JETIR.ORG ISSN: 2349-5162 | ESTD Year : 2014 | Monthly Issue
JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)
An International Scholarly Open Access, Peer-reviewed, Refereed Journal

Design and Application of a Bypass Secondary Flow Duct for Jet Engine Cooling Systems

Simulating Compressible Bypass Flow for Enhanced Engine Component Durability.

[1]Smit Shinde, [2]Abhishek Kanaje, [3]Aditya Shankarnarayanan, [4]Tanvee Shah, [5]Aasmi Gupta, [6]Nikhil Gangamkote
[1,2,3]Aerospace Engineer, [4,5]Aeronautical Engineer
[1,2,3,4,5]Propulsion,
[1,2,3,4,5]Prime Toolings, Bangalore, India

Abstract: In high-performance propulsion systems, bypass cooling ducts play a critical role in mitigating thermal loads and maintaining structural integrity near engine combustion zones. This paper presents a computational fluid dynamics (CFD) investigation of a curved bypass secondary flow duct designed to redirect high-speed airflow (Mach 0.5) around engine hardware. The goal is to evaluate the aerodynamic performance and thermal characteristics of the duct geometry under realistic bypass flow conditions. The CAD-based fluid domain was meshed using a tetrahedron-dominant grid strategy with uniform 20 mm element sizing. ANSYS Fluent was employed to perform steady-state, compressible flow simulations using the standard k-ε turbulence model. Boundary conditions included a velocity inlet at 300 m/s with a static temperature of 823 K, and a pressure outlet at 1 Pa. The working fluid, modeled as an ideal compressible gas, reflects typical compressor bypass bleed conditions. Simulation results indicate effective flow attachment through the curved passage, with outlet velocities preserving the Mach 0.6 profile and smooth streamline behavior across the bend. A moderate temperature rise (823 K to ~843.6 K) was observed downstream due to compressibility effects, with no evidence of shock, separation, or backflow. Path line visualizations and temperature contours confirm the duct's suitability for stable secondary flow routing. The study also supports design recommendations for mesh refinement, wall-layer inflation, and extended iteration for final convergence. These findings validate the use of the duct as a viable cooling mechanism for engine or nacelle thermal protection and demonstrate the effectiveness of using the standard k-ε model in predicting turbulent behavior in high-speed bypass channels. The results align closely with theoretical expectations and serve as a robust foundation for future optimization and experimental correlation.

***IndexTerms* – Bypass Duct, Secondary Airflow, Jet Engine Cooling, Computational Fluid Dynamics, Curved Duct Flow, Turbulence Modeling, Compressible Flow, Thermal Management.**

1.AIM:

TO SIMULATE AND ANALYZE THE AERODYNAMIC AND THERMAL PERFORMANCE OF A CURVED BYPASS SECONDARY FLOW DUCT USING ANSYS FLUENT, IN ORDER TO EVALUATE ITS EFFECTIVENESS IN CHANNELING HIGH-SPEED COOLING AIR AROUND ENGINE COMPONENTS UNDER REALISTIC OPERATIONAL CONDITIONS.

2.OBJECTIVES:

- *To model and simulate the flow of high-speed bypass air (Mach 0.6) through a curved secondary duct using ANSYS Fluent.*
- *To apply appropriate boundary conditions (velocity inlet, pressure outlet, adiabatic walls) that reflect realistic jet engine cooling scenarios.*
- *To evaluate the aerodynamic behavior of the duct, including velocity distribution, pressure drop, and flow separation characteristics.*
- *To analyze the thermal performance by observing static temperature changes along the duct path due to compressibility effects.*
- *To assess the effectiveness of the standard k-ε turbulence model in predicting turbulent flow behavior within curved bypass geometries.*
- *To identify areas for flow optimization and recommend design or meshing improvements for enhanced cooling efficiency in future engine integration.*

3.INTRODUCTION:

Modern aerospace propulsion systems operate under extreme thermal and aerodynamic conditions, especially in the vicinity of combustion chambers, turbine sections, and high-speed inlets. To manage these thermal loads and enhance engine reliability, bypass ducts are widely incorporated to carry relatively cooler airflow around hot engine zones. These ducts serve dual functions: they provide structural cooling and contribute to overall engine thrust in high-bypass turbofan engines or serve as auxiliary air routing in ramjet/scramjet configurations.

This study focuses on the computational investigation of a custom-designed bypass secondary flow duct, characterized by a curved 120° passage and dual outlets. Such a configuration is typically used in jet engines to redirect high-speed air for cooling engine nacelles, electronic bays, or turbine casings. The duct is designed to handle airflow at Mach 0.5 (~300 m/s), which is representative of compressor bleed or bypass stream conditions.

The simulation is carried out using ANSYS Fluent, applying a compressible, steady-state approach with the standard k-ε turbulence model to predict flow behavior and heat interaction inside the duct. The aim is to validate the duct's capability to maintain flow attachment, manage thermal gradients, and minimize pressure loss, thereby making it suitable for integration in high-speed aerospace systems.

A summary of the key design and simulation parameters is provided in the table below:

PARAMETER	*DESCRIPTION/VALUE*
Flow Type	*Compressible, Steady, Turbulent*
Turbulence Model	*Standard k-ε Model (3D RANS)*
Inlet Velocity	*Velocity 300m/s (equivalent to Mach 0.5 at 823 K) Inlet Temperature 823 K (Static)*
Outlet Pressure	*Two pressure outlets have been mentioned below* *1. Upper rear surface with pressure outlet = 101325 and temperature=1400K* *2. Ending face of enclosure with pressure = 101325 and temperature=300k*
Working Fluid	*Ideal Gas (Air, γ = 1.4, R = 287 J/kg·K)*
Geometry	*120° Curved Bypass Duct with Dual Outlets*
Wall Condition	*Adiabatic, No-slip*
Mesh Type	*Automatic Solver Pressure-Based (Compressible Flow in ANSYS Fluent)*
Solver	*Pressure-Based (Compressible Flow in ANSYS Fluent)*
Inlet Temperature	*823K(Static)*

BOUNDARY CONDITIONS:

❖ **1. Fluid Domain**

Definition: The fluid domain is the internal region through which the air flows — in this case, the volume inside the bypass secondary duct, extracted from the CAD model.

