International
Journal
Of Advanced Research in Engineering & Management (IJAREM)
ISSN: 2456-2033 || PP. 20-37

3D CFD ANALYSIS IN AN AFTERBURNER USING NUMECA

Anand R[1], Lokesharun D[2], Rajkumar S[3], Kirubakaran R[4]
[1]Assistant Professor
Department of Mechanical Engineering
Excel Engineering College.
Komarapalayam, Namakkal - 637303.
[2]Assistant Professor
Department of Aeronautical engineering
PARK College of Engineering & Technology.
Kaniyur, Coimbatore-641659.
[3]Master of Aeronautical Engineering,
Excel Engineering College.
Komarapalayam, Namakkal - 637303.
[4]Master of Aeronautical Engineering,
PARK College of Engineering & Technology.
Kaniyur, Coimbatore-641659.

Abstract: Afterburners are typically used as thrust augmentation devices critical for tactical maneuvers during combat. The current project proposes to study the internal flow patterns in an afterburner for non-reacting flows through numerical analyses using a commercial computational tool. Thrust generated on attaching the afterburner to the jet engine was provided by the non-reacting (cold) flow analysis. In the reacting (hot) flow analysis, a drastic increase in the exit velocity and temperature was observed as compared to the cold flow trials due to combustion, indicating approximately 35% increase in the thrust generated due to afterburning. The purpose of this project is to employ NUMECA software for the 3 Dimensional analyses in an afterburner by carrying out cold flow analysis. The liner is considered as a thin metal sheet. The holes on the liner are considered as a porous material. Heat source has been considered for the hot flow analysis. Conclusively, the study demonstrated the effect of an afterburner on a jet engine and the efficiency of the current afterburner design. It also provided an insight on ways to modify and improve the subsequent afterburner designs for obtaining even higher efficiencies.

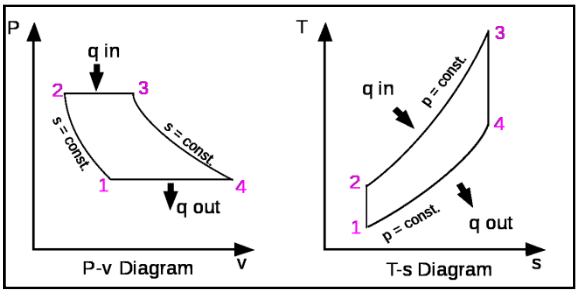
Keywords: Afterburner, Liner, V-gutter and Strut, Computational fluid dynamics, Non-reacting flow.

INTRODUCTION

Jet propulsion systems can be subdivided into broad categories: air-breathing and non-air- breathing. Air-breathing propulsion systems include the reciprocating, turbojet, turbofan, ramjet, turboprop and turbo shaft engines. Non-air-breathing engines include rocket motors, nuclear propulsion systems and electric propulsion systems. Gas turbine is a type of internal combustion engine, generally used to power aircraft, ships, generators, trains and tanks. These engines operate on three essential thermodynamics processes: isentropic compression, isobaric combustion and isentropic expansion, thus forming a Brayton cycle.

WORKING PRINCIPLE OF A GAS TURBINE ENGINE

Components of a gas turbine engine are compressor, a combustor and a turbine. The compression process increases the pressure and temperature of the gas (air). Energy of the gas is increased by the combustor after the compressor stage, by adding fuel to the gas and igniting the mixture. The hot combustion products are forced into the turbine, where the gas flowing at high velocity is directed through nozzles onto the blades of the turbine which turns the turbine shaft. Thus mechanical energy output of the turbine is enough to drive the compressor as well as obtain useful work.

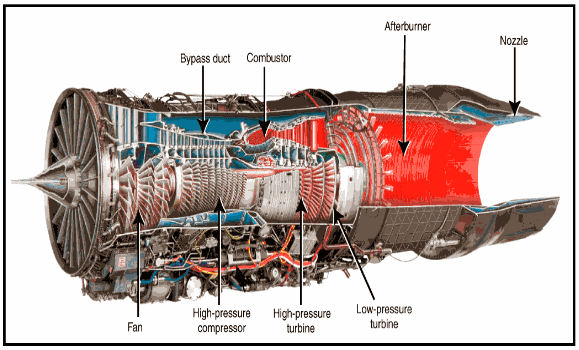


Brayton Cycle

The thermodynamic processes that make up the Brayton cycle are:

- (1-2) Isentropic compression
- (2-3) Isobaric (constant pressure)
- (3-4) Isentropic expansion:
- (4-1) Isobaric heat rejection

PARTS OF AN AERO GAS TURBINE ENGINE



- Intake
- Fan/Compressor
- Combustor
- Turbine
- Exhaust System
- Bypass Duct
- Afterburner (reheat)

THRUST AUGMENTATION

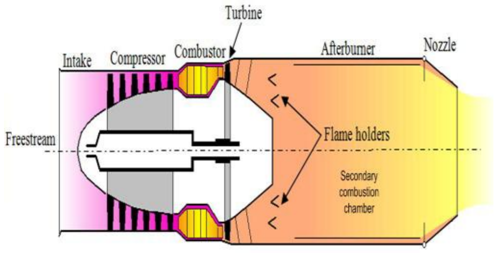
Thrust augmentation methods are employed in an aircraft to extract more power from the jet engine in order to curb the adverse climatic effects and minimize thrust losses. Fighter aircraft use these techniques for maneuverability and combat purposes. Injection and afterburning are the conventional techniques used for increasing the thrust.

- Injection
- Afterburning

AFTERBURNER

Afterburning (or reheat) is a method of augmenting the basic thrust of an engine to improve the aircraft take-off, climb and (for military aircraft) combat performance. The increased power could be obtained by the use of a larger engine, but as this would increase the weight, frontal area and overall fuel consumption, afterburning provides the best method of thrust augmentation for short periods. Afterburning consists of the introduction and burning of fuel between the engine turbine and the jet pipe propelling nozzle, utilizing the unburned oxygen in the exhaust gas to support combustion. The resultant increase in the temperature of the exhaust gas gives an increased velocity of the jet leaving the propelling nozzle and therefore increases the engine thrust. As the temperature of the afterburner flame can be in excess of 1,700 deg. C., the burners are usually arranged so that the flame is concentrated around the axis of the jet pipe. This allows a proportion of the turbine discharge gas to flow along the wall of the jet pipe and thus maintain the wall temperature at a safe value. The area of the afterburning jet pipe is larger than a normal jet pipe would be for the same engine, to obtain a reduced velocity gas stream. To provide for operation under all conditions, an afterburning jet pipe is fitted with either a two-position or a variable area propelling nozzle.

The nozzle is closed during non-afterburning operation, but when afterburning is selected the gas temperature increases and the nozzle opens to give an exit area suitable for the resultant increase in the volume of the gas stream. This prevents any increase in pressure occurring in the jet pipe which would affect the functioning of the engine and enables afterburning to be used over a wide range of engine speeds. The thrust of an afterburning engine, without afterburning in operation, is slightly less than that of a similar engine not fitted with afterburning equipment; this is due to the added restrictions in the jet pipe.

