Gas Turbofan Compressor Design

Gary Go

Andrew Kim

Haoran (Harry) Wei

ME 407: Computational Fluid Dynamics

Professor Scott Bondi

The Cooper Union for the Advancement of Science and Art

11 May 2020

Table of Contents

Introduction

Overview

An axial compressor is an essential part of a turbofan. Axial compressors pressurize air to achieve higher fuel efficiency in the combustion chamber. An axial compressor usually consists of a series of rotors and stators in multiple stages. The rotos drive air into subsequent stators, which then turn kinetic energy into pressure. An example of a turbofan is shown below with the compressor design in green and purple on the left side before the yellow combustion chamber.

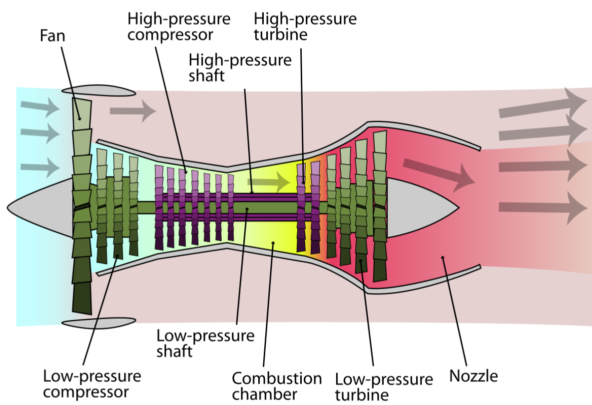


Figure 1. An example diagram of a turbofan courtesy of wikimedia.

Design and Modelling Plan

This project has a goal to design a compressor, and prove with computational fluids dynamics simulation that the design meets a list of criteria listed in the subsequent Design Requirements section of this report. The following steps were followed to obtain the design and prove its feasibility.

Model Description	Model Objective
Fine Mesh One Rotor and Stator Inviscid Model. Simulate at 40,000 ft and sea level	Improve Design and Maximize Compression Ratio; justify result of loose mesh model
Fine Mesh One Rotor and Stator Turbulent Model. Simulate at 40,000 ft and sea level	Observe impact of turbulence, justify use of inviscid model in loose mesh simulation
Coarse Mesh 4 Stages Model. Simulate at 40,000 ft and sea level	Extract pressure ratio for each stage to extrapolate total stages required

Table 1. Plan for Simulation Model Selection

After analyzing the design criteria, we estimated using hand calculation that the compressor might take up to 20 stages. Due to limited computation power, it is impractical to simulate all stages, thus we simulate a 4 stage compressor model, then extrapolate the number of stages required using results obtained from the 4 stage model.

The resolution of the 4 stage model was still limited, thus the geometry was further reduced to one stage, which consists of one rotor and one stator. Two simulations were run on the one stage geometry at relatively high resolutions, using inviscid and turbulent methods respectively. Each simulation was run twice, one using air properties at 40,000 ft and another using properties of air at sea level.

Design Requirements and Assumptions

Design Constraints

Table 2, below, shows the extreme conditions and performance requirements of the gas turbofan compressor.

Description	Requirement
Maximum Altitude	40,000 ft
Maximum Fan Speed	50,000 rpm
Outer Compressor Diameter	Between 4 ft to 6 ft
Minimum Pressure Ratio	20:1
Exit Condition	Exhaust tube matching the exit diameter

Table 2. Design Requirements for the gas turbofan compressor

Design Assumptions and Preliminary Calculations

- The engine is assumed to be on a test stand with entrance conditions at subsonic speeds.
- 4 ft for engine outer diameter for lower element count and therefore quicker iterations
  - Also, a smaller diameter can have slightly higher RPM before blade tip tangential velocities become supersonic.
- The air entering at sea level has properties of air in the FLUENT Database as follows:
  - Density: 0.07645 lb/ft^3
  - Viscosity: 1.202 X 10^{-5} lbm/ft-s
  - Thermal Conductivity: 0.01399 btu/h-ft-°R
  - Specific Heat: 0.2404 btu/lb-°R
- The air entering at an altitude of 40,000 ft is assumed to have these properties:

Air Properties at 40,000 ft Calculations

Pressure:

$$p = p_0 \left(1 - \frac{L\,h}{T_0}\right)^{\frac{g\,M}{R_0\,L}}$$

Assume the following properties of the variables above:

$$p_0 = 101325 \text{ Pa}$$

$$L_0 = 0.00976 \text{ K/m}$$

$$h = 40000 \text{ ft} = 12192 \text{ m}$$

$$T_0 = 288.16 \text{ K}$$

$$g = 9.80665 \ \frac{\text{m}}{\text{s}^2}$$

$$M = 0.02897 \ \frac{\text{kg}}{\text{mol}}$$

$$R_0 = 8.31446 \ \frac{\text{J}}{\text{mol K}}$$

$$p_{4000} = 18649 \text{ Pa} = 2.7 \text{ psi}$$

Temperature:

$$T_{4000} = 216.65 \text{ K or -69.7 deg F}$$

Timestep Calculations

Time step sizes are calculated using the following equations.

$$\text{Timestep} \quad \text{Size} = \left(\frac{\text{360 degrees}}{\text{Blade \#}}\right)\left(\frac{1}{\text{15 parts per timestep}}\right)\left(\frac{1}{\text{rotationspeed in degrees}}\right)$$
$$\text{Timestep} \quad \text{Size} = \left(\frac{\text{360 degrees}}{25}\right)\left(\frac{1}{\text{15 parts per timestep}}\right) = \frac{0.96}{\text{rotation speed}}$$

In order to run the model accurately at 10,000 rpm, the model was run at lower speeds for about 100 timesteps with varying timestep sizes as shown in the table below. The model was run for the total 300 timesteps at the full 10,000 rpm.

Rotational Speed (rpm)	Timestep Size (s)
500	3.20e-4
1500	1.07e-4
3000	5.33e-5
4000	4.00e-5
5000	3.21e-5
7000	2.29e-5
9000	1.78e-5
10000	1.61e-5

Table 3. Timestep size and RPM relationship

Inviscid Single Stage Design

The first design modeled the pressure increase through a single rotor-stator pair. The goal of this design was to compare the inviscid model to the turbulent model which would be run next. The inviscid model runs more efficiently while ignoring the effects of turbulence. This model was run in order to see if future models should be run using the inviscid model.

Geometry and Mesh

The blades for the model were designed using ANSYS BladeGen. The following figures show the design of rotor 1 and stator 1 blades.