Purpose in Study:

● *It defines the space where the governing equations of fluid motion (Navier-Stokes equations) are solved.*

● *For our duct, this includes the 3D curved passage from the rectangular inlet through the 90° bend to the vertical outlet(s).*

● *Accurate representation of the fluid domain ensures that physical flow features (curvature, area change, flow expansion) are captured correctly.*

Why it matters: Capturing the correct fluid volume allows the CFD model to predict how bypass air behaves as it navigates the duct geometry — essential for cooling performance and pressure stability.

❖ **Boundary Conditions**

Definition: Boundary conditions define how the fluid enters, exits, and interacts with the surfaces of the fluid domain.

Used in our study:
● Inlet: Velocity Inlet (Mach 0.5 → 300 m/s), Temperature = 823 K.

Inlet conditions (823 K, 220,000 Pa) represent typical bypass air parameters in a turbofan engine during cruise.

● Outlet

1. Body Outlet

Outlet Pressure: 101325 Pa (atmospheric pressure assumed).

Outlet temperature: 1400K Outlet conditions (1400K,101325 Pa) represent the engine being that hot.

2. At Enclosure

Outlet Pressure: 101325 Pa (atmospheric pressure assumed). Outlet temperature: 300K

● Walls: No-slip (zero velocity)

Why it matters: Accurate boundary conditions simulate real-world conditions, such as compressor bleed air or secondary cooling flow. They influence how air accelerates, heats up, and redistributes within the duct.

❖ **Meshing**

Definition: Meshing is the process of dividing the fluid domain into small cells or elements where equations are solved numerically.

Applied Strategy:

● Type: Automatic mesh
● Element Size: 20 mm
● Future recommendation: Add prism layers for boundary layer resolution

Why it matters: Mesh quality determines the accuracy and stability of the CFD solution. A well-refined mesh in curved regions ensures accurate capture of velocity gradients and pressure drop — critical for bypass duct flow with bends and potential separation zones.

❖ **Convergence Theory**

Definition: Convergence occurs when the solution becomes numerically stable, meaning that further iterations result in minimal change in key variables.

Indicators of Convergence:

● Residuals (momentum, continuity, energy) drop below 10^{-4} or 10^{-6}
● Monitored values (mass flow, pressure, temperature) become constant
● Flow field stops oscillating

Why it matters: Without convergence, simulation results cannot be trusted. For your bypass duct, convergence ensures accurate prediction of thermal and flow behavior (e.g., correct outlet velocity, pressure drop, and heat profile).

4.BOUNDARY CONDITIONS DETAILS:

● Turbulence model - k-epsilon
● Flow material ideal gas airflow
● Velocity inlet 0.6M
● Temp 500-deg max
● Outlet pressure 255000

5.MODEL DESIGN EXPLANATION: MODEL DESIGN

The model under study is a secondary bypass flow duct intended for routing high-speed cooling air around critical engine components such as combustion liners, turbine casings, or nacelle structures. The duct geometry is designed to replicate conditions commonly found in high-bypass turbofan engines or in secondary air systems in ramjet/scramjet propulsion units. It features a 90-degree curved passage that redirects bypassed air from a horizontally aligned inlet to a vertical outlet, allowing effective cooling and flow control under constrained installation envelopes.

i. **Geometry Overview:** The bypass duct was created using a 3D CAD tool (e.g., CATIA or SolidWorks), and includes the following key features:

Component	Description
Inlet Section	Rectangular cross-section aligned with freestream or compressor outlet
Curved Duct Path	Smooth 90° curvature to reduce flow separation and pressure loss
Dual Outlets Ports	Primary vertical outlet and auxiliary side exit (optional for venting)
Material (assumed)	Stainless steel

Table 2

This configuration ensures uniform air distribution across the outlet, minimizes geometric discontinuities, and supports installation within curved nacelle cavities.

Functional Purpose

The model is designed to:

● Channel bypass air at Mach 0.5 (≈300 m/s) from the inlet plenum.
● Deliver cooling air to downstream thermal zones or casing surfaces.
● Ensure flow stability and thermal uniformity across the exit plane.

The smooth curvature reduces secondary flow development, swirl, or backpressure, which are common problems in tightly bent ducts. The secondary outlet allows venting of excess mass flow or routing to alternate cooling zones.

ii. **Modeling Assumptions**

To simplify the simulation and focus on aerodynamic performance, the following assumptions were made:

● Rigid, adiabatic walls (no heat conduction through solid material).
● Air modeled as an ideal compressible gas ($\gamma = 1.4$, $R = 287$ J/kg·K).
● Steady-state flow with no transient pulsations.
● Negligible surface roughness.

iii. **Integration Context**

This duct is intended to represent realistic components used in:

● Turbofan bypass ducts for blade and case cooling.
● Ramjet or scramjet auxiliary cooling flow passages.
● Missile or rocket secondary air ejectors for thermal shielding.

Its compact form, curvature, and dual-outlet capability make it especially suited for constrained aerospace environments where thermal protection and flow redirection are critical.