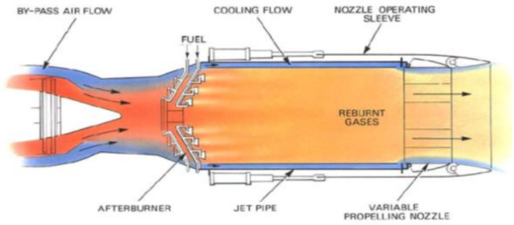


Schematic of single-spool turbojet engine with afterburner

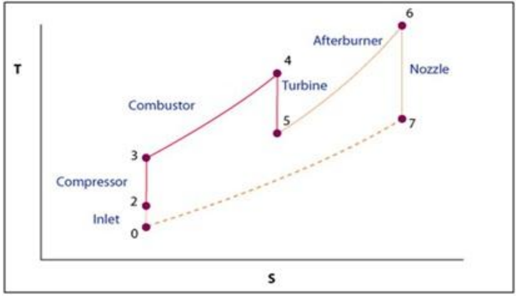
WORKING PRINCIPLE OF THE AFTERBURNER

Jet engine thrust is governed by the general principle of mass flow rate. Thrust depends on two things: the velocity of the exhaust gas and the mass of that gas. A jet engine can produce more thrust by either accelerating the gas to a higher velocity or by having a greater mass of gas exit the engine. Designing a basic turbojet engine around the second principle produces the turbofan engine, which creates slower gas but more of it. Turbofans are highly fuel efficient and can deliver high thrust for long periods of time, but the design trade-off is a large size relative to the power output. To generate the increased power with a more compact engine for short periods of time, an engine requires an afterburner. The afterburner increases thrust primarily by accelerating the exhaust gas to a higher velocity. While the mass of the fuel added to the exhaust does contribute to an increase in thrust, this effect is small compared to the increase in exhaust velocity.

The temperature of the gas in the engine is highest just before the turbine, and the ability for the turbine to withstand these temperatures is one of the primary restrictions on total dry engine thrust. This temperature is known as the Turbine Entry Temperature (TET), one of the critical engine operating parameters. To prevent the turbine from overheating, not all the oxygen can participate in the combustion and so additional fuel could be added to reignite the gases and cause a substantial increase in exhaust velocity and therefore total engine thrust. After passing the turbine, the gas expands at a near constant entropy, thus losing temperature. The afterburner then injects fuel downstream of the turbine and reheats the gas. In conjunction with the added heat, the pressure rises in the tailpipe and the gas is ejected through the nozzle at a higher velocity. The mass flow is also slightly increased by the addition of the fuel. Afterburners do produce markedly enhanced thrust as well as (typically) a very large flame at the back of the engine. This exhaust flame may show shock diamonds, which are caused by shock waves formed due to slight differences between ambient pressure and the exhaust pressure. These imbalances cause oscillations in the exhaust jet diameter over distance and cause the visible banding where the pressure and temperature is highest.



Afterburner system



T-S diagram of a simple turbojet engine with afterburner

Limitations

Due to their high fuel consumption, afterburners are usually used as little as possible; a notable exception is the Pratt & Whitney J58 engine used in the SR-71 Blackbird. Afterburners are generally used only when it is important to have as much thrust as possible. This includes takeoffs from short runways (as on an aircraft carrier) and air combat situations.

AFTERBURNER COMPONENTS

Diffuser

The flow entering the afterburner is first slowed to a Mach number that provides a balance between the total pressure loss and the afterburner cross-sectional area. A short diffuser length is desired without producing flow separation to reduce engine weight and length. In augmented turbofan engines, the diffuser may be combined with a mixer so that a mixed stream enters the combustion section.

Fuel Injection

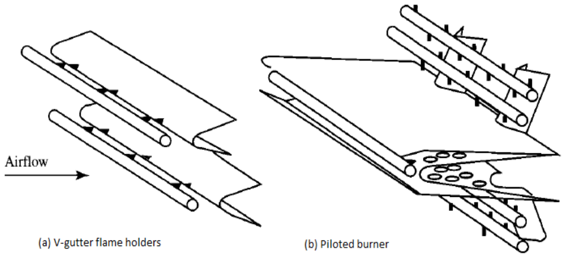
The goal of the fuel injection stream is to produce a specified distribution of fuel vapor in the gas stream entering the afterburner. In most engines, fuel is introduced I a staged manner so that the heat addition rate can be increased gradually from zero to the desired value. Because ignition, flame stabilization, and flame spreading are easiest to achieve when the fuel/air ratio is close to the stoichiometric value, staging is usually produced by adding fuel to successive annular stream tubes has its own set of fuel injectors and control systems which can be activated independently.

Ignition

Ignition of the fuel/air mixture in the afterburner is usually accomplished by using a spark or arc ignite or a pilot burner .Once initiated in the primary stream tube, combustion continues in the wake of a flame stabilizer and the process will spread to the rest of the flame stabilizer if the wakes of the stabilizer overlap. The spark or arc igniter uses a high-energy electric arc to initiate combustion of the primary stream tube.

Flame Stabilization

Two general types of flame stabilization devices that have been used in afterburner i.e., bluff body vee-gutter flame holder and piloted burners where a small piloting heat source is used to ignite the main fuel flow. The bluff body vee- gutter flame holder have the advantage of low flow blockage and low total pressure loss. They are simple and lightweight and have a good development history. The wake of flame holder is divided into two regions: a recirculation zone and mixing zone. The recirculation zone is characterized by a strong circulating flow, very low reaction rates, and a temperature that is nearly equal to the approaching stream. The mixing zones are characterized as turbulent regions of very strong shear, steep temperature gradients, and vigorous chemical reaction.



Types of flame stabilizers

Multiple-Flame-Holder-Arrays

The material presented above on flame holder stabilization was restricted to a single flame holder located on the centerline of a constant-area duct. When multiple flame holder are positioned in a single plane perpendicular to the approaching flow and spaced so that each lies on the centerline of equivalent ducts of equal height, then the above analysis can be used directly to estimate the stability characteristics. When flame holders are spaced irregularly, the analysis presented above does not directly apply. However, this analysis is useful in a qualitative manner.

Flame Speed

To achieve maximum combustion efficiency, the afterburner length needs to be longer than the burning length. The complex flame holder shape and interactions do not permit determination of afterburner length from the burning length measurements of basic flame holder experiments. However, most modern afterburners use 17 degree total angle foe estimating spread.

Afterburner Liner

The afterburner liner solves dual purpose of acting as a cooling liner as well as screech/anti-howl liner. It acts as a cooling liner to minimize combustion instabilities by isolating very high temperatures from the outer casing. Distributing a film of cool air along the length of the liner reduces the metal temperature and also subjects the outer hull to the afterburner pressure and cooling flow temperature. Finally, it is also used as a screech liner to prevent high frequency and amplitude pressure fluctuations due to combustion instabilities. This is accomplished by incorporating small holes along the initial length of the liner.

