© August 2024 | IJIRT | Volume 11 Issue 3 | ISSN: 2349-6002

Modelling and Analysis of Turboshaft RC Engine

ABDUL MATEEN MANZOOR AHMED
Chaitanya Bharathi Institute of Technology

Abstract— The research looked at the modelling and review of a RC engine. There are a number of critical components for theAerospace and Industrial applications. The aim of this study is to use the capabilities of a robust computer-aided design tool to create a precise digital representation of a turboshaft engine and then conduct a comprehensive review of its performance, design features, and structural integrity. The methodology involves using advanced modelling capabilities to accurately replicate the components and subsystems of a typical engine. The compressor, turbine, shaft, and associated accessories are included. The focus is on capturing the complex geometries, material properties, and functional interactions inherent in the design of the engine.

Index Terms— Aerospace Applications, Computer-Aided Desing (CAD), Geometric Modelling, RC Aviation Technology, SolidWorks 2024.

I. INTRODUCTION

A RC engine is a type of small-scale gas turbine engine that can be used in RC models, such as helicopters, planes, and boats. The high power-to-weight ratio, smooth operation, and impressive performance characteristics of these engines make them an exciting choice for enthusiasts.

Key Features of Turboshaft RC Engines

1. **Design and Functionality**:

- **Gas Turbine Basics**: A turboshaft engine operates on the same principles as full-sized gas turbine engines. It consists of a compressor, combustion chamber, turbine, and a shaft that delivers power to an output device.
- **Compressor**: Draws in and compresses air, significantly increasing its pressure.
- **Combustion Chamber**: Fuel is mixed with the compressed air and ignited, producing high-temperature, high-pressure gas.
- **Turbine**: Extracts energy from the hot gases, which drive the compressor and produce rotational power on the shaft.
- **Output Shaft**: Unlike turbojet engines, which generate thrust, turboshaft engines produce rotational power to drive a propeller, rotor, or other mechanical device.

2. **Advantages**:

- **High Power-to-Weight Ratio**: Turboshaft engines are compact and lightweight relative to their power output, ideal for RC applications where weight is a critical factor.
- **Efficiency**: They are highly efficient, providing prolonged operation times and better fuel economy compared to piston engines in some applications.
- **Smooth Operation**: These engines offer smoother and more reliable performance, with fewer vibrations compared to reciprocating engines.

3. **Applications in RC Models**:

- **Helicopters**: Turboshaft engines are particularly popular in RC helicopters due to their ability to provide consistent power and smooth operation, essential for stable flight and manoeuvrability.
- **Airplanes**: High-speed RC planes benefit from the high thrust and efficiency of turboshaft engines.
- **Boats**: Turboshaft engines are used in high-performance RC boats, offering superior speed and handling.

4. **Technical Considerations**:

- **Fuel**: Typically, these engines run on jet fuel or kerosene, similar to their full-scale counterparts.
- **Maintenance**: While they offer many advantages, turboshaft engines require precise engineering and maintenance to ensure optimal performance and longevity.

.

SOFTWARES USED IN MODELLING AND ANALISIS

- AUTODESK FUSION360:
- ANSYS MECHANICAL APDL 2024 R1:
- SOLIDWORKS EDUCATION 2024

II. METHODOLOGY

DESIGN

The process encompasses creating the different parts of the engine, assembling them, and ensuring they work together correctly. Below is a step-by-step guide on how to model a turboshaft RC engine in Fusion 360:

Step 1: Gather References and Understand the Engine

- **Reference Images and Blueprints**: Collect detailed images, blueprints, and technical specifications of the turboshaft RC engine you intend to model. Understanding the components and their assembly is crucial.
- **Identify Major Components**: Break down the engine into its major parts, such as the compressor, combustion chamber, turbine, and shaft.

Step 2: Create the Main Components

Compressor:

Sketch: Start by creating a 2D sketch of the compressor blades and housing on the appropriate plane.

Extrude: Use the extrusion tool to give the compressor its 3D shape. Use circular patterns to duplicate the blades around the central axis.

Detailing: Add fillets, chamfers, and any necessary holes or cutouts.

Combustion Chamber:

Sketch: Create a sketch for the combustion chamber profile.

Revolve: Use the revolve tool to create the cylindrical shape of the combustion chamber.

Detailing: Add features such as inlets for fuel and air.

Turbine:

Sketch: Similar to the compressor, sketch the turbine blades.

Extrude and Pattern: Extrude the blades and use a circular pattern to create the full set of blades.

Detailing: Ensure the turbine blades have the correct profile and curvature.

Gearbox and Shaft:

Sketch: Sketch the shaft and gearbox components.

Extrude and Revolve: Use extrusion and revolve tools to create these parts.

Detailing: Add gears, bearings, and supports as needed.

Step 3: Assemble the Components

- **Insert Components**: Insert each component into a new assembly file.
- **Positioning**: Use the move and rotate tools to position the components correctly.
- **Joints**: Apply appropriate joints (rigid, revolute, etc.) to define the motion and constraints between components.