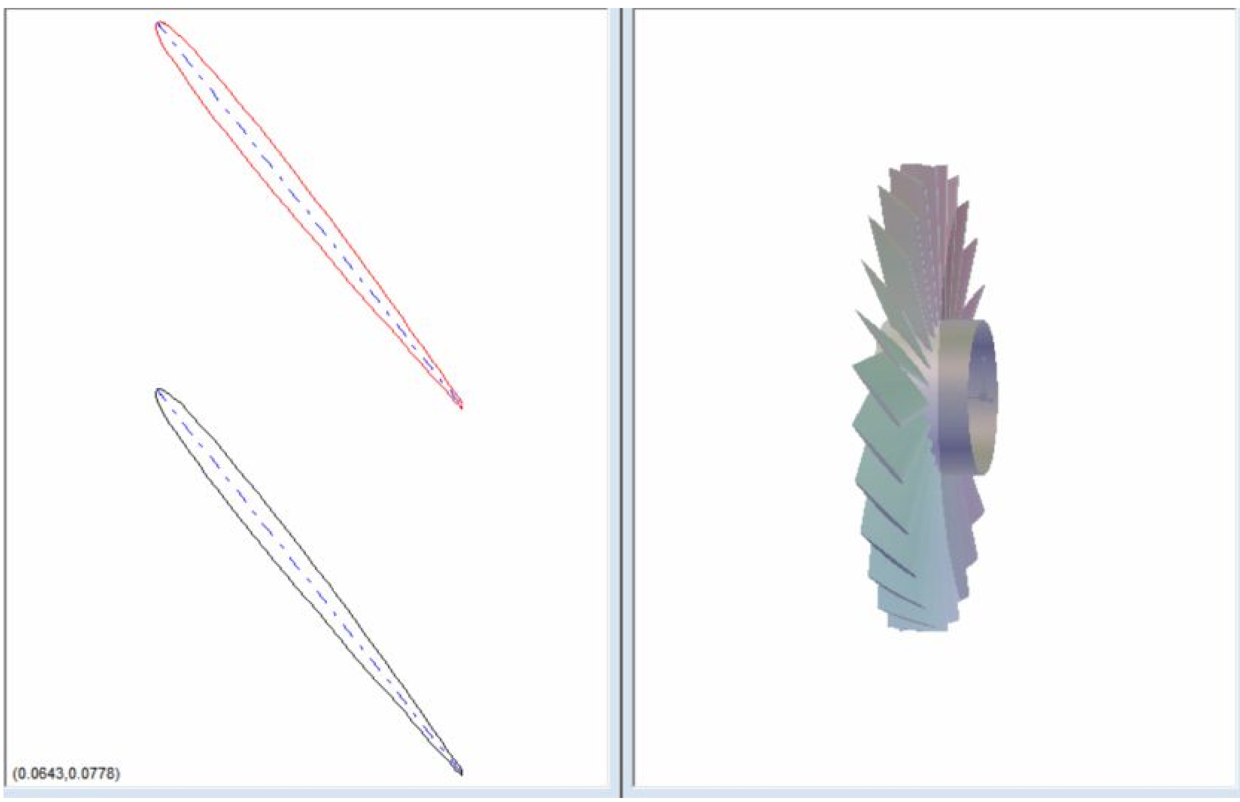


Figure 2. Rotor 1 for Inviscid Model

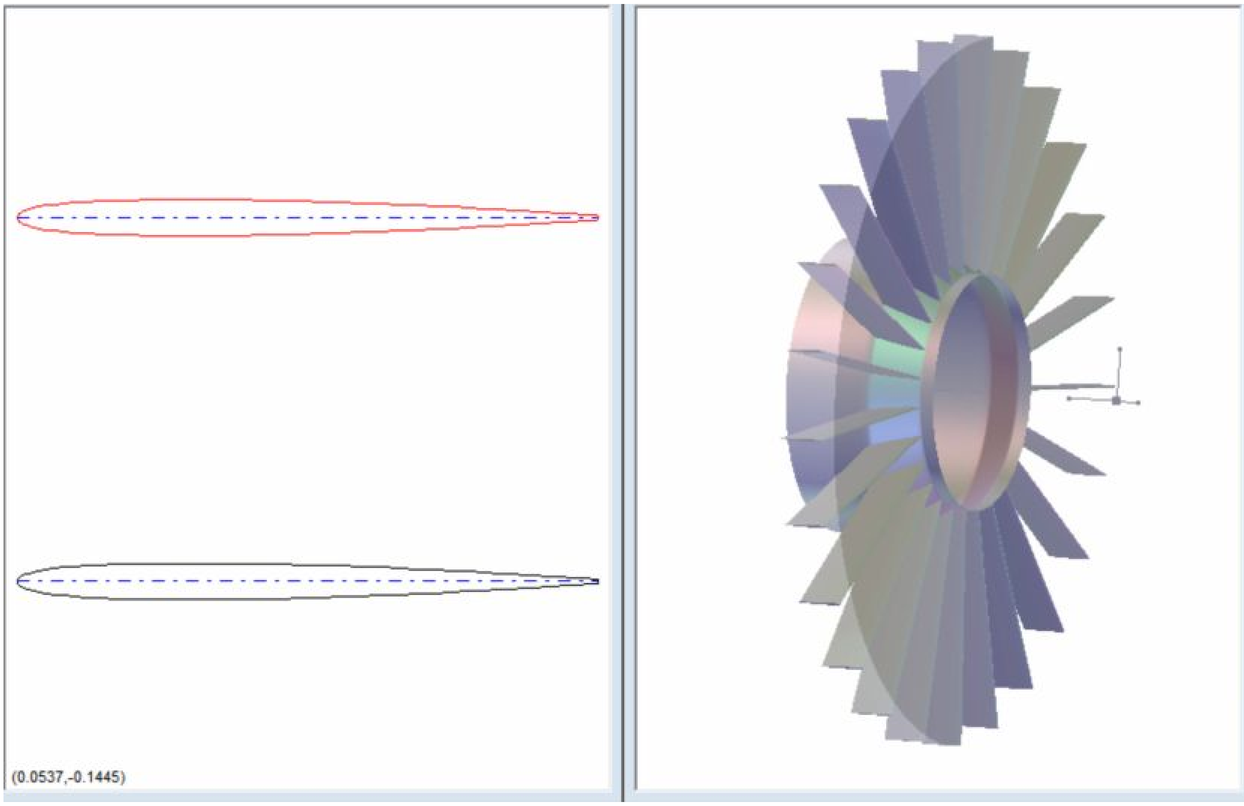


Figure 2. Stator 1 for Inviscid Model

The following parameters generate the rotor and stator for the inviscid model using bladegen.

	Rotor 1	Stator 1
Model start	0 in	10 in
Model end	10 in	20 in
Blade positions	4 - 9 in	11 - 14 in
Blade length (horizontal)	5 in	3 in
Blade Angle	45 deg	0 deg