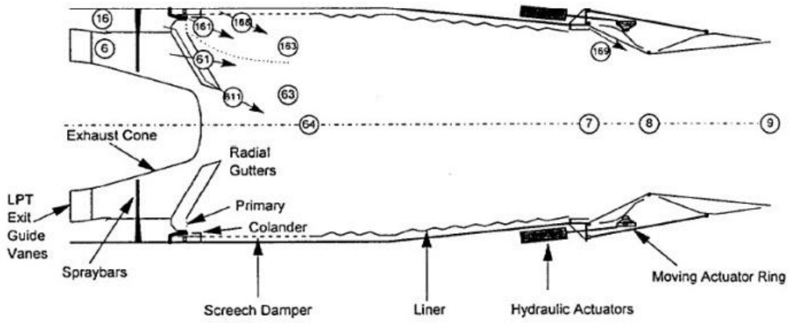
In a turbofan engine, the liner also separates the cold stream from the compressor entering the bypass and hot stream from exhaust entering the core region of the afterburner. Typically, the liner thickness ranges from 1-1.5mm, and the liner material must be chosen such that it can withstand the temperature variations on either side.

Casing

Casing is the outer hull of an afterburner, primarily used for accommodating all the components in it. The casing material must be light in weight, but strong enough to sustain the weight of entire system.

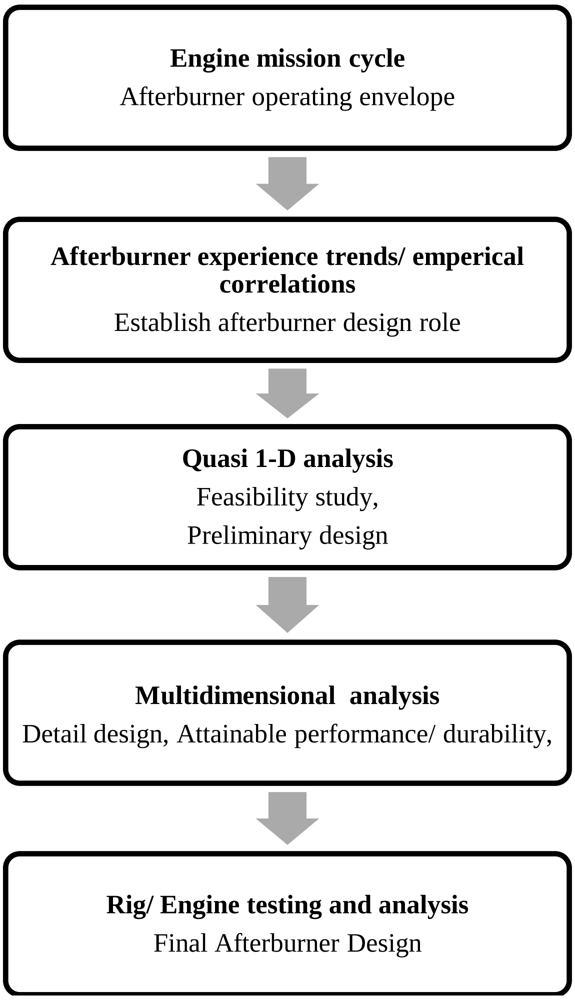
Exhaust Nozzle

The goal of convergent-divergent exhaust nozzle is to increase the velocity of the gas stream flowing through the afterburner, in turn augmenting the thrust produced by the engine. Modern day afterburners consist of a variable geometry nozzle to provide smooth operations during both non-afterburning (dry) and afterburning (wet) conditions. In case of a fixed geometry nozzle, the nominal conditions at the inlet of afterburner might get affected when the operating mode is switched from dry to wet condition or vice-versa.(7)



Longitudinal section of the EJ200 afterburner and nozzle

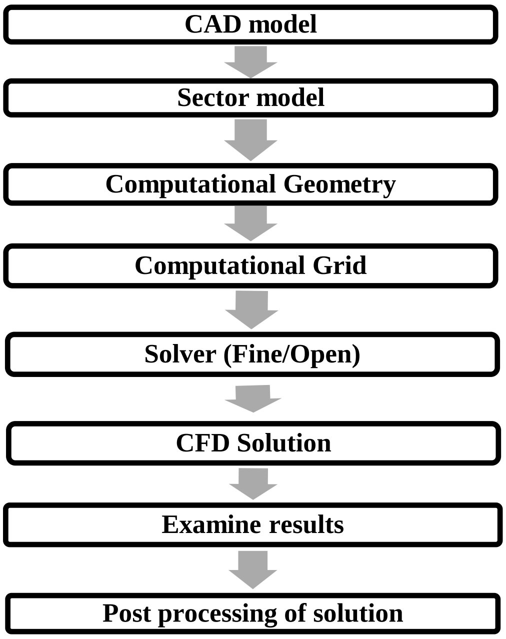
AFTERBURNER DESIGN APPROACH



Schematic of afterburner design approach

CFD

The CFD packages contains physical models for a wide range of applications including turbulent flows, heat transfer, reacting flows, chemical mixing, combustion, and multiphase flows. These physical models on unstructured meshes, imparts the benefits of easier problem setup and greater accuracy using solution-adaptation of the mesh. A CFD package is a state-of-the-art computer program for modelling fluid flow and heat transfer in complex geometries provides complete mesh edibility, including the ability to solve flow problems using unstructured meshes that can be generated about complex geometries with relative ease is used. Supported mesh types include



Flowchart of CFD Process

INTRODUCTION TO NUMECA

The resolutions of Computational Fluid Dynamics (CFD) problems involve three main steps:

1. spatial discretization of the flow domain
2. flow computation
3. visualization of the results

To perform these steps NUMECA has developed three software systems. The software systems, HEXPRESS are an automated all-hexahedral unstructured grid generation system. HEXPRESS also includes a module HEXPRESS /Hybrid, which allows meshing complex geometries with an isotropic hybrid mesh including mainly hexahedral cells with tetrahedral, pyramid and prism cells. The second software system, Fine/Open is a powerful CFD Flow Integrated Environment dedicated to complex internal and external flows. The third software system, CFView is a highly interactive Computational Flow Visualization system.

These three software systems have been integrated in a unique and user friendly Graphic User Interface (GUI), called FINE/Open, allowing the solution of complete simulations of 3D internal and external flows from the grid generation to visualization, without any file manipulation, through the concept of a project.