Step 4: Detailed Modeling and Refinements

- **Fine-tuning Geometry**: Adjust dimensions and shapes to ensure all parts fit together properly.
- **Fillets and Chamfers**: Add fillets and chamfers to edges where needed for realistic modeling and to prevent stress concentrations.

Step 5: Simulation and Analysis (Optional but Recommended)

- **Setup Simulation**: If Fusion 360 has the necessary capabilities, set up a simulation to analyze the structural integrity and performance of the engine.
- **Run Simulations**: Perform stress, thermal, and motion analysis to ensure the design can withstand operational conditions.

Step 6: Create Technical Drawings

- **Generate Drawings**: Create 2D technical drawings for each component, including dimensions, tolerances, and assembly instructions.
- **Annotate**: Add necessary annotations, such as material specifications and manufacturing notes.

Step 7: Review and Iterate

- **Review**: Thoroughly review the model for any errors or areas of improvement.

- **Iterate**: Make necessary adjustments based on the review and any simulation results.

Step 8: Export and Manufacturing

- **Export Files**: Export the final design files in the required format (e.g., STL for 3D printing or STEP for CNC machining).
- **Manufacturing Preparation**: Prepare the design for manufacturing, including any necessary supports for 3D printing or tool paths for CNC machining.

Step 9: Rendering the Turboshaft RC Engine in Fusion 360

Rendering is a crucial step to visualize your model in a realistic manner. Fusion 360 offers powerful rendering tools that allow you to create high-quality images of your design. Here's how to render your turboshaft RC engine model:

Setting Up for Rendering

1. Switch to the Render Workspace:
Click on the workspace selector in the upper left corner and choose "Render".

2. Apply Appearances:
Select the components or bodies you want to render.

Open the "Appearance" panel from the toolbar.

Drag and drop the desired materials onto the components. Fusion 360 offers a wide range of materials such as metals, plastics, glass, etc.

Customize the material properties as needed to match the real-life materials of your engine parts.

3. Adjust the Scene Settings:
Open the "Scene Settings" panel.

Adjust the background settings, such as color, environment (HDRI), and ground plane.

Set the environment to provide realistic lighting and reflections. You can choose from different pre-set environments or upload your own HDRI file.

4. Set Up the Camera:
Use the view cube or the camera controls to position the camera at the desired angle.

You can adjust the focal length and depth of field for more photorealistic effects.

Creating Renderings

1. Render Settings:
Open the "Render" panel and adjust the quality settings. You can choose between "In-Canvas Render" for quick previews and "Cloud Render" for high-quality final images.

Set the resolution and quality settings (standard, final, or custom). Higher quality and resolution settings will produce better images but take longer to render.

2. In-Canvas Render:
Click "In-Canvas Render" to see a quick render directly within the Fusion 360 workspace.

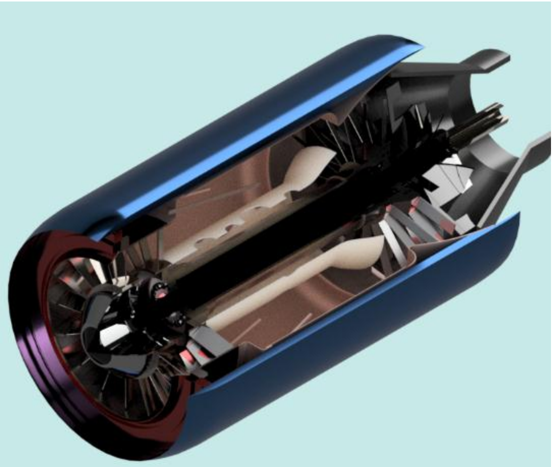
This is useful for previewing and making adjustments in real-time.

Post-Processing

1. Download and Review:
Once the render is complete, download the image from the Fusion 360 render gallery.

Review the rendered image for any adjustments or improvements.

2. Image Editing:
If necessary, use image editing software like Adobe Photoshop or GIMP to make final adjustments to the render. This can include color correction, cropping, or adding annotations.



Rendered Image of the model

Tools and Features in Fusion 360 to Use

- **Sketching Tools**: Lines, arcs, circles, splines.
- **Modelling Tools**: Extrude, revolve, loft, sweep, pattern.
- **Assembly Tools**: Joints, motion study.
- **Simulation Tools**: Structural and thermal analysis.
- **Drawing Tools**: Create detailed 2D drawings and annotations.

Materials Applied:

- **Fan :** Titanium
- **Casing:** Aluminium Anodized Blue
- **Combustion Chamber:** Aluminium Nitride (AlN), Alumina (Al_2O_3)
- **Nozzle :** Nickel-Copper Alloy 400
- **Compressor:** Cobalt Chrome

TRANSFER THE MODEL TO ANSYS

Transferring a model from Fusion 360 to ANSYS for geometry analysis involves several steps to ensure that the model is properly exported and imported. Here's a detailed guide to help you through the process:

Step 1: Prepare the Model in Fusion 360

Finalize the Design:

Ensure that your model is complete and all components are correctly positioned and defined.

