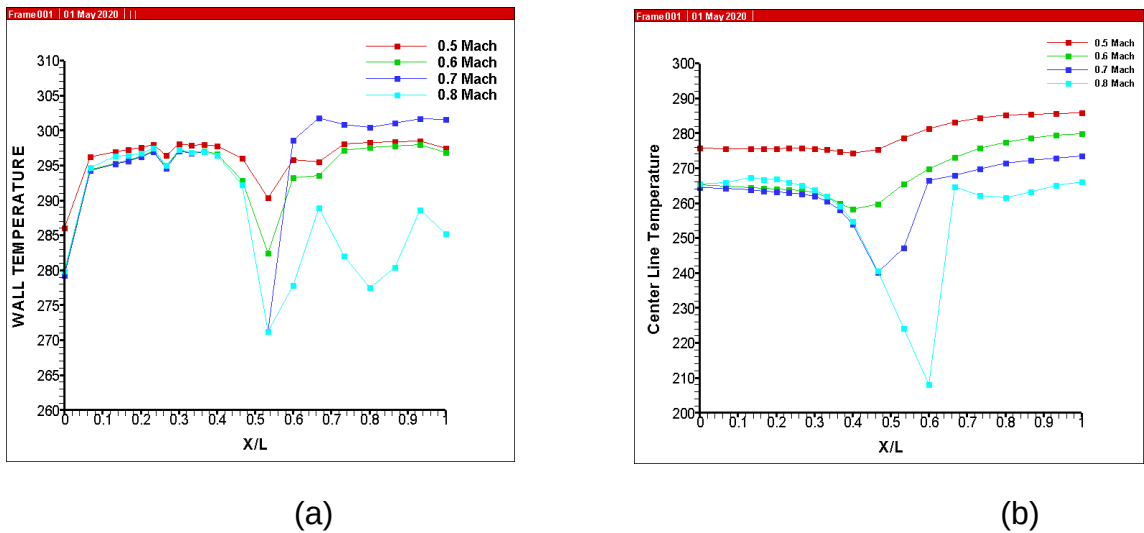


Fig. 5.5: Temperature variation through S-duct for different Mach numbers (a) wall temperature (b) centerline temperature

5.2.2 Variation Mach number along the duct

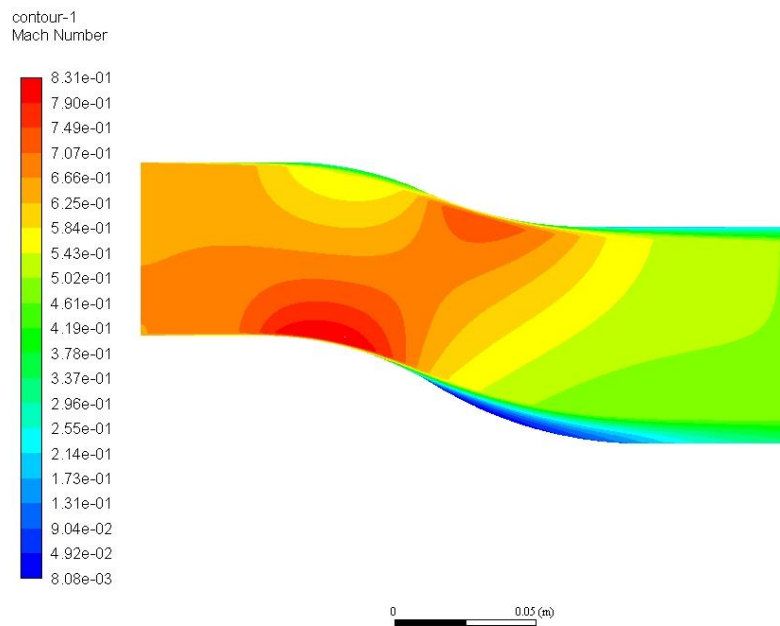
Corresponding to the Mach number required then inlet pressure is given. When we consider $M = 0.5$, the pressure inlet is $P = 120196$, the Mach number is not even throughout the duct having higher at the curvy and lower at the wall of the divergent portion as shown in the Fig. 5.6(a). As long as, we go around the wall the convergent portion is having the Mach number of around 0.79 and above and divergent portion having Mach number of 0.008. The Mach number is decreasing from inlet to outlet gradually.