NUMECA is focused on innovation in CFD and multiphysics analysis and optimization. NUMECA has been FIRST to develop these unique, state-of-the-art software systems:

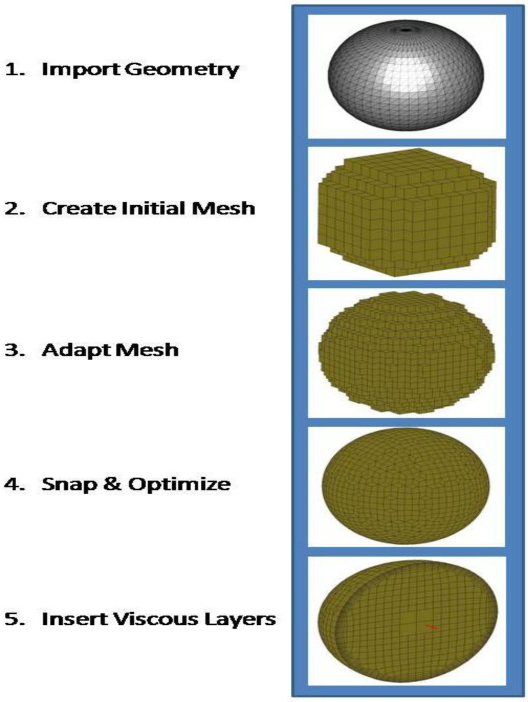
- Full Hex Automatic Meshing Solutions and Integrated CAD Cleaning with Auto Grid5™, HEXPRESS™
- Ultra-Fast Multiphysics CFD solvers with the CPU-Booster™
- Full unsteady multistage turbo machinery simulation with the Non-Linear Harmonic (NLH) method, leading to a CPU gain of 2 to 3, in orders of magnitude, compared to full unsteady simulations

- Full-Integrated blade shape optimization with FINE™/Design3Dand Auto Blade™ for all types of turbomachinery
- Open Labs, enabling the creation of your own CFD and multiphysics models without any programming.(9).
- Full integrated Aero-Vibro acoustic-CFD system with FINE™/Acoustics and Flow-Noise for fast broadband noise predictions
- Complete CFD environment dedicated to naval architects and marine engineers: FINE™/Marine
- The module for Uncertainty Quantification to reduce the risks associated to your simulation based design decision process.

HEXPRESS

HEXPRESS automatically converts the multi block structured mesh into an unstructured hexahedral mesh. It is a NUMECA's fully automatic hexahedral grid generator. It is automatic unstructured hexahedral mesh generator software designed to automatically generate meshes in complex 2D and 3D geometries.

Once model is repaired in cad fix, the model can be imported in HEXPRESS software for meshing purpose. HEXPRESS is automatic unstructured hexahedral mesh generator software designed to automatically generate meshes in complex 2D and 3D geometries.



Boundary Conditions

Boundary conditions are defined on the surface of the computational domain and are independent of the mesh. Since facets of the mesh are connected to the computational domain, boundary condition information can be easily retrieved for the mesh entities. It allows defining the type of boundary condition imposed on a patch or a group of patches.

Each block face can be divided in a certain number of patches on which will be applied the boundary conditions. It allows for a same face to have several different boundary conditions. Boundary conditions setting consist of imposing to each patch of the grid a type describing the physics of the flow. The boundary conditions setup is normally performed when the grid has been generated.

FLOW SLOVER

FINE™/Open with OpenLabs™ allows users to develop, improve and customize physical and numerical models as for instance:

- Transport equations (convection-diffusion-source)

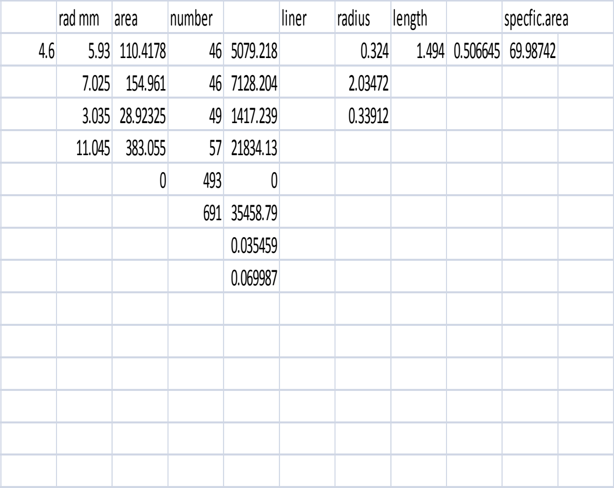
- Steady or unsteady source terms (heat, force, mass)
- Diffusion coefficients
- Algebraic relations
- Initial and boundary conditions
- Turbulence models, combustion models, radiation models
- atmospheric boundary layer models,
- actuator disk simulations,
- soot modeling,

FINE™/Open with Open Labs™ covers a large range of applications, such as aircrafts (including appendages), turbomachinery (cooled turbine blades and disk), automotive aerodynamics, environmental flows.

POROUS MEDIA MODEL

The Porous Media Model in FINE^{TM} / OPEN allows the user to simulate the flow in porous media. The source terms are added in both the momentum and energy equations for the porous media flow simulation with the objective to mimic:

- Pressure drop through medium
- Thermal effects due to heat exchangers between the medium and the fluid.



	rad mm	area	number		liner	radius	length		specfic.area	
4.6	5.93	110.4178	46	5079.218		0.324	1.494	0.506645	69.98742	
	7.025	154.961	46	7128.204		2.03472				
	3.035	28.92325	49	1417.239		0.33912				
	11.045	383.055	57	21834.13						
		0	493	0						
			691	35458.79						
				0.035459						
				0.069987						

CFView

CFView is a polyvalent scientific visualization software system offering all type of qualitative and quantitative outputs for 2D and 3D scalar or vector fields. Steady or unsteady data can be loaded, with 2D/3D steady state or transient animations.

CFView supports scripts for parametric visualization. Recorded templates can be used for automatic replay of saved screenshots on a new data set. Furthermore, it offers specific features for rotating and turbomachinery problems, such as meridional, blade-to-blade, pitch averaged and cylindrical views

DESCRIPTION OF WORK

- Extract geometry from full engine drawing.
- Check its coordinates if all points are valid by making graphs in excel sheets.
- Import to the grid generator.
- Create grid of 20^0 sector.
- Generate grid quality.
- Provide aerodynamic data.
- Execute the program.
- Analyse the results.
- Compare the results with fluent data.

PREMIXED FLAME

Premixed gases, a combustible mixture is available from the outset. Once the flame has been initiated at some point in the mixture, it will propagate throughout the entire volume of combustible mixture. The speed at which it propagates and the factors affecting its rate of propagation are of special interest to the designer of practical combustion systems. Turbulence is of prime importance because most flowing fuel-air mixtures are turbulent and turbulence is known to enhance flame speed considerably.

LAMINAR PREMIXED FLAME

The burning velocity of a flame, i.e., the rate at which a plane combustion wave will propagate through a gaseous flammable mixture, is determined partly by the rate of chemical reaction in the thin flame zone, and by heat and mass transfer from the flame to the unburned gas.

TURBULENT PREMIXED FLAME

Although it has long been recognized that flame speeds can be appreciably increased by turbulence, as evidenced by the very high burning rates achieved in both piston and gas turbine engine, the manner and extent of this influence are still not fully resolved.(16).