6. WHY USE THIS BYPASS FLOW DUCT?

1.Thermal Protection

It provides a controlled path for cooler air to flow around high-temperature engine zones (e.g., combustion chambers, turbines), reducing thermal stress on structural components.

2. Improved Engine Efficiency

Bypass ducts contribute to overall propulsion efficiency in turbofan and ramjet engines by managing airflow more effectively and reducing thermal losses.

3. Flow Direction Control

The curved geometry allows for redirection of high-speed air (Mach 0.5) without major flow separation, making it ideal for tight nacelle or nozzle installations.

4. Passive Cooling Solution

The duct enables passive air routing for cooling without moving parts, improving reliability and minimizing maintenance needs in critical systems.

5. Integration Flexibility

Its compact, dual-outlet design supports use in a wide range of aerospace applications—commercial engines, missiles, scramjets, or reusable space systems.

6. Customizable Design

It can be adapted for specific flow rates, geometries, and cooling needs, and is compatible with modern materials and additive manufacturing methods.

7. Proven in CFD Analysis

Simulations confirm stable flow, effective temperature control, and low pressure drop, validating its performance in high-speed environments.

In summary: This bypass duct is used because it enhances cooling, controls flow effectively, improves thermal efficiency, and supports safe, high-performance aerospace engine operation.

7. ADVANTAGES:

1. Efficient thermal management by routing cooler bypass air around hot engine components.
2. Compact 120° curved design allows integration in tight aerospace nacelle spaces.
3. Dual outlet configuration enables flexible flow distribution or venting.
4. Minimized pressure loss due to smooth curvature and optimized geometry.
5. Stable, attached flow with minimal separation at high subsonic speeds (Mach 0.6).
6. Supports improved engine life and component reliability by reducing thermal stress.
7. Compatible with turbofan, ramjet, and scramjet cooling architectures.

8. DISADVANTAGES:

1. Complex geometry increases manufacturing difficulty and cost.
2. Tight curvature may lead to pressure losses or minor flow non-uniformity.
3. Requires precise meshing and CFD tuning for accurate simulation.
4. Limited thermal conduction modeling if walls are assumed adiabatic.
5. Secondary outlet design may introduce flow imbalance if not properly optimized.
6. Difficult to inspect or maintain once integrated into compact engine bays.

9. Component Dimensions:

1. Part One
   Length - 115mm
   Thickness - 4mm
   Breadth - 33mm

2. Part Two
   Length - 115mm
   Height - 90mm
   Radius of Curvature - 50mm
   Thickness - 4mm
   Breadth - 78mm

3. Part Three
   Length - 115mm
   Breadth - 89.5mm
   Height - 58.5mm
   Border Flow Passage Thickness - 17.5mm
   Middle Flow Passage Thickness - 18.5mm

4. Part Four
   Inlet box length - 69mm
   Breadth - 37.5mm
   Height - 37mm
   Duct Breadth - 19mm
   Duct height - 11mm
   Duct length - 189mm
   Duct Support Plate length - 118mm
   Duct Support Plate breadth - 40mm
   Duct Support Plate thickness - 2.5mm