When we consider $M = 0.6$, the pressure inlet is $P = 129242$, it has the highest Mach number of 1.08 and lower Mach number of 0.0008. When compared to the 0.5 Mach number the divergent portion is having lower Mach number and the flow is having proper developing zone. We observe them at the convergent and divergent portion of the duct. the flow gets gradually changing from the curved edges and combine in the flow having inlet Mach number of 0.76 and gradually increasing to a Mach number of 0.85 and decreases continuously until it reaches the constant flow Mach number of 0.51 Mach number as shown in the Fig. 5.6(b).

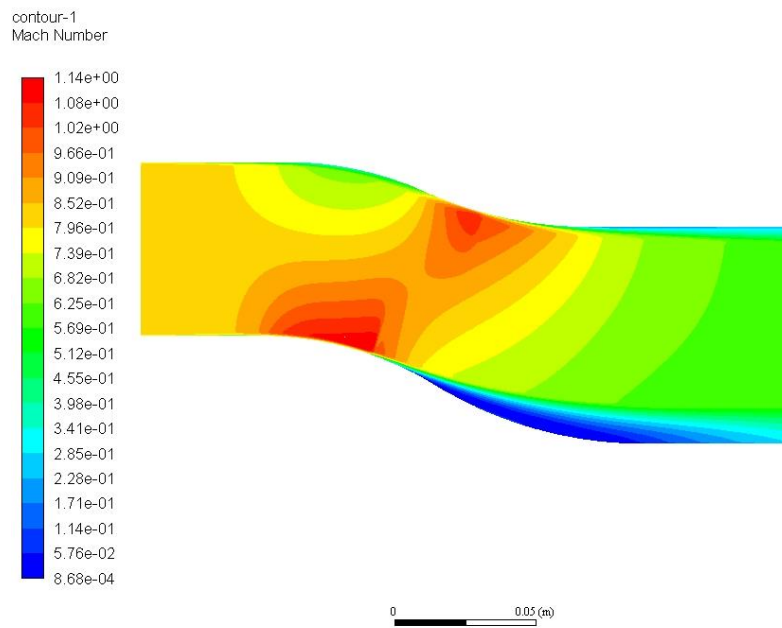
When we consider $M = 0.7$, the pressure inlet is $P = 140554$, the flow is so even in the duct such that it maintains around a Mach number of 0.7, the flow is gradually increases when we reach the curved region and suddenly changes from Mach 1.7 to 0.7 which explains that there is formation of shock in the duct, and the wall region before shock formation is having Mach number around 0.77 and after the shock region it is having a Mach number of 0.008 as shown in the Fig. 5.6(c).

When we consider $M = 0.8$, the pressure inlet is $P = 154459$, the flow is so even in the duct such that it maintains around a Mach number of 0.7 though at some of the portions there is some differences, the flow is gradually increases when we reach the curved region and suddenly changes from Mach 1.7 to 0.6 which explains that there is formation of shock in the duct, and the wall region before shock formation is having Mach number around 0.77 and after the shock region it is having a Mach number of 0.008 as shown in the Fig. 5.6(d). as there is no much change when compared to 0.7 Mach number.