The ratio of turbulent to laminar flame speed in this region of high turbulence is given by

$$\frac{S_T}{S_L} = 0.5 \frac{u' \delta_L}{S_L \eta}$$

AERODYNAMIC DATA

In many calculations of mass transfer and spray evaporation it is convenient to work only with mean or average diameter instead of the complex drop size distribution. The most important mean diameter for combustion applications is the Sauter mean diameter, which is usually abbreviated to SMD or D_{32}. This is the diameter of a drop within the spray whose ratio of volume to surface area is the same as that of the whole spray.

Representative Diameter

There are many possible choice of representative diameter, each of which could play a role in defining the drop-size distribution. the various possibilities include the following :

$D_{0.1}$ = drop diameter such that 10% of the total liquid volume is in drops of smaller diameter.

$D_{0.5}$ = drop diameter such that 50% of the total liquid volume is in drops of smaller diameter. This is generally known as the volume median diameter.

$D_{0.632}$ = drop diameter such that 63.2% of the total liquid volume is in drops of smaller diameter.

$D_{0.9}$ = drop diameter such that 90% of the total liquid volume is in drops of smaller diameter.

If the drop-size data corresponds to a Rosin-Rammler distribution, all the representative diameters are uniquely related to each other via q. For example we have,

$$\text{MMD} = \text{SMD}(0.693)^{\frac{1}{q}} \Gamma\left(1 - \frac{1}{q}\right)$$

$$D_{0.1} = \text{MMD}(0.152)^{\frac{1}{q}}$$

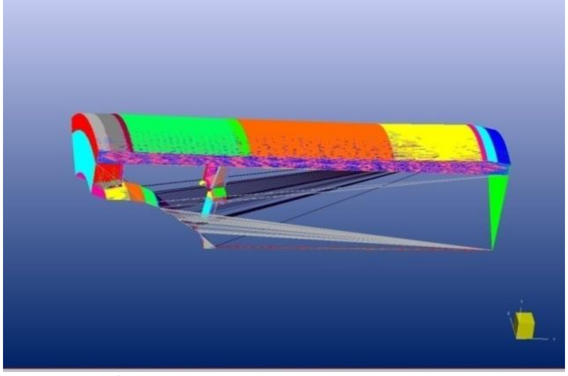
$$D_{0.9} = \text{MMD}\ (3.32)^{\frac{1}{q}}$$

$$D_{0.99} = \text{MMD}\ (9.968)^{\frac{1}{q}}$$

GEOMETRIC AND MODELLING

Computational Domain

The above geometry was replicated in HEXPRESS and a large computational domain was provided downstream of the nozzle exit to achieve a fully developed flow shown in figure.



Afterburner with computational domain

Once errors are removed in CADFIX the afterburner is then becomes suitable for importing in HEXPRESS. Using CAD manipulation tool bar the combustor model is checked. The combustor is imported in HEXPRESS as parasolid domain file. For convince the outer casing, liner, holes and cowl part are imported separately in CADFIX and HEXPRESS by closing the model in CATIA and then it is further assembled together. The afterburner parts are meshed separately as shown below

V gutter with computational domain

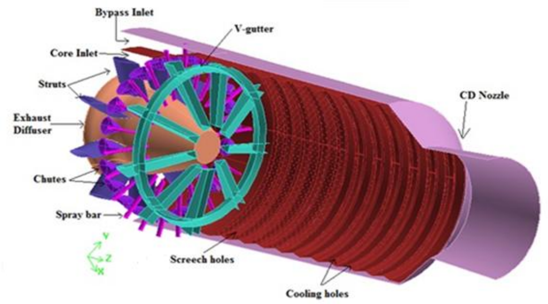
Boundary Conditions

Core Inlet conditions: p=2.31376 bar; T = 900K

Bypass Inlet condition : p =2.84306bar; T =475k

Geometric Modeling

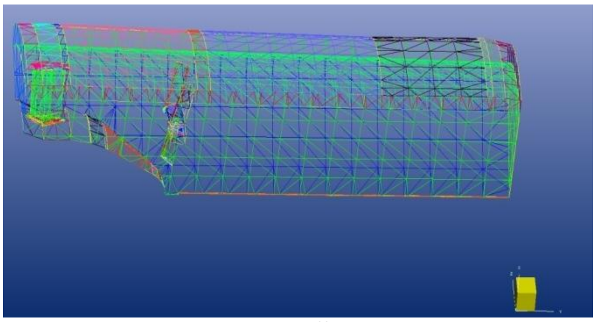
Sectional isometric view of the afterburner used in the study is presented in Fig 8.4. The model consists of core and bypass regions separated by a corrugated liner. The core region further houses the exhaust diffuser, struts for de-swirling and directing the exhaust gases from the turbine, chutes at the core-bypass interface, radial spray bars for fuel injection, v-gutters for flame stabilization, and a liner with cooling holes and screech holes. The mixed flow from core and bypass regions is finally compressed in a variable convergent-divergent nozzle at the exit. A variable C-D nozzle has been designed to adjust accordingly during non-afterburning and afterburning conditions.



Sectional view of afterburner

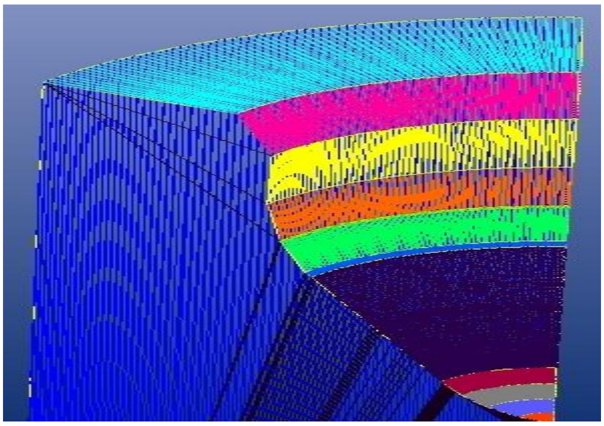
Component Geometry

The model analyzed in the present study is a 60^0 sector model with periodic rotational interfaces at both ends, shown in Fig. 6.2. The geometry has one radial v-gutter and one strut distributed at suitable spacing from the interfaces. Configuration of the C-D nozzle has been given appropriately for cold flow analyses. Exhaust cone struts and spray bars have been obliterated from the model due to limitations in the computational cells.