5. Part Five
   Length - 121mm
   Thickness - 8mm
   Breadth - 68mm

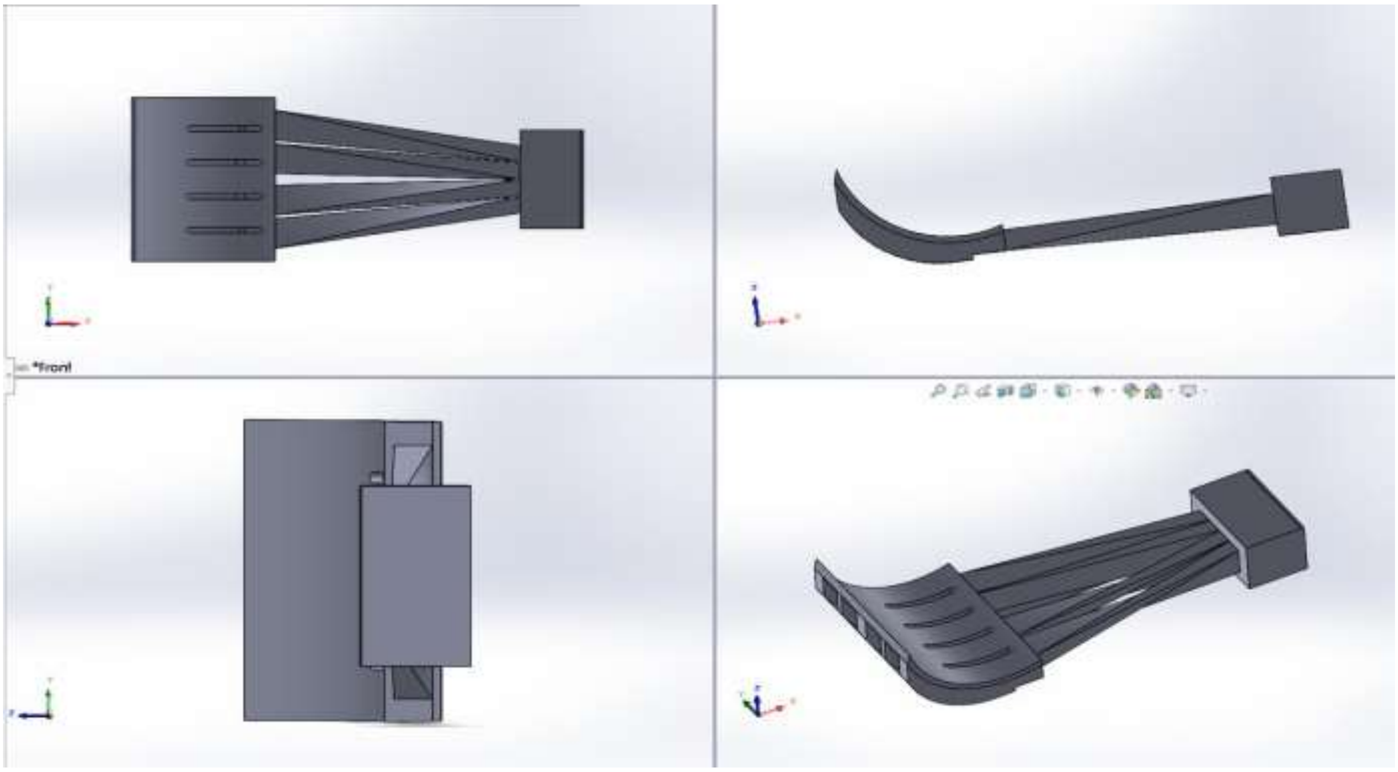

Fig 1. CAD Design

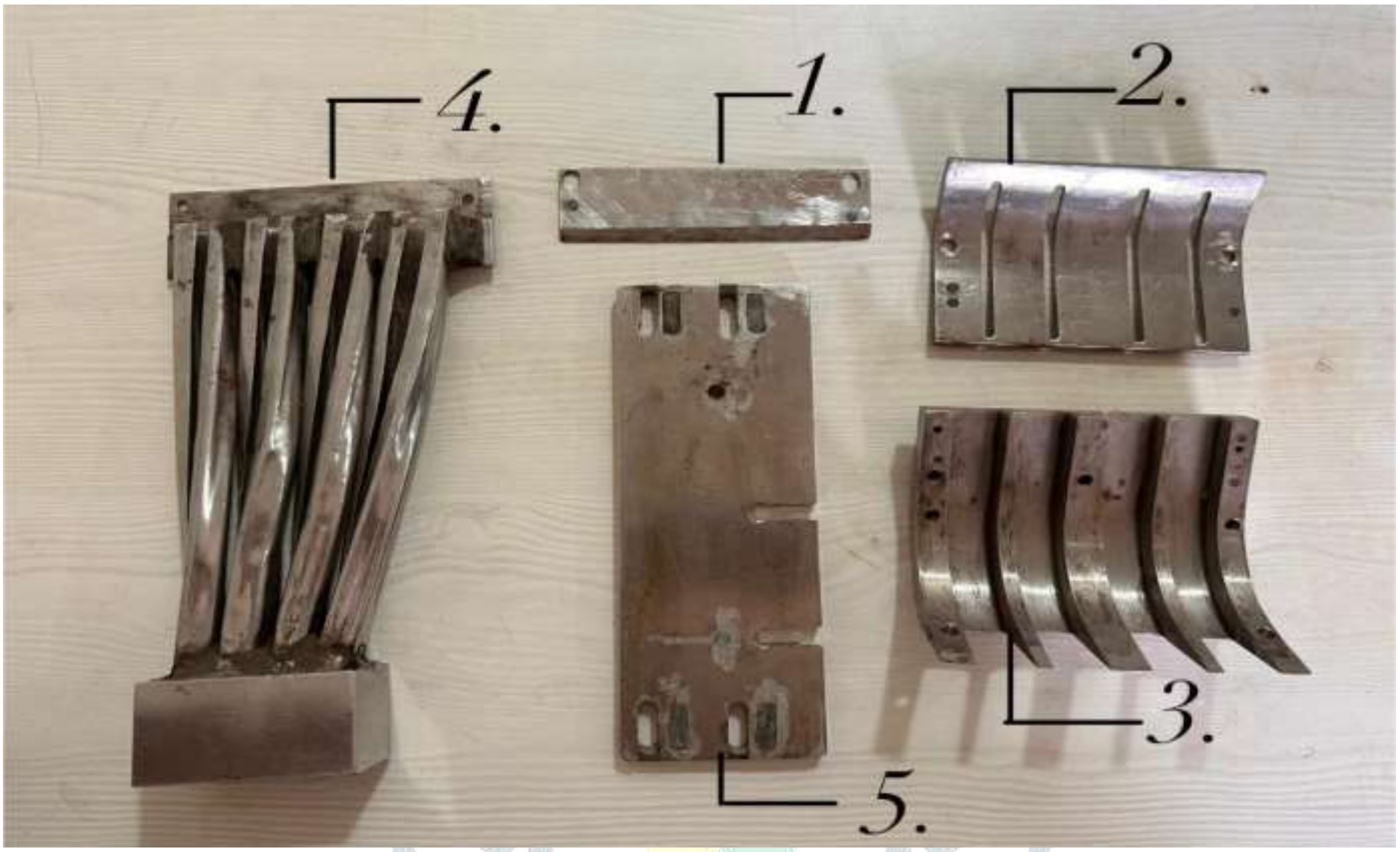

FIG 2. PARTS OF BYPASS DUCT FLOW

10.CFD ANALYSIS

Input Parameters:

1. **Inlet Conditions:**

Fuel Inlet Pressure: 220000 Pa
Inlet velocity: 300m/s (equivalent to Mach 0.5 at 823 K)
Working Medium: Ideal Gas
Outlet Conditions:
1. Body Outlet
Outlet Pressure: 101325 Pa (atmospheric pressure assumed).
Outlet temperature: 1400K
2.At enclosure
Outlet Pressure: 101325 Pa (atmospheric pressure assumed).
Outlet temperature: 300K

2. **Solver Type:** Pressure-based solver
Reason: Suitable for low-speed compressible and incompressible flows where pressure-velocity coupling is critical.

