Global Journal of Researches in Engineering: D
Chemical Engineering
Volume 14 Issue 1 Version 1.0 Year 2014
Type: Double Blind Peer Reviewed International Research Journal
Publisher: Global Journals Inc. (USA)
Online ISSN: 2249-4596 & Print ISSN: 0975-5861

Design and Analysis of Transonic Wind Tunnel

By T. Kumaraswamy, V. V. S. Nikhil Bharadwaj & Parthasarathy Garre
MLRIT, India

Abstract- The Transonic Wind Tunnel is used to test aircraft models at speeds from Mach number 0.2 to 1.4. Transonic flows consist of mixed subsonic and supersonic flow regions. Shocks can occur in these flows but often do not have a strong enough pressure gradient to assume flow properties similar to those of supersonic flows. These regions are difficult to model mathematically because they have characteristics of subsonic and supersonic flows. In this regard, the paper is aiming towards the design and analysis of the transonic wind tunnel performance by considering two phases namely automated design and its evaluation. Modern optimization software is combined with isentropic relations; simulations are analyzed to design a Mach 1.2 nozzle with maximum test length. The optimal design has an unconventional shape described as compound curvature, which makes the contour appear slightly wavy in AutoCAD. The same is evaluated and found satisfactory for the proposed modification of the test section in the wind tunnel in fluent analysis.

Keywords: transonic, mach number, AutoCad, fluent, hypermesh, analysis.

GJRE-D Classification : FOR Code: 090199

DESIGNANDANALYSISOFTRANSONICWINDTUNNEL

Strictly as per the compliance and regulations of :

Design and Analysis of Transonic Wind Tunnel

T. Kumaraswamy [α], V. V. S. Nikhil Bharadwaj [σ] & Parthasarathy Garre [ρ]

Abstract- The Transonic Wind Tunnel is used to test aircraft models at speeds from Mach number 0.2 to 1.4. Transonic flows consist of mixed subsonic and supersonic flow regions. Shocks can occur in these flows but often do not have a strong enough pressure gradient to assume flow properties similar to those of supersonic flows. These regions are difficult to model mathematically because they have characteristics of subsonic and supersonic flows. In this regard, the paper is aiming towards the design and analysis of the transonic wind tunnel performance by considering two phases namely automated design and its evaluation. Modern optimization software is combined with isentropic relations; simulations are analyzed to design a Mach 1.2 nozzle with maximum test length. The optimal design has an unconventional shape described as compound curvature, which makes the contour appear slightly wavy in AutoCAD. The same is evaluated and found satisfactory for the proposed modification of the test section in the wind tunnel in fluent analysis.

Keywords: transonic, mach number, AutoCad, fluent, hypermesh, analysis.

I. Introduction

Test facilities such as wind tunnels require stringent control of test parameters such as wind speed, temperature, pressure, etc., in order to satisfy the objectives of the simulation [1]. Gases consist of molecules moving in random motion with negligible cohesive forces. Depending on the speed of the gas, it can be approximated as either an incompressible or compressible substance. An incompressible substance is one that does not experience significant density changes throughout the substance, or the medium. This situation can be found in low speed flows, where the kinetic energy is negligible to the thermal energy found in the medium. Higher speed flows, where the kinetic energy is comparable in magnitude to the thermal energy, will have varying densities throughout the medium and are thus considered compressible substances [2]. When a body is placed in a gas flow, disturbances will form and propagate through the medium. A given disturbance will affect those molecules closest to the body and will collide with the surrounding molecules. In incompressible flows, the distance between the molecules will be maintained through propagation of the disturbance, but in compressible flows, the molecules will contract closer to each other before returning to their original distance after the disturbance [3]. The speed at which a disturbance propagates through a medium is the speed of sound. The non-dimensional number that expresses free-stream velocity with respect to the speed of sound of a medium is the Mach number (*M*). Flow properties can be established by the value of the Mach number. Flows that have a low Mach number, $M < 0.3$, can be assumed incompressible flow, while anything above this is considered compressible flow. The compressible flow regime can be further broken down into subsonic flow, $M < 1$, transonic flow, $0.7 < M < 1.2$, and supersonic flow, $M > 1$ [4]. Transonic flow is a special case of subsonic flow. The transonic flow field is characterized by regions of mixed flow where there may have been a shock incident on a surface yielding supersonic flow upstream but subsonic flow immediately behind it. In most situations, shocks can be considered negligibly thin compared to any other length scale in the flow (thicknesses on the order of 10^{-5} cm are typical). In addition, despite the fact that the Mach number lies between 0.8 and 1, the analytical solution of the conservation equations is much more difficult since neither the elliptic equations used to solve problems in the subsonic regime nor the hyperbolic equations that govern the supersonic flow regime are strictly applicable for the transonic flow regime. There are many parameters that characterize the TWT such as test section dimensions, operating characteristics (Reynolds number, Mach number), general capabilities of the facility (Mach number range and maximum Stagnation pressure) [5]. A maximum Mach number of 1.8 is possible in the transonic test section [6].