Exhaust Cone (Diffuser) Model

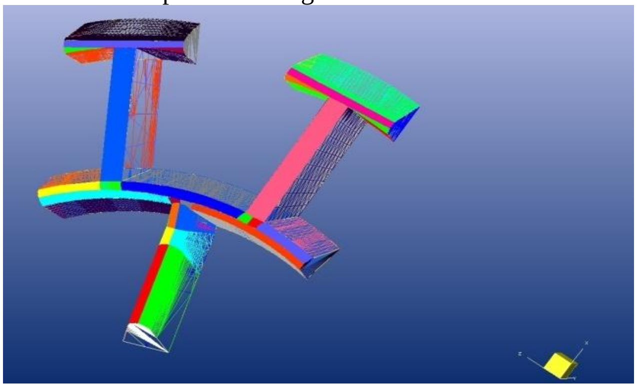
The sector model of exhaust cone is given in Figure. A smooth diffuser has been designed for the present model as it tends to have a uniform total temperature at the nozzle exit as compared to a truncated diffuser according to recent literature.



V-Gutter Model

Diffuser sector model

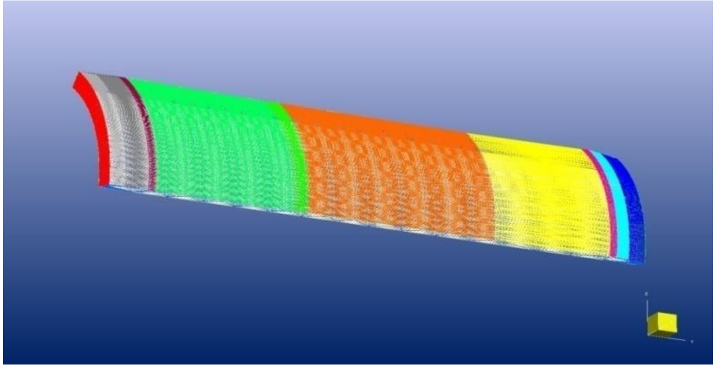
It shows the geometric model of the radial v-gutter and 60^0 sector model of the annular v-gutter that have been used as flame stabilizers in the present configuration.



V-gutter arrangement

Liner Geometry

Sector model of the liner containing screech holes and cooling rings is shown in figure.



Liner Geometry

RESULTS AND DISCUSSION

An afterburner works in two modes: non-reacting mode and reacting mode. The duration of thrust augmentation (reacting mode) is very less as the fuel consumption is very high for the process. Since the afterburner is an integral part of the jet engine, study of non-reacting flow is useful for estimating the performance of the engine during the non-afterburning periods. Reacting flow analysis is useful to test the efficiency of the afterburner.

PERFORMANCE PARAMETERS

Performance of the afterburner is evaluated by the thrust and thrust boost generated by the system, and also the total pressure loss across the system.

Pressure Loss

The total pressure loss across a component is calculated using the following formula.

$$Total\ Pressure\ loss\ \% = \frac{P_{0_i} - P_{0_o}}{P_{0_i}} * 100$$

In equation, P_{0_i} is the total pressure at inlet and P_{0_0} if the total pressure at the outlet of the component. Total pressure of the mixture at inlet is estimated by the mass-flow averaged values at core and bypass, as given in equation.

$$P_{0_i} = \frac{\left(P_{0_{core}} * \dot{m}_{core}\right) + \left(P_{0_{bypass}} * \dot{m}_{bypass}\right)}{\dot{m}_{core} + \dot{m}_{bypass}}$$

Thrust Estimation

The formula for calculating thrust is given in the equation below.

$$Thrust = \dot{m}_e v_e + (p_e - p_a)A_e$$

In equation 7.3, $\dot{m}_e$, v_e and p_e are the mass-flow rate, velocity and pressure at the nozzle exit and A_e is the nozzle exit area.

Thrust Boost

The additional thrust generated by combustion in an afterburner is termed as thrust boost.

$$Thrust\ boost\ \% = \frac{Th_{wet} - Th_{dry}}{Th_{dry}} * 100$$

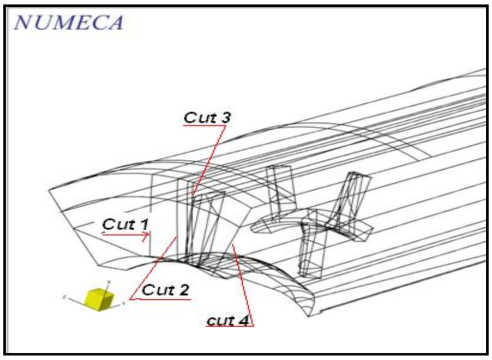
In equation , Th_{wet} is the thrust generated in hot flow condition and Th_{dry} is the thrust generated during cold flow condition.

INTERPRETATION OF RESULTS FOR COLD FLOW

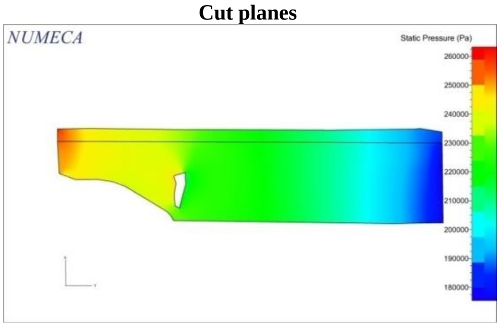
There are four cut planes for which static pressure, total pressure, static temperature, velocity vectors, and mach numbers have been generated. These planes are shown in There are 4 cut planes which pass through lower v-gutter, central v-gutter and outer v-guter.

Cut plane 1

This figure shows the static pressure distribution. It can be seen that pressure has reduced from gradually from 2.6 bar to 1.8 bar.

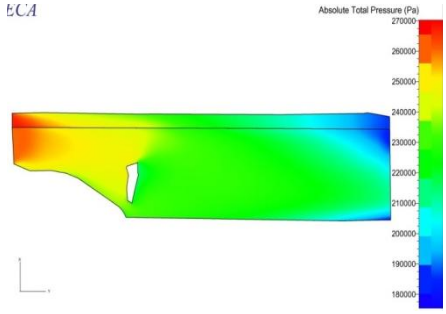


Cut planes

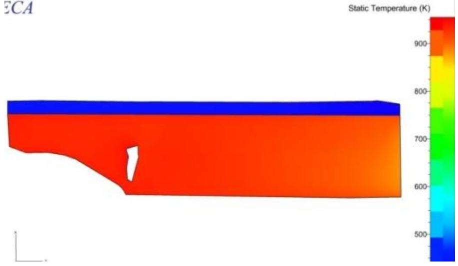


Static pressure distribution

The following shows the total pressure, static temperature, velocity vectors and Mach number distribution. It can be observed that the total pressure is reducing. The loss is found to be at higher values 19%. The static temperature is found to have a very less change. The flow temperature reduces near the rear probably due to mass flow entrainment from the bypass duct.