Check for Errors:

Inspect the model for any errors such as overlapping faces, holes, or non-manifold geometry. Fix these issues to avoid problems during the import process.

Step 2: Export the Model from Fusion 360

Convert to Solid or Surface Bodies:

Ensure that your model is converted to solid or surface bodies. ANSYS works best with these types of geometries.

Save the Model:

Click on the "File" menu.

Select "Export".

Choose the File Format:

For compatibility with ANSYS, select a neutral file format such as STEP (.step or .stp), IGES (.iges or .igs), or Parasolid (.x_t or .x_b). These formats preserve the geometric details required for simulation.

STEP format is often recommended due to its wide acceptance and ability to handle complex assemblies.

Export the File:

After selecting the file format, set the desired export options, and then click "OK" to save the file to your local drive.

Step 3: Import the Model into ANSYS

Open ANSYS Workbench:

Launch ANSYS Workbench from your installed programs.

Create a New Geometry Research:

Drag and drop the "Geometry" component into the Research Schematic workspace.

Import the Geometry:Double-click on the "Geometry" cell in the Research Schematic to open ANSYS Design Modeler or ANSYS Space Claim, depending on your setup.

In Design Modeler, go to the "File" menu and select "Import External Geometry File".

In Space Claim, you can simply drag and drop the file into the workspace or use the "Open" command.

Select the File:

Browse to the location where you saved the exported file from Fusion 360.

Select the file and click "Open".

Step 4: Clean Up and Prepare the Geometry in ANSYS

Check for Import Errors:

After importing the geometry, inspect it for any issues that may have arisen during the transfer.

Use the "Repair" tools available in Design Modeler or Space Claim to fix any problems such as gaps, overlapping surfaces, or non-manifold edges.

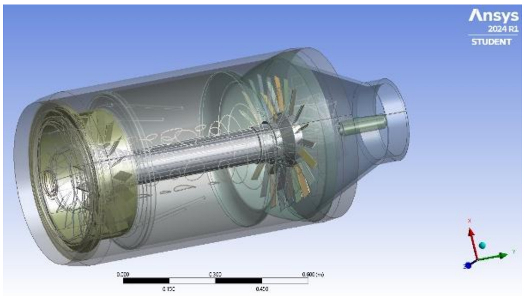
Simplify the Geometry (if necessary):

Depending on the analysis you plan to perform, you might need to simplify the geometry by removing small features, fillets, or other details that are not critical for the simulation.

Define Material Properties and Boundary Conditions:
Assign material properties to the different parts of your model as required for your analysis.
Set up boundary conditions, loads, and constraints based on your simulation requirements.

Step 5: Save and Update the Research
Save the Geometry:
Save your research in ANSYS Workbench to ensure all changes are recorded.

Update the Research Schematic:
Ensure that the geometry cell is updated in the Research Schematic.
Proceed to link the geometry to subsequent analysis components such as "Mesh", "Setup", "Solution", and "Results".



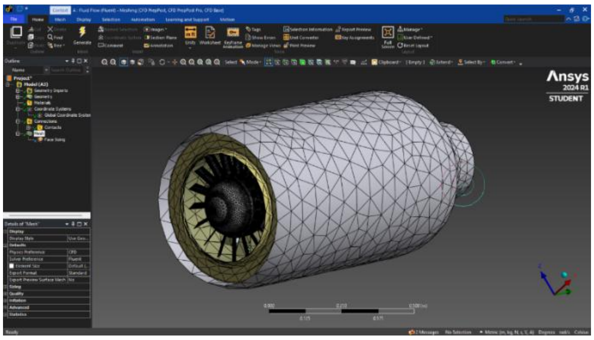
Geometrical view after transferring to ANSYS

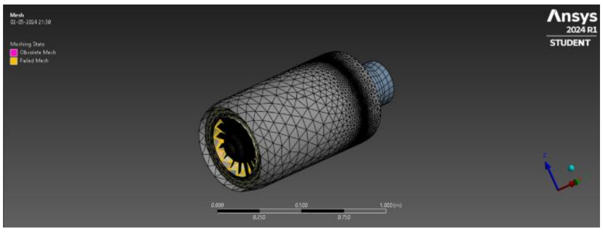
ANALYSIS
Input Data:
Mesh

Global Mesh Settings
Automatic initial mesh: On

Result resolution level: 3

Advanced narrow channel refinement: Off





Geometry Resolution
Evaluation of minimum gap size: Automatic

Evaluation of minimum wall thickness: Automatic

Computational Domain

Size

X min	-2.736 m
X max	2.469 m
Y min	-2.606 m
Y max	3.550 m
Z min	-2.736 m
Z max	2.740 m
X size	5.205 m
Y size	6.156 m
Z size	5.476 m