Author α: Assistant Prof, Aeronautical Department, MLRIT, Hyderabad, AP, India. e-mail: gps.mlrit@gmail.com
Author σ: B. Tech Student, Aeronautical Department, MLRIT, Hyderabad, AP, India.
Author ρ: Associate Professor, Aeronautical Department, MLRIT, Hyderabad, AP, India.

II. Dessign of Transonic Wind Tunnel

Computations generally begin at the throat using a transonic scheme to compute the flow near the throat, assuming the flow is nearly parallel there. Thus, the upstream subsonic flow must deliver a nearly parallel flow to the throat. These transonic schemes are only valid for Mach numbers very near one and require some input regarding the shape of the nozzle near the throat, such as the throat radius of curvature. Full-scale aircraft or vehicles are sometimes tested in large wind tunnels, but these facilities are expensive to operate and some of their functions have been taken over by computer modeling and analysis in our case. Testing at transonic speeds presents additional problems, mainly due to the reflection of the shock waves from the walls of the test section. Therefore, perforated or slotted walls are required to reduce shock reflection from the walls. Since important viscous or inviscid interactions occur (such as shock waves or boundary layer interaction) both Mach and Reynolds number are important and must be properly simulated by selecting optimum design of nozzle and then analyzed in CFD.

The wind tunnel consists of five basic parts. The first part of the design was the contraction nozzle section that takes the high pressure low speed air from the settling chamber and converts it to low pressure high speed flow. The second part that was designed was the test section where the object being analyzed will be placed and tested. The choke block was the third part in the design. The choke block set the Mach number in the test section by changing the area ratio at the end of the test section. The diffuser was the forth designed part, and it reduces the flow speed and routes it out of the facility. The last part of the wind tunnel design was the stand. The stand allows for easy alignment in the settling chamber and for easy movement of the wind tunnel to storage when it is not in use. The AutoCAD software was used to model all of the components for the wind tunnel. An AutoCAD drawing of the full transonic wind tunnel design except for the diffuser support is shown in figure 1.