Blade inner radius	8 in - 8 in	8 in - 10 in
Blade outer radius	24 in - 24 in	24 in -24 in

Table 4. Bladegen parameters for Inviscid Model

The geometry and mesh of the single stage model is shown below

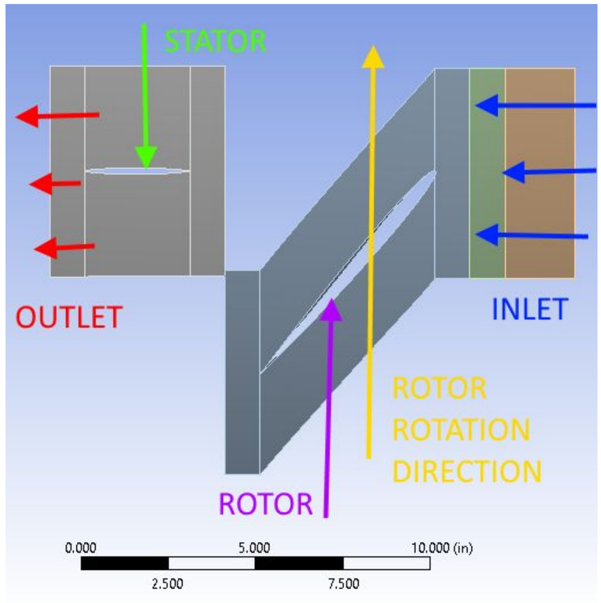


Figure 3. Geometry Single Stage Model

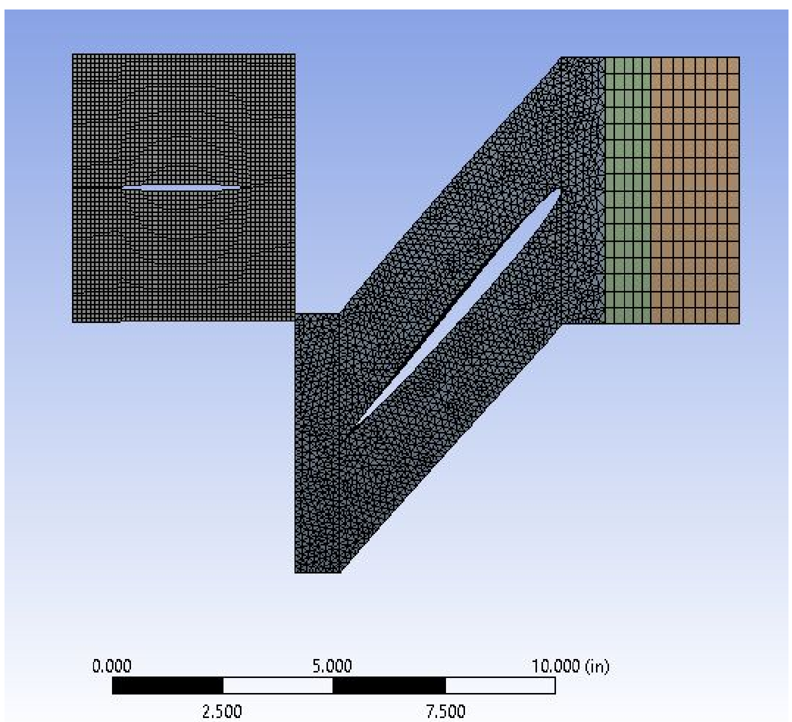


Figure 4.Mesh Single Stage Model

The following table details the parameters used for meshing.