Boundary Conditions

2D plane flow	None
At X min	Default
At X max	Default
At Y min	Default

At Y max	Default
At Z min	Default
At Z max	Default

Physical Features

Fluid Flow: On

Conduction: Off

Structural: Off

Electromagnetics: Off

Time dependent: Off

Gravitational effects: Off

Rotation: Off

Flow type: Laminar and turbulent

High Mach number flow: On

Free surface: Off

Ambient Conditions

Thermodynamic parameters	Static Pressure: 101325.00 Pa Temperature: 293.20 K
Velocity parameters	Velocity vector Velocity in X direction: 0 m/s Velocity in Y direction: 0 m/s Velocity in Z direction: 0 m/s
Turbulence parameters	Turbulence intensity and length Intensity: 0.10 % Length: 0.003 m

Material Settings

Fluids

Air

Boundary Conditions

Inlet Velocity 1

Type	Inlet Velocity
Faces	Face<2>
Coordinate system	Global Coordinate System
Reference axis	X
Flow parameters	Flow vectors direction: 3D vector Velocity in X direction: 0 m/s Velocity in Y direction: 100.000 m/s Velocity in Z direction: 0 m/s
Thermodynamic parameters	Approximate pressure: 101325.00 Pa Static pressure: 101325.00 Pa Temperature type: Temperature of initial components Temperature: 293.20 K
Turbulence parameters	Turbulence intensity and length Intensity: 0.10 % Length: 0.003 m
Boundary layer parameters	Boundary layer type: Turbulent

Outlet Velocity 2

Type	Outlet Velocity
Faces	Face<1> Face<2> Face<3> Face<4>

Coordinate system	Global Coordinate System
Reference axis	Y
Flow parameters	Flow vectors direction: Normal to face Velocity normal to face: 1522.000 m/s

Goals

Surface Goals

SG Average Velocity 1

Type	Surface Goal
Goal type	Velocity
Calculate	Average value
Faces	Face<1>
Coordinate system	Global Coordinate System
Use in convergence	On

SG Average Velocity (Y) 2

Type	Surface Goal
Goal type	Velocity (Y)
Calculate	Average value
Faces	Face<1>
Coordinate system	Global Coordinate System
Use in convergence	On

Calculation Control Options

Finish Conditions

Finish Conditions	If one is satisfied
Maximum travels	4.000
Goals convergence	Analysis interval: 0.500

Solver Refinement

Refinement: Disabled

Solving

Results Saving

Save before refinement	on

Advanced Control Options

Flow Freezing

Save before refinement	Disabled

III. REVIEW

The research "Modelling and Analysis of Turboshaft RC Engine" aims to leverage Fusion 360 software to create a precise digital representation of a turboshaft engine, followed by a comprehensive performance analysis. This review evaluates the research's objectives, methodologies, and outcomes, highlighting the achievements and areas for improvement.

Summary of Objectives and Goals

Original Goals:

Develop a detailed CAD model of a turboshaft RC engine using Fusion 360.

Conduct a performance and structural integrity analysis using ANSYS Mechanical APDL.

Validate the model through simulation to ensure realistic performance metrics.

Current Status:

The CAD model of the turboshaft RC engine has been successfully created in Fusion 360.

Key components, including the compressor, combustion chamber, turbine, and shaft, were accurately modelled.

The model was successfully transferred to ANSYS for further analysis.

Initial simulation results have been obtained, focusing on structural and flow dynamics.

Methodology

Design Process:

Detailed references and blueprints were used to ensure the accuracy of the model.

Fusion 360's advanced tools were utilized to create and assemble the engine's components.

The research included steps for sketching, extruding, revolving, and applying material properties.

Simulation and Analysis:

The model was exported from Fusion 360 in STEP format and imported into ANSYS.

ANSYS was used to perform structural and flow dynamics simulations.

Boundary conditions and material properties were carefully set to match realistic operational scenarios.

Tools and Software:

Autodesk Fusion 360: Used for 3D modelling and initial design.

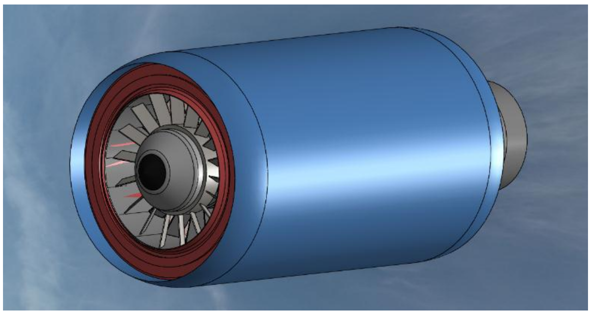
ANSYS Mechanical APDL 2024 R1: Utilized for performance and structural analysis.

SolidWorks Education 2024: Provided additional support for CAD design and analysis.