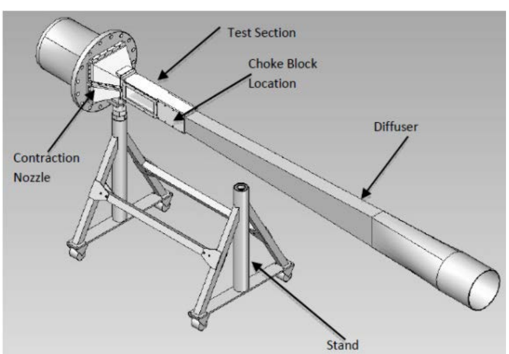

Figure 1 : CAD drawing of wind tunnel

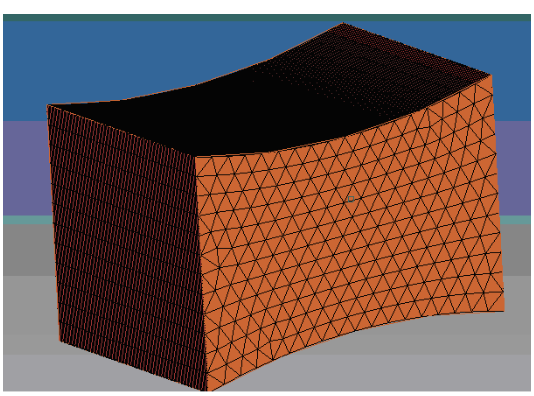

Figure 2 : Tetra mesh of wind tunnel

The meshing for analysis was done in Hypermesh as shown in figure 2. Blow down tunnels use the difference between a pressurized tank and the atmosphere to attain transonic speeds at the Test Section of the wind tunnel so that models of interest can be tested at transonic conditions for a short duration of time. The Sizing of Blow down wind Tunnel of 4 X 4 ft test section for varying M= 0.8 to 1.2 is considered here. The key driving design factors for wind tunnel can be the wind tunnel must be able to produce uniform transonic flow at the design Mach number in the test section, it must have the flexibility of being able to operate within a range of specified mach numbers, the run time must also be sufficient for testing purposes and theoretical run time calculations were performed in order to decide on the test section size. The basic fluid mechanics assumptions made for the flow in the wind tunnel includes fluid is air, flow is isentropic in the tunnel, ideal gas and air in the tanks undergoes polytrophic expansion.

High pressure air storage system will be dependent on the mass flows required and the frequency of runs. Pressure storage tanks are available on the shelf bases which are mounted horizontally or vertically. Tanks are painted black to absorb heat. They are provided with safety disk or pressure relief valve. As air is drawn from the storage, polytropic expansion takes place within the tank. This results in drop of reservoir temperature which is very bothersome. Fall of stagnation temperature causes resultant change in the stream temperature for a given Mach number. Change in temperature results in the change of viscosity which in turn affects the boundary layer thickness. Changes in Reynolds number and Mach number during a run are thus consequential to the fall in reservoir temperature. To maintain constancy of stagnation temperature, it is a practice to stack the reservoir volume with empty metallic cans. They serve as heat storing matrix during compression and release heat during the expansion process. Another way to maintain the constant stagnation temperature is by providing heater units in the reservoir.

The test section is situated downstream of the Nozzle. The dimensions of the test section were given to be 120cm X 120cm cross sectional area. The length of the test section was taken to be 180 cm. The temperature and velocity turbulence level of the flow in the test section region is low provided the velocity in the settling chambers is low and the contraction ratio going from the settling chamber to the minimum section of the nozzle is kept large (on the order of 24cm). From the point of view of the pressure loss calculation, the test chamber will be considered as a constant section duct with standard finishing surfaces. Apart from temperature ratios, pressure ratios, cross-Sectional area ratios, some other parameters are dynamic pressure, mass flow rate,

Global Journal of Researches in Engineering (D) Volume XIV Issue I Version I Year 2014

test section velocity, maximum velocity and free stream Reynolds number.

Design of transonic nozzle is passage used to transform pressure energy into kinetic energy and delivering flow at transonic speeds. It needs a combination of convergent and divergent nozzle (CD Nozzles). To deliver a transonic flow at the desired Mach number, the flow should be wave free and parallel. An improper contour results in the formation of finite shocks inside the nozzle by coalescence of weak waves and can prevent a uniform flow. The Method of Characteristics (MOC) provides a technique for properly designing the contour of the transonic nozzle. A method of characteristics is a very elaborate procedure for the creation of an accurate set of data points to create a nozzle, be it for a sharp expansion nozzle or a wind tunnel nozzle with a radius placed at the throat for more uniform flow at the exit plane. A transonic wind tunnel uses this method to create an expansion nozzle similar to those found on axisymmetric rocket engines to expand air to transonic speeds at the test section. The flow accelerates through a converging duct (M <1) and arrives at the throat (At) beyond which the geometry for smooth expansion is derived from the Method of Characteristics.

A diffuser is a device that is used to convert kinetic energy into enthalpy, pressure energy for an incompressible flow. For subsonic or transonic operation diffuser area increases and for supersonic flow, diffuser area decreases. In supersonic or transonic wind tunnels, most commonly used diffuser is of convergent divergent type (also called the second throat diffuser).

The supersonic or transonic nozzle is fitted behind the settling chamber. The settling chamber, to which a pressure gauge is attached, is used to maintain the stagnation pressure to provide flow to the nozzle. Settling chamber is made to with-stand a pressure about 10 atmospheres. It is made of mild steel usually.