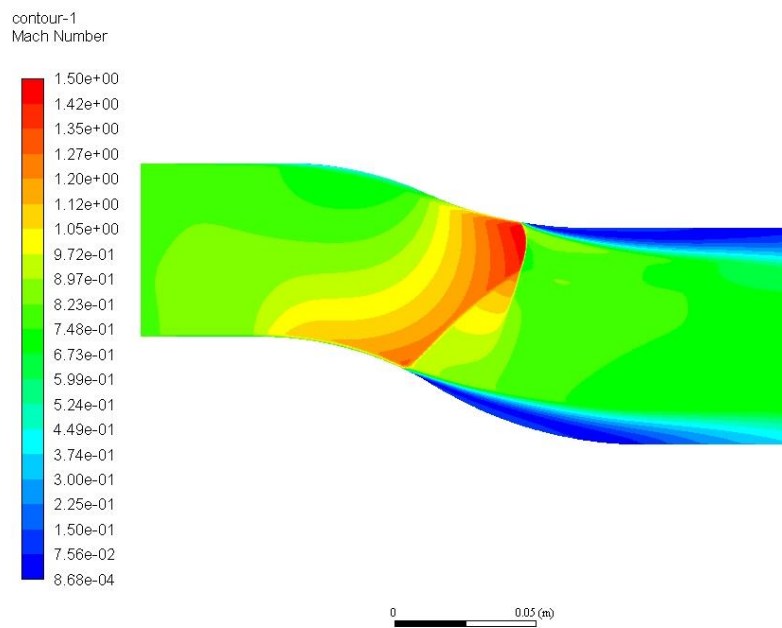
When comparing the contours, the zone of influence of the curved area is gradually decreasing as the Mach number is increasing constantly, and there is formation of shock at the convergent area. It is that till the 0.6m there is only change in Mach number but in the case of 0.7,0.8 Mach number there we can see a sudden jump in the Mach number where the variations of properties is having with slight variation then we can conclude that there is a formation of weak shock.



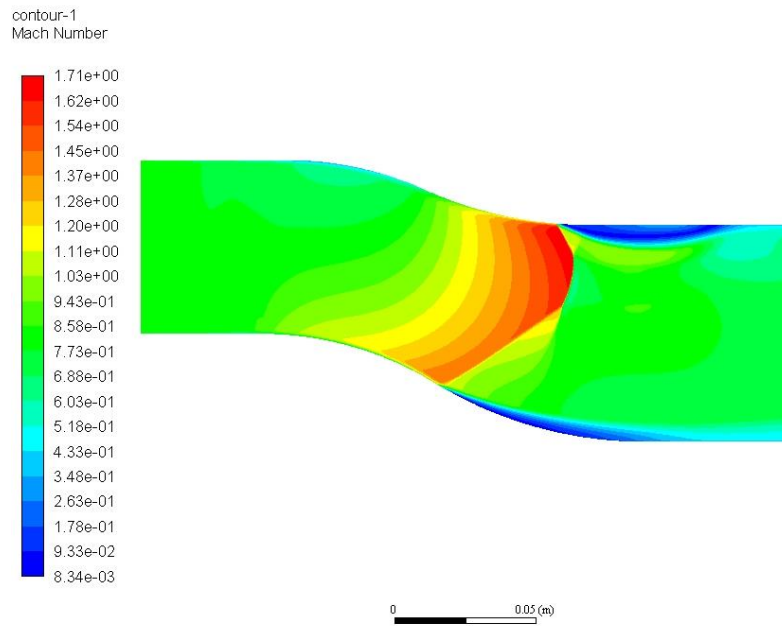
(a) $M = 0.5$



(b) $M = 0.6$



(c) $M = 0.7$



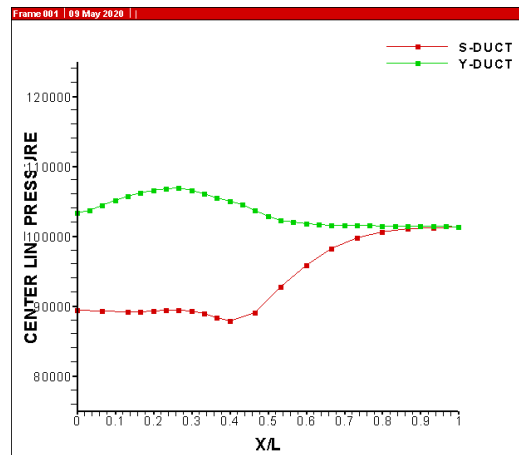
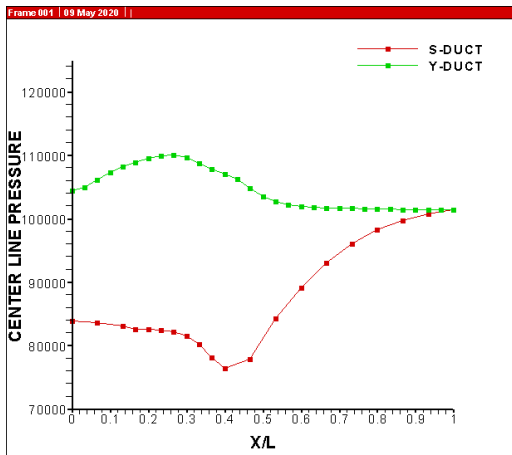
(d) $M = 0.8$