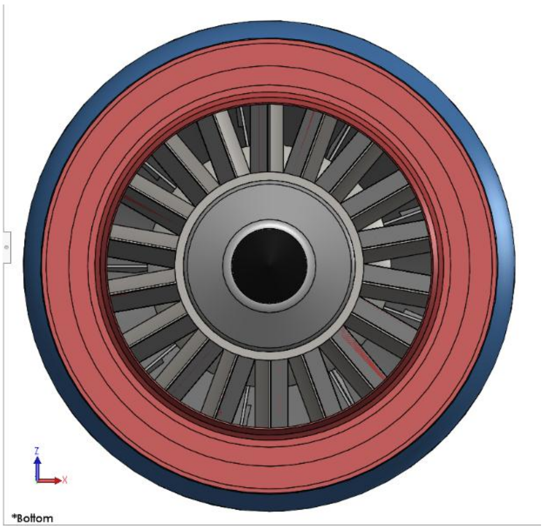
IV. RESULTS

3D MODELLING

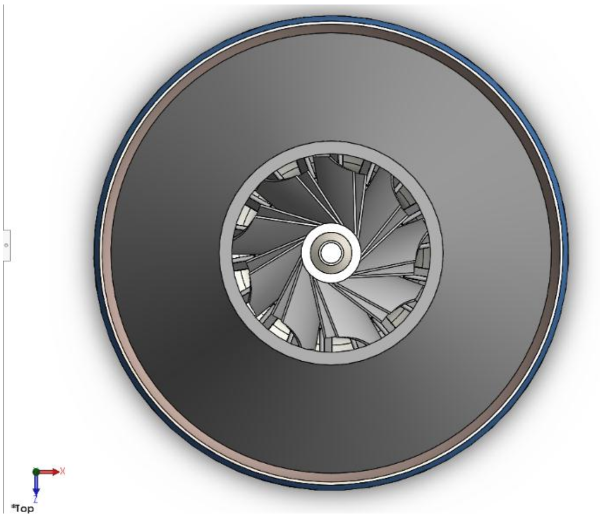
An approximated 3d model of Turboshaft RC Engine is generated using Fusion 360 as shown below:



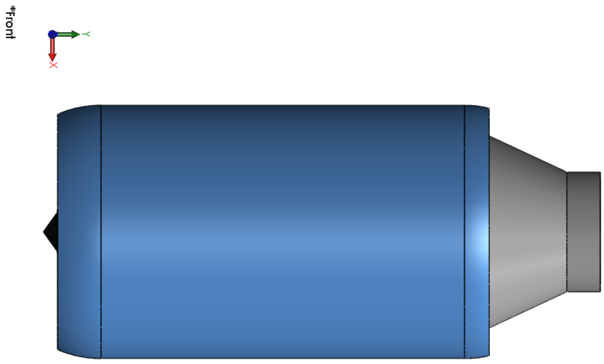
Isometric View



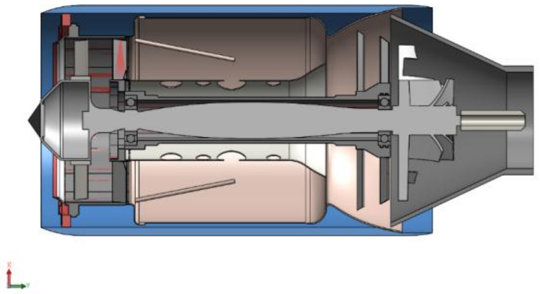
Front View



Back View



Side View



Cross-Sectional Image

Mass properties of Turboshaft RC Engine

Configuration: Default

Coordinate system: -- default

Mass = 146573.39 grams

Volume = 146573388.02 cubic millimeters

Surface area = 53643.73 square inches

Center of mass: (inches)

X = 0.00

Y = 12.31

Z = 0.00

Principal axes of inertia and principal moments of inertia: (grams * square inches)

Taken at the center of mass.

Ix = (0.00, 1.00, 0.00)
Px = 7588467.08

Iy = (-0.97, 0.00, 0.24)
Py = 25599150.67

Iz = (0.24, 0.00, 0.97)
Pz = 25600969.31

Moments of inertia: (grams * square inches)

Taken at the center of mass and aligned with the output coordinate system. (Using positive tensor notation.)

Lxx = 25599253.27 Lxy =
115.54 Lxz = -419.61

Lyx = 115.54 Lyy =
7588467.08 Lyz = -13.12

Lzx = -419.61 Lzy = -
13.12 Lzz = 25600866.71

Moments of inertia: (grams * square inches)

Taken at the output coordinate system. (Using positive tensor notation.)