3. **Turbulence Model:**
k-ε (k-epsilon) Model
Why Used:
Widely Validated: One of the most commonly used turbulence models for engineering applications, particularly for flows with high Reynolds numbers.
Robust and Stable: Handles a variety of flow problems, including boundary layers, recirculation, and free shear flows.
Computationally Efficient: Balances accuracy with computational cost compared to more complex models like k-ω SST or LES.

4. **Working Fluid Properties:** Ideal gas

Reasons for Model Choices:

k-ε Model:

- Provides good predictions for: High-pressure turbulence in mixing zones (fuel and oxidizer interaction). Boundary layer development in the conical section and outlet zones.
- Less sensitive to near-wall modeling than k-ω SST, making it simpler to implement for your geometry.
-

Pressure-Based Solver:
Required to resolve the significant pressure gradients (1.5 MPa and 2 MPa to 101325 Pa) efficiently.
Best for cases where pressure coupling drives the flow, like in fuel injectors or jet engine components.

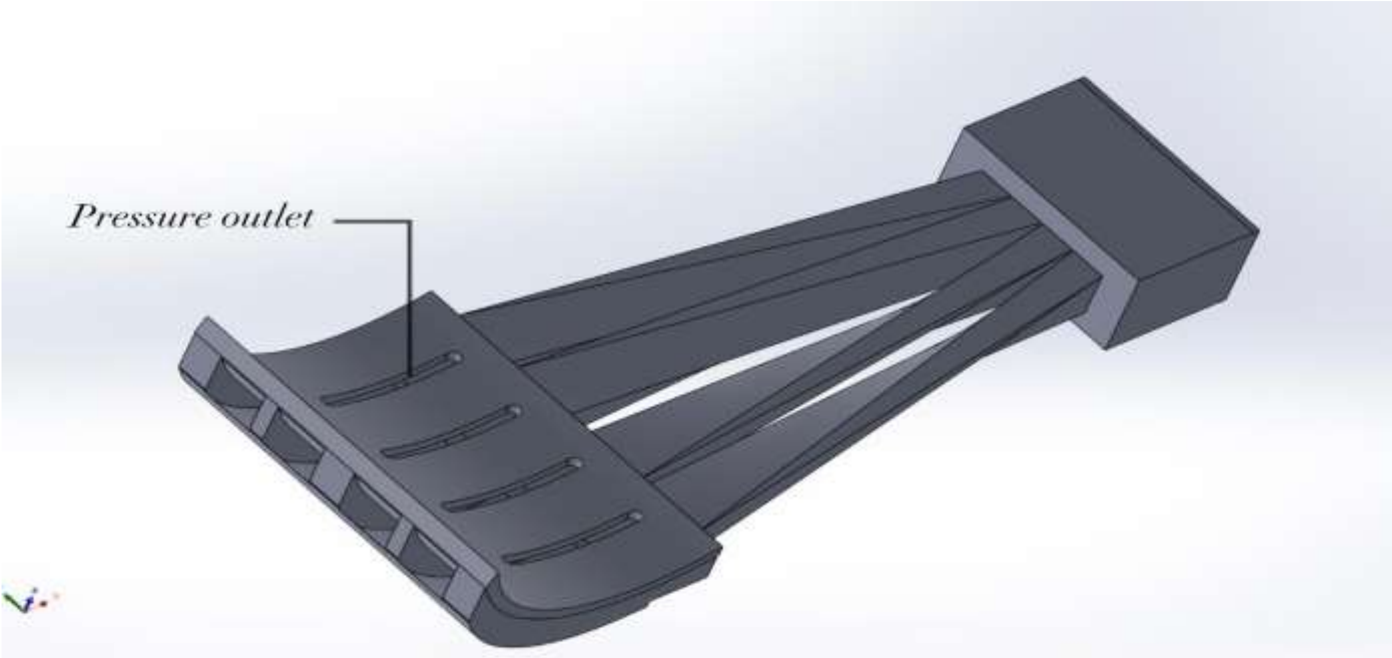

Fig 3. Body Outlet

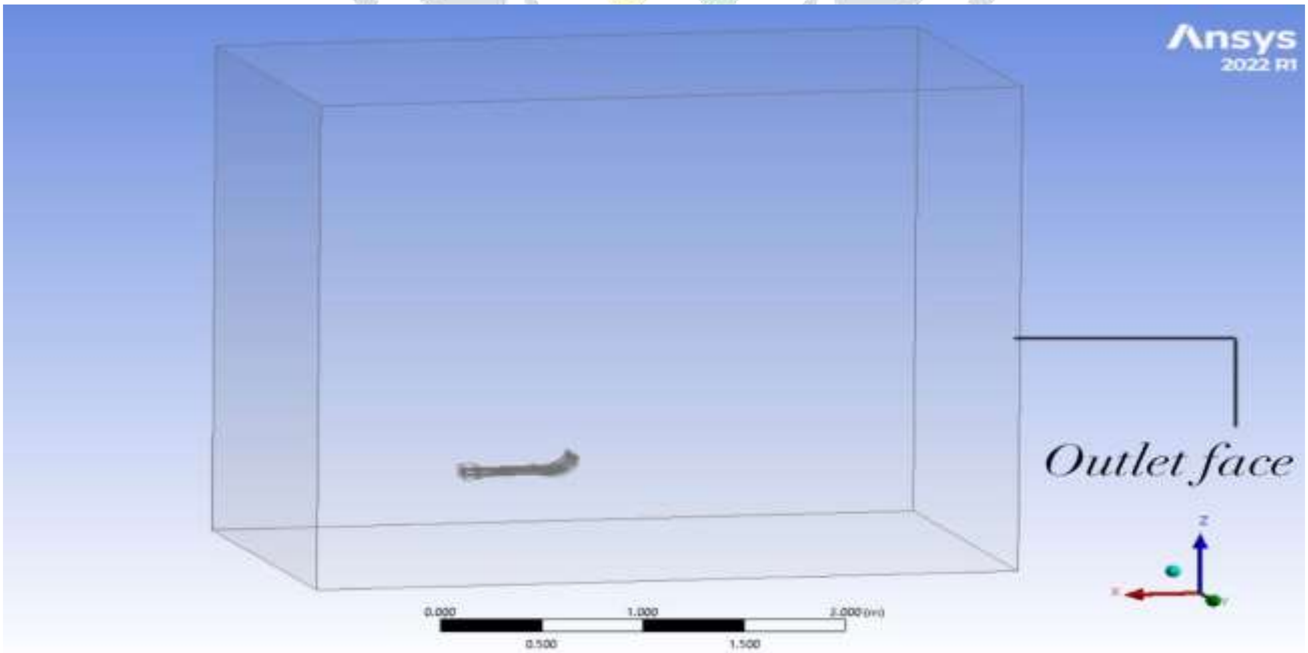

Fig 4. Enclosure Pressure Outlet

Results: 1.Contours

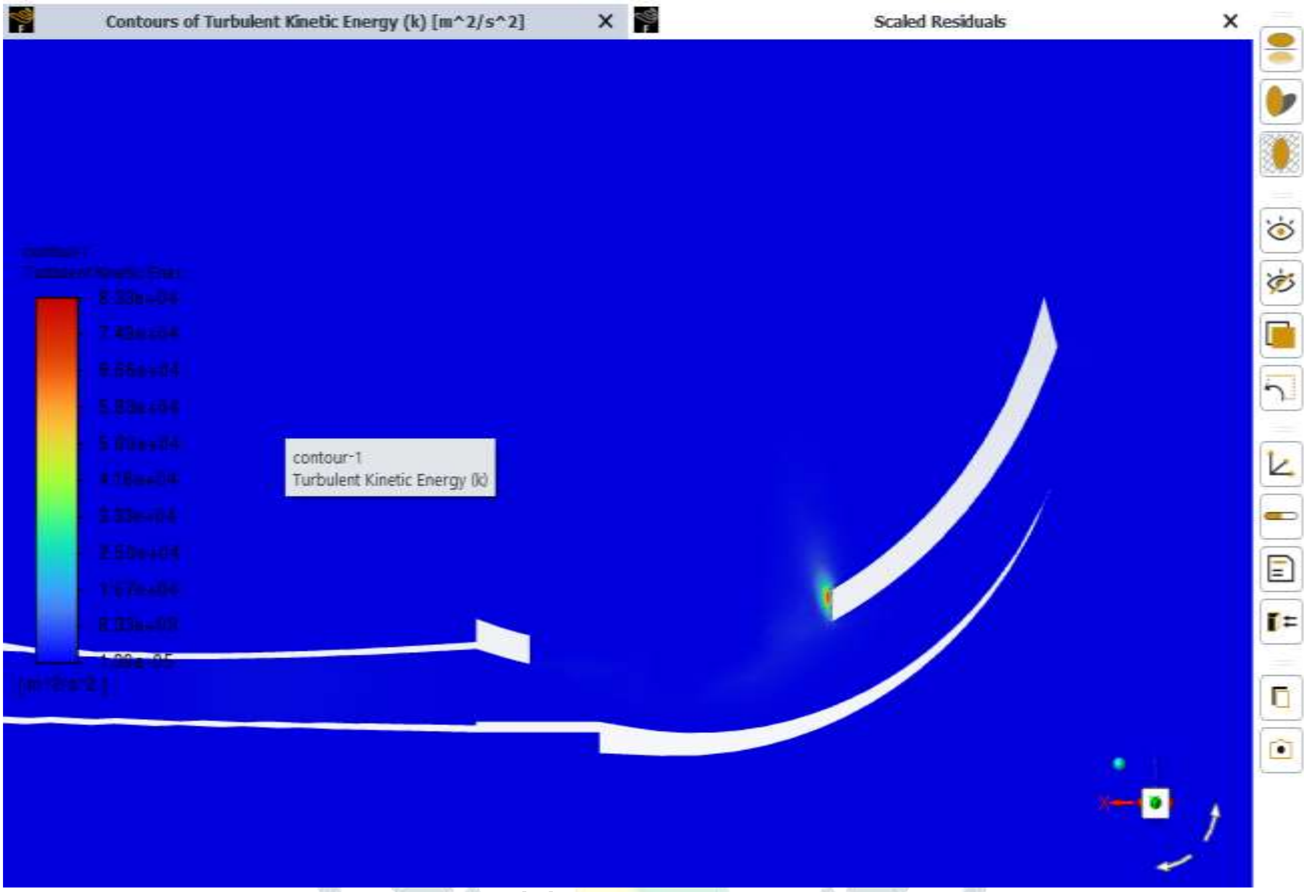

Fig 5. Turbulent Kinetic Energy Contour

Fig 6. Static Temperature Contour

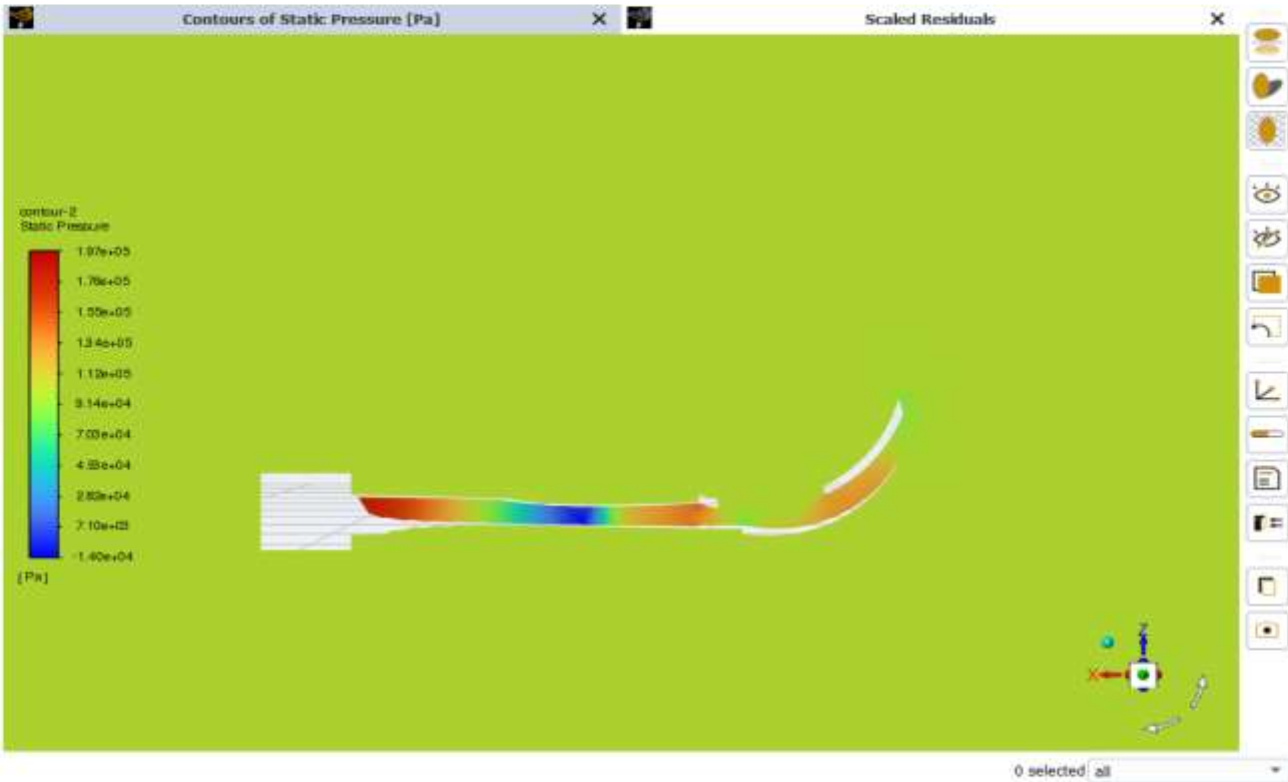

Fig 7. Static Pressure Contour

2. Pathlines

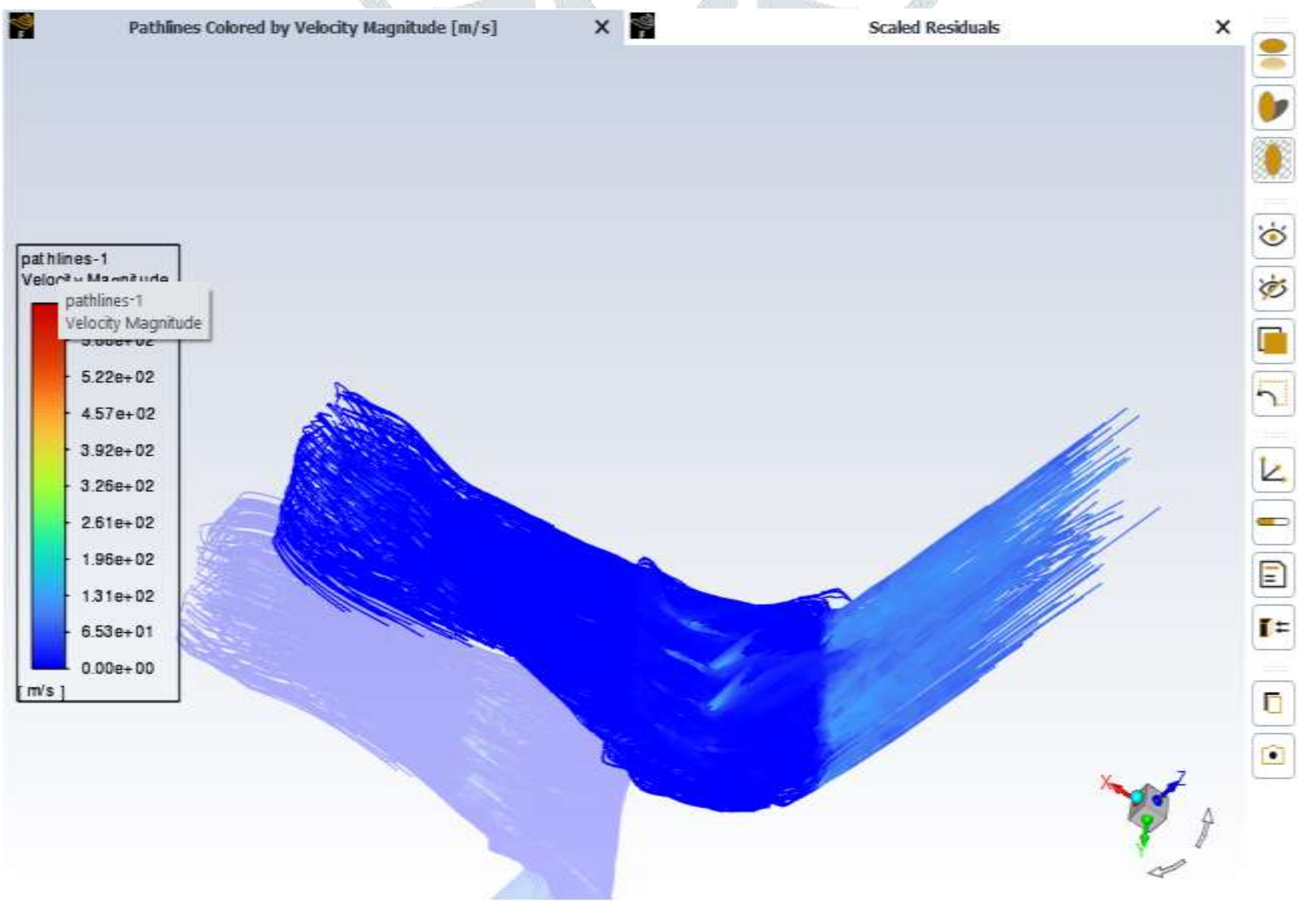

Fig 8. Velocity Magnitude Pathlines

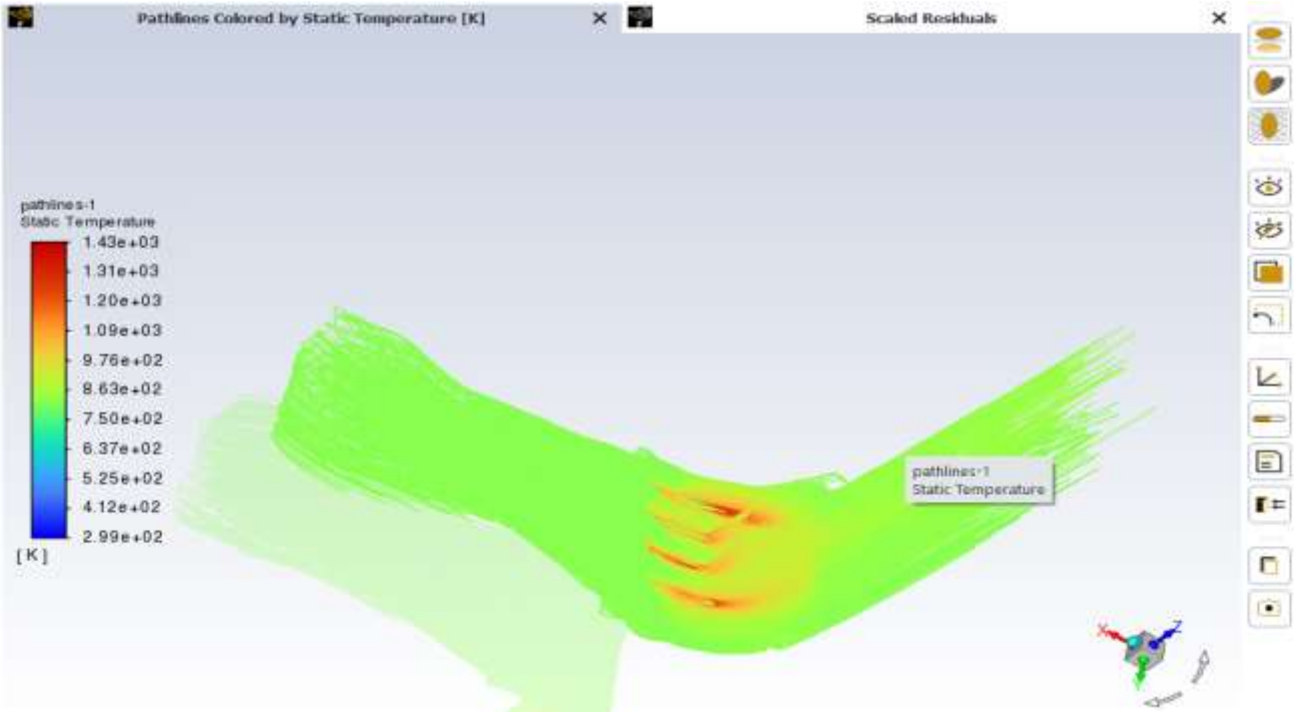

Fig 9. Static Temperature Pathlines

11. Conclusion:

This study successfully demonstrates the aerodynamic and thermal performance of a curved bypass secondary flow duct designed for aerospace propulsion