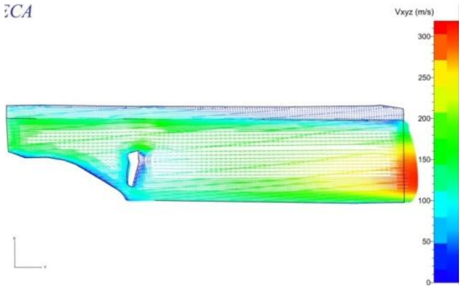


Total pressure distribution



Static temperature distribution

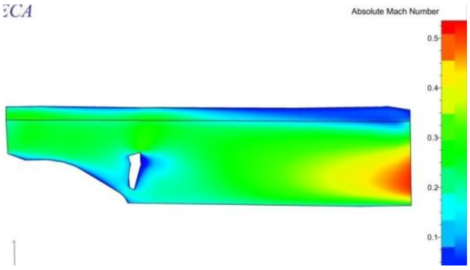
Though temperature does not affect the thrust directly, it plays a key role in determining several factors in an afterburner. Gas properties vary with respect to temperature, which is the reason for high exit velocity in the reacting condition as compared to the non-reacting condition, even though the compression of gas is almost equal in both the cases. Another factor affected by the temperature distribution is the material selection of an afterburner



Velocity vector distribution

The velocity vectors are shown in Figure. It can be observed that there is a recirculation zone near the v-gutter zone. It has captured the bluff body phenomena satisfactorily. The flow in the bypass duct has a velocity of 100 m/s at inlet and gradually it has reduced.

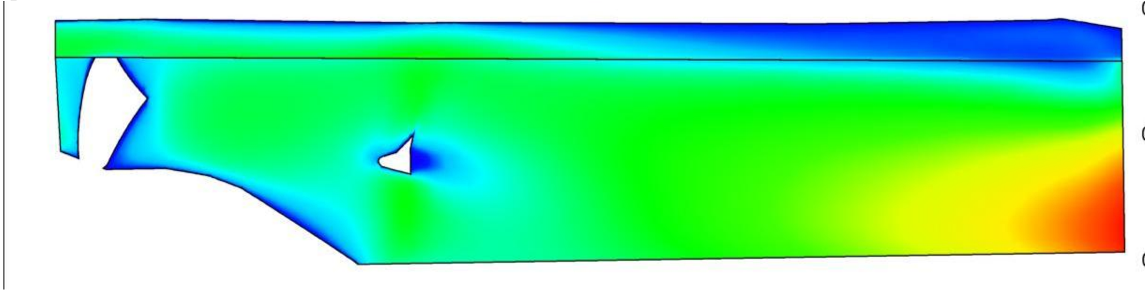
The Mach number has reduced gradually from .3 to .15. The average Ach number is found to be 0.4. The most essential parameter to predict the afterburner performance is the thrust that it generates. Since thrust directly depends on the velocity at nozzle exit, a comparison was made between the exit velocities in cold flow conditions.



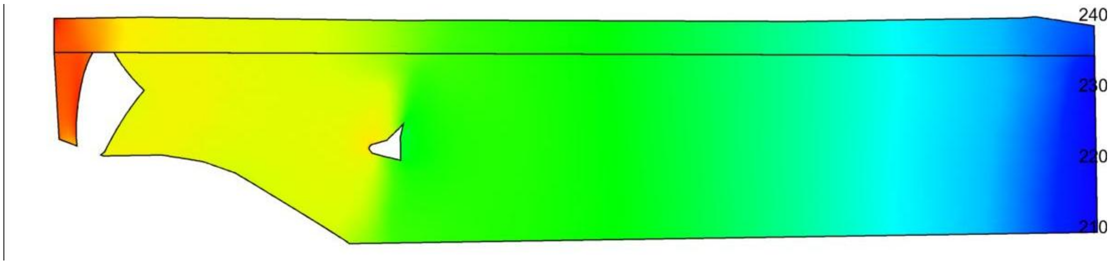
Mach Number distribution

Cut plane 2

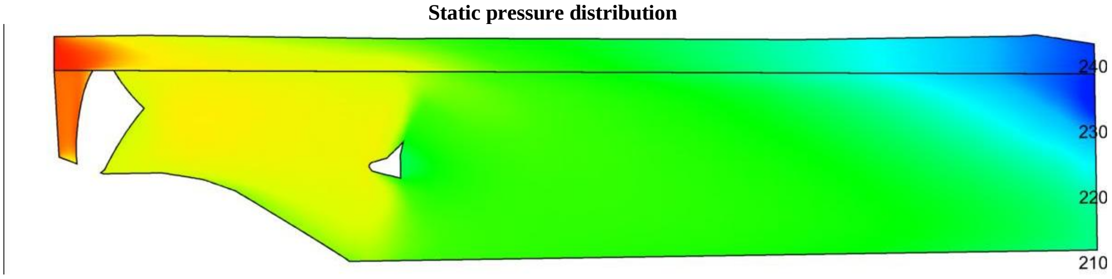
The following figure shows the Mach number, static and total pressure, velocity vectors and static temperature distribution.