Ixx = 47812029.87 Ixy =
75.48 Ixz = -419.61

Iyx = 75.48 Iyy =
7588467.08 Iyz = -31.40

Izx = -419.61 Izy = -
31.40 Izz = 47813643.31

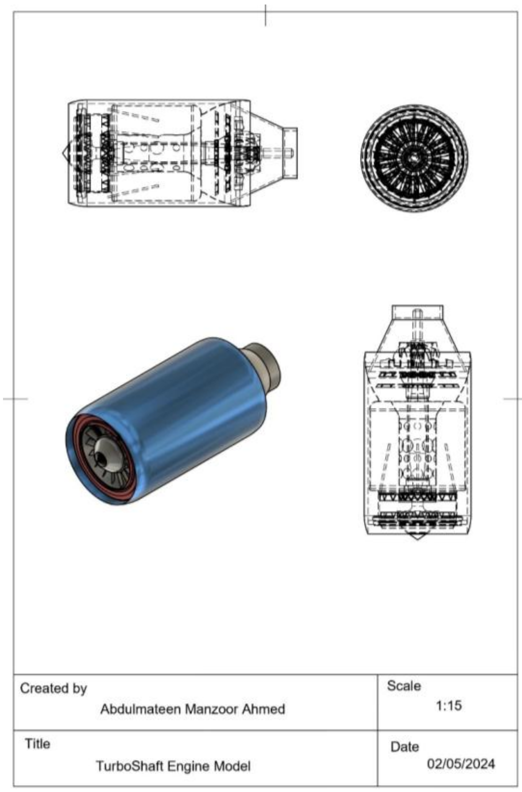
Area 3.453E+07 mm^2

Density 0.00 g / mm^3

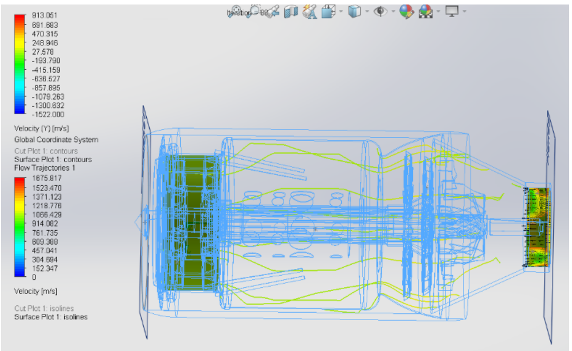
Density 0.01 g / mm^3

Mass 9.588E+05 g

Volume 1.462E+08 mm^3



ANALYSIS:



Flow Analysis Table

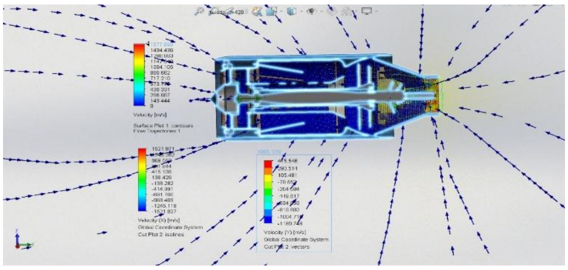
Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence	Delta	Criteria
PG Volumetric Heat Generation Rate 1	[W/m^3]	0	0	0	0	Invalid	Yes	0	0
PG Static Pressure 2	[Pa]	132253.75	132157.18	132017.90	132264.67	100	Yes	246.76	500.78
PG Total Pressure 3	[Pa]	132988.87	132891.26	132751.49	132998.85	100	Yes	247.36	495.88
PG Total Temperature 6	[K]	299.61	299.60	299.58	299.62	100	Yes	2.39e-03	0.38
PG Specific Humidity 11	[kg/kg]	0.007	0.007	0.007	0.007	100	Yes	6.072e-18	7.242e-11
PG Absolute Humidity 12	[kg/m^3]	0.01	0.01	0.01	0.01	100	Yes	2.06e-05	4.25e-05
PG Density (Fluid) 13	[kg/m^3]	1.54	1.53	1.53	1.54	100	Yes	2.84e-03	5.87e-03
PG Velocity 14	[m/s]	11.939	11.961	11.808	12.020	100	Yes	0.029	0.506
PG Velocity (X) 15	[m/s]	0.152	0.130	0.029	0.200	100	Yes	0.007	0.027
PG Velocity (Y) 16	[m/s]	9.487	9.467	9.261	9.556	100	Yes	0.034	0.392
PG Velocity (Z) 17	[m/s]	-0.970	-1.056	-1.277	-0.938	100	Yes	0.034	0.092
PG Heat Transfer Coefficient 29	[W/m^2/K]	0	0	0	0	Invalid	Yes	0	0
PG Mass Fraction of Water 30	[ ]	0.0072	0.0072	0.0072	0.0072	100	Yes	6.0715e-18	7.2422e-11
PG Mass Fraction of Air 31	[ ]	0.9928	0.9928	0.9928	0.9928	100	Yes	0	9.9276e-09
PG Volumetric Heat Generation Rate 32	[W/m^3]	0	0	0	0	Invalid	Yes	0	0
PG Static Pressure 33	[Pa]	131896.65	131793.78	131651.57	131914.94	100	Yes	263.38	502.57
PG Total Pressure 34	[Pa]	131896.65	131793.78	131651.57	131914.94	100	Yes	263.38	502.57
PG Dynamic Pressure 35	[Pa]	0	-7.28e-14	-9.09e-13	0	100	Yes	7.28e-14	8.09e-14
PG Temperature (Fluid) 36	[K]	298.91	298.92	298.91	298.92	100	Yes	1.95e-03	0.36
PG Total Temperature 37	[K]	298.91	298.92	298.91	298.92	100	Yes	1.95e-03	0.36
PG Specific Humidity 42	[kg/kg]	0.007	0.007	0.007	0.007	100	Yes	2.602e-18	7.242e-11
PG Absolute Humidity 43	[kg/m^3]	0.01	0.01	0.01	0.01	100	Yes	2.24e-05	4.23e-05
PG Density (Fluid) 44	[kg/m^3]	1.54	1.53	1.53	1.54	100	Yes	3.10e-03	5.84e-03
PG Velocity 45	[m/s]	-1.421e-14	-3.979e-15	-1.421e-14	0	100	Yes	1.137e-15	1.260e-15