Fig. 5.6 Mach number variation through the Y-Duct at different Mach numbers

5.3 Centerline Comparison of Ducts for different Mach Numbers

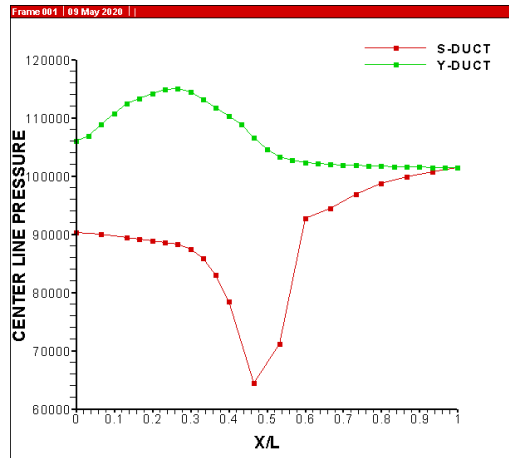
5.3.1 Centerline Comparison of Pressure

On increasing the Mach number, the pressure gets increased in the y duct but in the case of s duct the pressure decreases imply the variation between the ducts increases (Fig. 5.7) which explains that pressure ratio is higher for the y duct when compared to s duct at same operating condition.



(a)

(b)

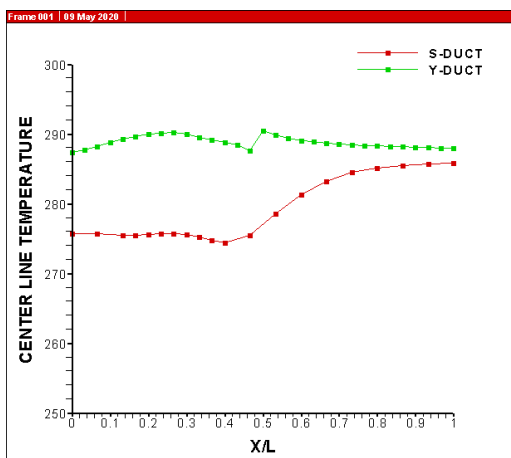


(c)

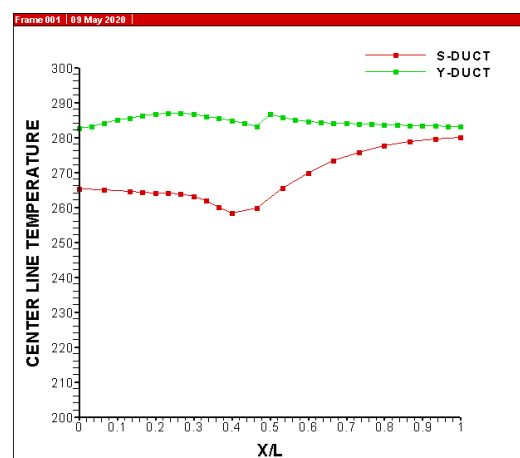
Fig. 5.7: Comparison of Centerline Pressure variation through S-duct and Y-duct for different Mach numbers (a) 0.5 Mach numbers (b) 0.6 Mach numbers (c) 0.7 Mach numbers