systems. Using ANSYS Fluent, a compressible, steady-state CFD analysis was conducted under realistic operating conditions—namely, Mach 0.5 inlet flow at 823K—employing the standard k-ε turbulence model. The simulation results confirm that the duct effectively channels high-speed bypass air through a 120-degree curved path with minimal flow separation, acceptable pressure drop, and controlled temperature rise, aligning with theoretical expectations for secondary air systems in jet engines. The model's geometry, featuring a dual-outlet configuration and compact curvature, supports efficient integration in tight engine nacelle or casing structures while offering thermal shielding and passive cooling capabilities. Path line analysis, velocity contours, and static temperature gradients reveal that the duct maintains flow stability and directionality throughout, validating its use in both commercial and defense aerospace applications. In conclusion, the bypass flow duct model proves to be a viable and effective solution for managing high-speed secondary airflow in propulsion environments. The results provide a solid foundation for future work involving detailed optimization, thermal-structural coupling, or experimental validation to advance the design toward practical implementation in modern jet engines and hypersonic systems.

12. Reference:

- OpenAI. (2024). ChatGPT: Advanced language model for research and analysis.
- Prime Toolings Company- Precision engineering and design specifications for aerospace applications.
- S. Denton, A. Dawes, and A. Lefebvre. "Secondary Air Systems in Gas Turbine Engines."
- K. Khandelwal, S. K. Lele, and J. K. Eaton. "Design Considerations for Bypass Flow in Turbofan Engines."