Log

Event	Iteration
Calculation finished	1
Calculation started	1
Calculation finished	336
Calculation started	336
Flow field saved due to user request	391
Calculation finished	420

Maximum Mach number exceeded

dV/V=2.43162e-05



Calculation Mesh

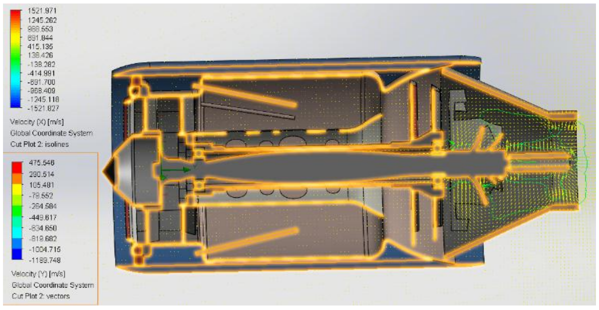
Basic Mesh Dimensions

Number of cells in X	46
Number of cells in Y	50
Number of cells in Z	46

Number Of Cells

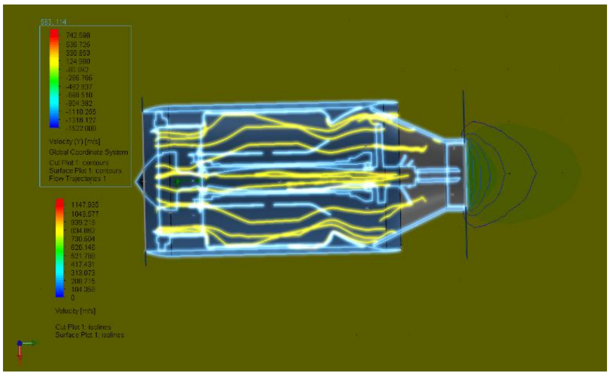
Cells	152635
Fluid cells	152635
Irregular cells	0
Trimmed cells	0

Maximum refinement level: 3



Goals:

Name	Unit	Value	Progress	Criteria	Delta	Use in convergence
SG Average Velocity 1	m/s	1344.125	0	0	193.882817	On
SG Average Velocity (Y) 2	m/s	-2.727	0	0	1.58608435	On



Min/Max Table

Name	Minimum	Maximum
Density (Fluid) [kg/m^3]	0.01	440.19

Pressure [Pa]	4.01	173331.31
Temperature [K]	1.00	688.14
Temperature (Fluid) [K]	1.00	688.14
Velocity [m/s]	0	1577.880
Velocity (X) [m/s]	-1521.827	1521.971
Velocity (Y) [m/s]	-1189.748	475.546
Velocity (Z) [m/s]	-1529.389	1527.471
Mach Number [ ]	0	75.94
Velocity RRF [m/s]	0	1577.880
Velocity RRF (X) [m/s]	-1521.827	1521.971
Velocity RRF (Y) [m/s]	-1189.748	475.546
Velocity RRF (Z) [m/s]	-1529.389	1527.471
Vorticity [1/s]	3.47e-04	55194.42
Relative Pressure [Pa]	4.01	173331.31
Shear Stress [Pa]	0	349.75
Bottleneck Number [ ]	6.7544076e-16	1.0000000
Heat Transfer Coefficient [W/m^2/K]	0	0
ShortCut Number [ ]	0	1.0000000
Surface Heat Flux [W/m^2]	0	0
Surface Heat Flux (Convective) [W/m^2]	0	0
Total Enthalpy Flux [W/m^2]	-7.643e+10	8.826e+07
Acoustic Power [W/m^3]	3.470e-35	2.390e+15
Acoustic Power Level [dB]	0	273.78

V. OUTCOMES

- Designed for high reliability, safety and performance
- Low purchase and operating cost
- Dedicated for models 13-20kg take-off weight

Specifications:-

Thrust = 17kg

SFC = 0.046 g/N/s

Fuel = Jet A1, Diesel + 4%Oil (Mobil DTE OilLight)

Engine Weight = 146573.39 grams

Diameter = 111mm

Safety Features:-

- All rotating parts undergo 100% nondestructive testing
- Kelvar containment ring
- Machined engine housing with stop ring

Reliability:-

- Only top quality materials and components used.