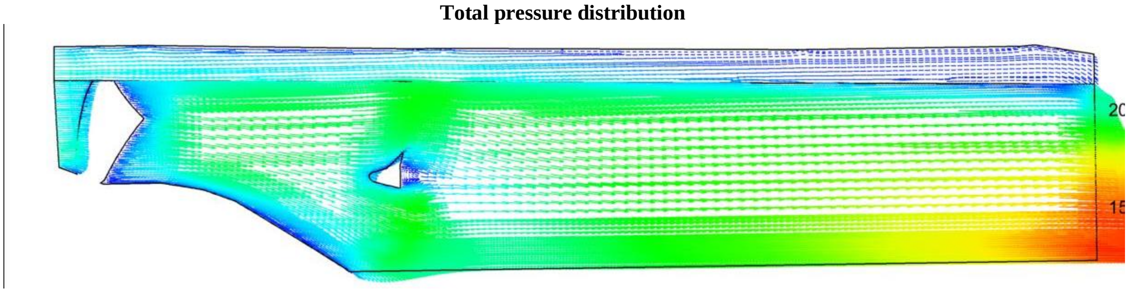
Mach number distribution



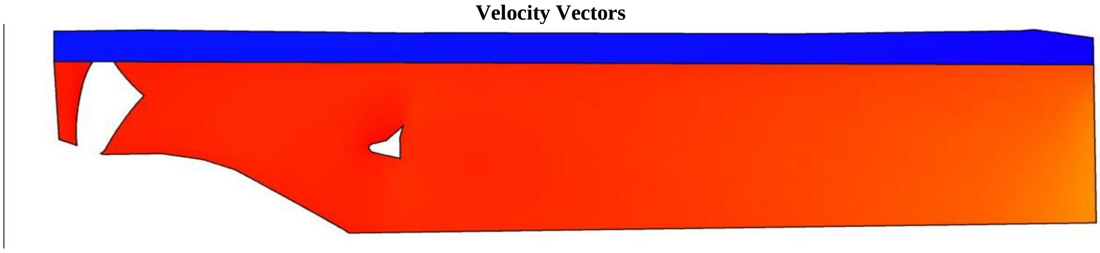
Static pressure distribution



Total pressure distribution



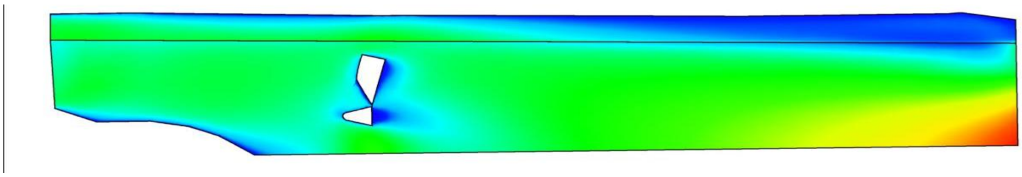
Velocity Vectors



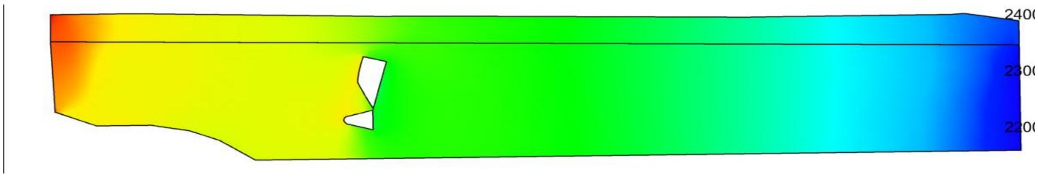
Static temperature distribution

Cut plane 3

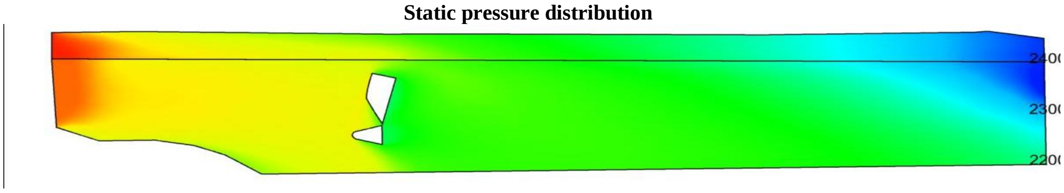
The following figure shows the Mach number, static and total pressure, velocity vectors and static temperature distribution in cut plane 3.



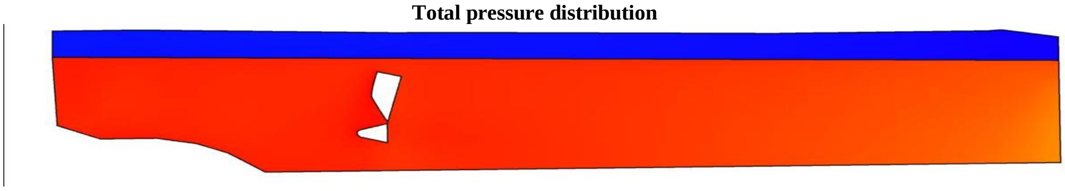
Mach Number distribution



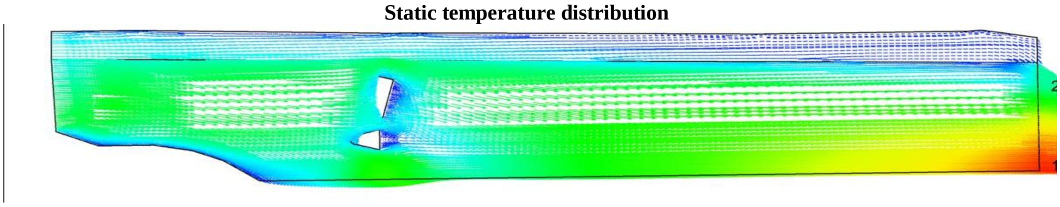
Static pressure distribution



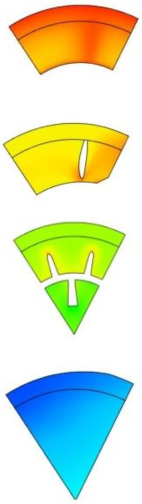
Total pressure distribution



Static temperature distribution

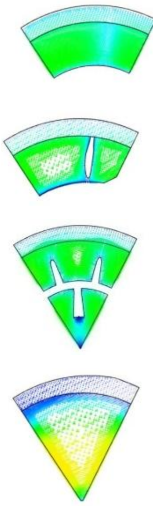


Velocity vector



Temperature distribution in the afterburner

pressure distribution in the afterburner



Velocity vectors in the afterburner

The total pressure loss in cold condition has been found to be 17%. The results are comparable with the results obtained by FLUENT software.

CONCLUSION

Numerical prediction of flow was carried out inside a practical gas turbine afterburner under non-reacting conditions. The following conclusions were drawn from this study.

1. A validation study of CFD software was conducted using data available from open literature.
2. In the present afterburner analyses, a decrease in the velocity of gas stream along the path of diffuser was observed. The flame stabilizer (v-gutter) also created a low velocity zone downstream.

The three dimensional cold flow analysis has been successfully done for a practical afterburner system. The flow field shows that desired wakes are formed behind the flame holders. The velocity increases in the CD nozzle along its length and reaches its maximum value at the exit plane of the nozzle. The flow is highly de-swirled at the jet plane exit nozzle.

REFERENCES

[1]. J.D. Mattingly, Elements of Propulsion: Gas Turbines and Rockets, 2nd Edition, American Institute of Aeronautics and Astronautics, Virginia, 2006.

[2]. T.E. Kuhn, H.C. Mongia, T.W. Bruce, E. Buchanan, "*Small turbine engine augmentor design methodology*", In Proceedings of 18th AIAA/SAE/ASME Joint Propulsion Conference, Cleveland, Ohio, USA, 1982.

[3]. A. Trovati, F. Turrini, C. Vinci, "*Afterburner design and development*", In Proceedings of North Atlantic Treaty Research and Technology Organization Meeting, Turin, Piedmont, Italy, 1999.

[4]. J.A. Lovett, T.P. Brogan, D. Philippona, "*Development needs for advanced afterburner designs*", In Proceedings of 40th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, Fort Lauderdale, Florida, USA, 2004.

[5]. R.K. Kumar, "*A method of estimating pressure loss across flame holders in high velocity streams*", Combustion Science and Technology, Volume 21, Issue 5-6, pp. 199-203, October 1980.

[6]. J.D. Anderson, Computational Fluid Dynamics: The Basics with Applications, 1st Edition, McGraw Hill, New York, 1995.

[7]. EXERGETIC ANALYSIS OF AN AIRCRAFT TURBOJET ENGINE WITH AN AFTERBURNER, Mehdi Aliehyaei EHYAEIa*, Abdolhasan ANJIRIDEZFULIb, and Mark A. ROSEN.

[8]. Preliminary Design of a Short Afterburner for Single-Spool Expendable Turbojet Engine, Basharat Ali Haider_ and Nikola Davidovi_ University of Belgrade, Serbia Naveed Durrani z University of Sheld, UK

[9]. User Manuals of NUMECA FINE/Open International 5, Avenue Franklin Roosevelt 1050 Brussels Belgium.