5.3.2 Centerline Comparison of Temperature

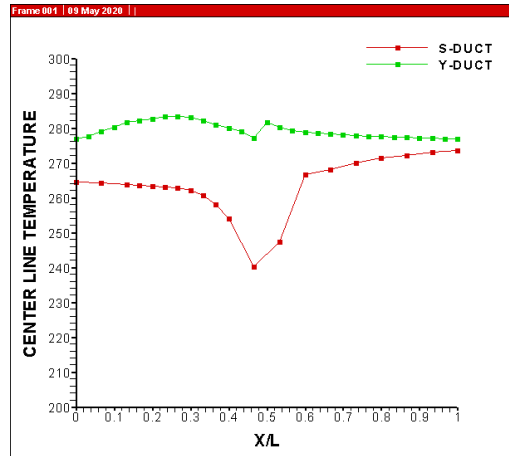
As we increase the Mach number there is temperature variation which increases with respect to Mach number. The difference in temperature is decreasing at inlet and increases by the convergent portion on the duct as shown in the Fig. 5.8. For y duct there will be higher temperature when compared to s duct.



(a)



(b)



(c)

Fig. 5.8: Comparison of Centerline temperature variation through S-duct and Y-duct for different Mach numbers (a) 0.5 Mach numbers (b) 0.6 Mach numbers (c) 0.7 Mach numbers

Chapter 6

6. Summary and Conclusion

6.1 Conclusion:

Detailed flow distribution has been made with two different ducts having common inlet and exit length designed as intakes. By investigation about the flow and by making the plots of obtained result, we can conclude that the magnitude of static pressure in dual passage y duct is more when compared to that of the single passage s duct. The main cause could be increase in streamwise pressure gradient immediately downstream of the first bend.

6.2 Scope of Future Work:

Using the open jet facility and secondary flow or suction pump from the laboratory we can add vortex generator slots, suction ports and even combination of them in order to find the static pressure variation and skin frictional effects in the ducts and compare them. Asymmetric y-duct flow can be studied. We can also investigate the flow by using different models having different areas of inlet to exit ratios and shapes.

References

1. 1. S. R. Wellborn, B. A. Reichert, and T. H. Okiishi, "An Experimental Study of the Flow in a Diffusing S-Duct," AIAA-92-3622, 28th Joint Propulsion Conference and Show, Nashville, Tennessee, 6-8 July 1992.
2. 1. "Naiver-Stokes analysis and experimental data comparison of compressible flow in a diffusing s-duct," 13th international conference on numerical methods of fluid mechanics, Harloff, g. j., Reichert, b. a., and Wellborn, s. r. aiaa-92-3622, Rome, Italy, 6-10 July 1992.
3. C. Fiola, (2013). Separated and Secondary Flows in Diffusing S-Ducts for Air Breathing Propulsion (M) Numerical Simulation of Separated and Secondary Flows in Diffusing S-Ducts for Air Breathing Propulsion (M)AkshoyRanjanPaul,¹ ShreyJoshi,² AmanJindal,² ShivamP.Maurya,³ andAnujJain¹, "Experimental Studies of Active and Passive Flow Control Techniques Applied in a Twin Air-Intake". Hindawi Publishing Corporation the Scientific World Journal Volume2013, ArticleID523759.
4. B.J Prior And C.N Hall investigated on "Subsonic Wind Tunnel Test of Various Forms of Air intakes Installed in Fighter Type Aircraft."
5. J. Seddon and W.J.G Treble made a study on "Model Tests on The Asymmetry of Airflow Occurring in Twin Intake Systems at Subsonic Speeds."
6. 1. R.K. Sullery, Sourabh Mishra, and A.M. Pradeep, "Application of Boundary Layer Fences and Vortex Generators in Improving Performance Of S-Duct Diffuser," Journal of Fluids Engineering, Vol. 124, No. 1, MARCH 2002.
7. Kristina J. Cook is number one. "Mixing Evaluation of a Passive Scaled-Up Serpentine Micromixer with Slanted Grooves," YanFeng Fan Ibrahim Hassan¹, Journal of Fluids Engineering AUGUST 2013, Vol. 135 / 081102