Screens are used in settling chambers to gain the uniformity in the flow by reducing the turbulences. The screens are subjected to large stresses because of the large stagnation pressures used in the tests and because of the possibility of pressure shocks during the initial and final phases of the runs. In order to decrease the stresses, the screens may be placed in such a way that they present curved surfaces to the stream. Often supporting screens of large mesh and large diameter wires are placed behind each screen of fine mesh.

III. Analysis in Fluent

Fluent is a state of the art computer program for modeling fluid flow and heat transfer in complex geometries fluent provides complete mesh flexibility, including the ability to solve your flow problems using unstructured meshes that can be generated about complex geometries with relative ease supported mesh types includes 2D triangular-quadrilateral, 3D tetrahedral, hexahedral, pyramid, wedge, polyhedral and mixed meshes fluent also allows to refine your grid based on the flow solution. Fluent is written in the C computer language and makes full use of the flexibility and the power offered by the language. Consequently, true dynamic memory allocation, efficient data structures and flexible solver control all are possible. Initial conditions used are the specific heat ratio as 1.4, the stagnation temperature as 300K, the initial air supply tank temperature as 300K, the initial tank pressure as 11 bar, final pressure as 7.5 bar, stagnation pressure as 2 bar, the air supply tank volume as 18931m^3 and n as 0.768 for air as the polytropic exponent of expansion with 10 number of tanks. The wall boundary conditions are as shown in figure 3. Analysis results of the pressure coefficients contours are shown in figure 4.

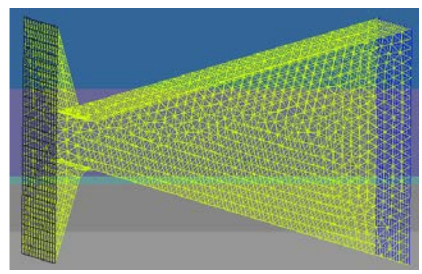

Figure 3 : wall boundary conditions

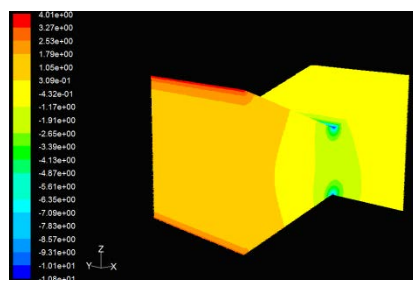

Figure 4 : contours of pressure coefficients

IV. Conclusion

The sizing of the new transonic wind tunnel is done. The results of CFD fluent keeping Area ratio constant with change in velocity, static pressure and pressure coefficient were found satisfactory. A response surface is constructed from a user-specified set of contour shapes for optima which is defined as the shortest nozzle with the maximum test length. This is achieved by delaying transition along the nozzle wall. The new design incorporates a section of increased diameter with the intention of enabling the tunnel to start in the presence of larger blunt models. The resulting flow fields are analyzed to see the shock effects and shear layers have on the test section flow and are good for nozzle design of Mach 1.2.

References Références Referencias

1. R. Mark Rennie, Peter Sutcliffe, Alexander Vorobiev, Alan B. Cain, "Mathematical modeling of wind speed transients in wind tunnels", 51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition 07-10 January 2013, Grapevine (Dallas/Ft. Worth Region), Texas.
2. Butler K, Cancel D, Earley B, Morin S, Morrison E, Sangenario M. Design and Construction of a Supersonic Wind Tunnel. Major Qualifying Project. Worcester Polytechnic Institute 2010. ME-JB3-SWT2.
3. Drikakis D. Introduction to Compressible Flows. Encyclopedia of Aerospace Engineering: John Wiley and Sons, 2010.
4. Balachandran, P. *Fundamentals of Compressible Fluid Dynamics*. New Delhi: Prentice-Hall of India, 2007. Print.
5. Mauricio G Silva, Victor O.R. Gamarra, Victor Koldaev, "Control of reynold number in a high speed wind tunnel", Journal of Aerospace Technology and Management, V.1, n.1, Jan-Jun 2009.
6. High Speed Wind Tunnel and Test Systems Design Handbook, Lockheed Martin Misiles And Fire Control, Texas, Publication Number Aer-Eir-13552-E.