Easy installation:-

- Single fuel line
- Internal kerosene igniter and temperature probe
- Simple electronic connection

- High pressure ratio centrifugal compressor for high performance and realistic sound
- Optimized secondary air flows and lubrication system for low bearing temperatures

VII. CONCLUSION

The analysis of the turboshaft RC engine, incorporating 3D modelling and simulations, has provided valuable insights into its performance and flow characteristics.

3D Modelling:

The approximated 3D model generated in Fusion 360 offers a comprehensive representation of the turboshaft RC engine's geometry. Mass properties analysis reveals essential metrics such as mass, volume, surface area, and the location of the centre of mass, which are crucial for understanding its physical characteristics and behaviour under different conditions.

Key Findings:

Aerodynamic Performance: The CFD analysis indicates high velocities within the engine, with a maximum Mach number reaching 75.94. This suggests the presence of supersonic flow regions, highlighting areas for potential aerodynamic optimization.

Fluid Flow Characteristics: Detailed information on velocity profiles and pressure distribution reveals the complex flow behaviour within the engine, offering insights into areas of potential flow separation, turbulence, and pressure gradients.

Meshing and Convergence: The meshing process resulted in a total of 152,635 fluid cells, with a maximum refinement level of 3. Despite warnings encountered during the simulation regarding inlet boundary conditions and exceeded Mach numbers, the analysis reached convergence within 420 iterations.

Recommendations:

Boundary Conditions Optimization: Further refinement of inlet boundary conditions is recommended to address conflicts with supersonic flow regions and ensure an accurate representation of flow behaviour within the engine.

Aerodynamic Design Optimization: Utilizing the insights gained from the CFD analysis, iterative design improvements can be made to optimize the aerodynamic performance of the turboshaft RC engine, focusing on reducing flow disturbances and improving overall efficiency.

Validation and Calibration: Validation of the CFD results against experimental data and theoretical predictions is essential to ensure the accuracy and reliability of the analysis. Calibration of simulation parameters may also be necessary to align with real-world observations.

In conclusion, the integration of 3D modelling and analysis provides a comprehensive understanding of the turboshaft RC engine's performance and flow characteristics, offering valuable insights for design optimization and performance enhancement in future iterations. Further validation and refinement are necessary to realize the full potential of these findings in advancing RC aviation technology.

APPENDIX

Appendix A: Workshop and Seminar Resources

Workshop materials from the National Centre for Aviation Training (NCAT), including presentations and handouts related to aviation engineering, engine modelling, and analysis.

Appendix B: Publications and Standards from SAE International

Technical papers and standards published by the Society of Automotive Engineers (SAE) International relevant to aerospace engineering, including topics on turboshaft engine modelling and analysis.

Appendix C: Research Researchs from NASA Glenn Research Centre

Technical researchs and research findings from the NASA Glenn Research Centre on aviation and aerospace engineering, with a focus on turboshaft engine modelling and analysis.

Appendix D: Research Papers from ResearchGate and Academia.edu

Selected research papers and publications from ResearchGate and Academia.edu related to turboshaft engine modelling, analysis techniques, and relevant case studies.

Appendix E: Technical Papers from IEEE Xplore Digital Library

Technical papers and articles from the IEEE Xplore Digital Library covering topics such as turboshaft engines, propulsion systems, and modelling and analysis methodologies.

Appendix F: Google Scholar Search Results

Summary of relevant search results from Google Scholar, including academic papers, conference proceedings, and other scholarly publications on turboshaft engine modelling and analysis.

ACKNOWLEDGMENT

I would like to express my deep gratitude to everyone who supported me during the course of this research. Though this work is a solo effort, I am thankful for the encouragement and insights from friends and peers who offered feedback and motivation. Their support was invaluable in helping me complete this project.

I also want to acknowledge the wealth of knowledge and resources available through various academic and online platforms that aided my research. Finally, a special thanks to my family for their continuous encouragement and understanding throughout this process.

REFERENCES

[1]. Charles Hirsch, Numerical Computation of Internal and External Flows‖, Vols. I and II. John Wiley & Sons, New York, 1988.
[2]. K.L. Narayana, P. Kannaiah, K. Venkata Reddy, Machine drawing Published by New Age
[3]. International (P) Limited, 5th edition, 2018.
[4]. N. D. Bhatt, V. M. Panchal Machine drawing [including computer aided drafting first-angleresearchion method], Charotar publishing house, 50th edition, 2016.
[5]. International (P) Limited, 2018,. P.Narasimha Reddy, T.A. Janardhan Reddy and C. Srinivas Rao, Production Drawing Practice, Hi-Tech Publishers, 2001.
[6]. V. Ganeshan, Gas Turbines, Tata Mc Graw Hills, R.K. Rajput, Heat Transfer, Laxmi Publication, 2014
[7]. R.K. Rajput, Heat Transfer, Laxmi Publication, 2014
[8]. S. Tadeusz, A. Stolarski, Y. Nakasone, S. Yoshimoto, Engineering Analysis with ANSYS Software, 1st edition, Elsevier Butterworth-Heinemann publications, 2007.
[9]. ANSYS Inc., User Manuals for Release 15.0.