Property	**Details**
Inlet Sizing	0.25 in
Adaptive Sizing	0.2 in
Rotor Blade Region Sizing	0.15 in
Rotor Method	Automatic
Stator Blade Region Sizing	0.10 in
Stator Method	Multizone
Outlet Sizing	0.25 in

Table 5. Mesh Parameters for Single Stage Model

Boundary Conditions & Solver Setting

The following two tables list boundary conditions and solver settings for the single stage model.

Property	**Detail**
Turbulence Model	Inviscid
Air Density	Ideal Gas Model
40,000 ft Operating Pressure	2.7 psi
Sea Level Operating Pressure	14.7 psi
40,000 ft Air Temperature	-69.7°F
Sea Level Air Temperature	77°F
Rotational Velocity	10,000 RPM

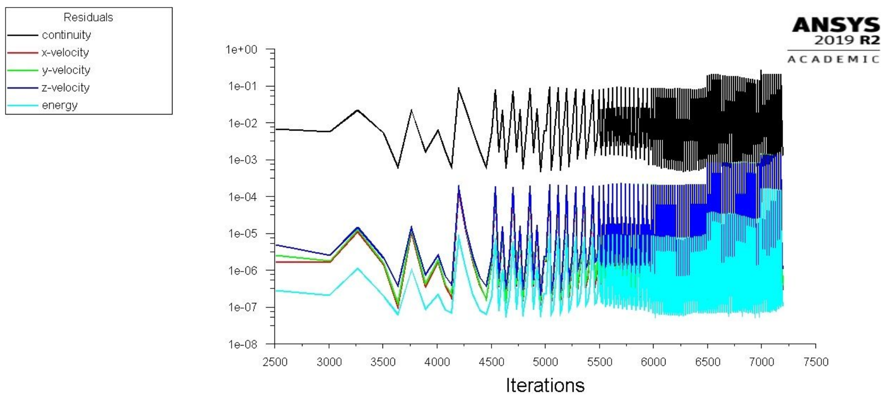
Table 6. Boundary Conditions for Single Stage Model

Property	**Detail**
Scheme	Coupled
Gradient	Least Squares Cells Based
Pressure	Second Order
Momentum	First Order Upwind
Turbulent Kinetic Energy	First Order Upwind
Turbulent Dissipation Rate	First Order Upwind

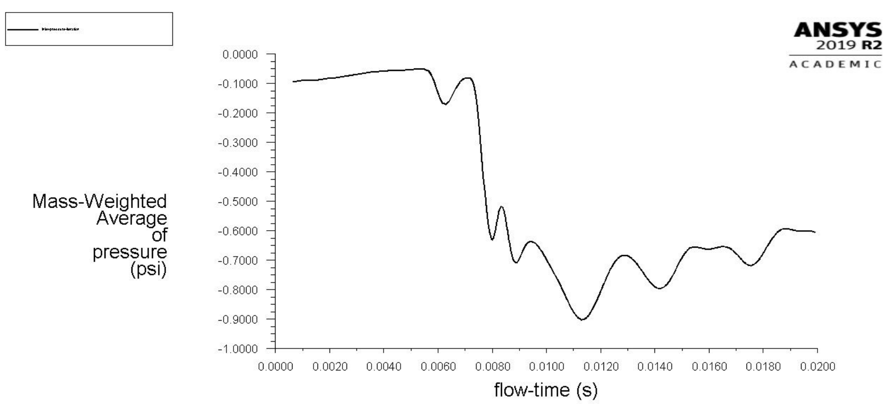
Table 6. Solver Settings for Single Stage Model

Results of the 40,000 ft condition with the Inviscid Method

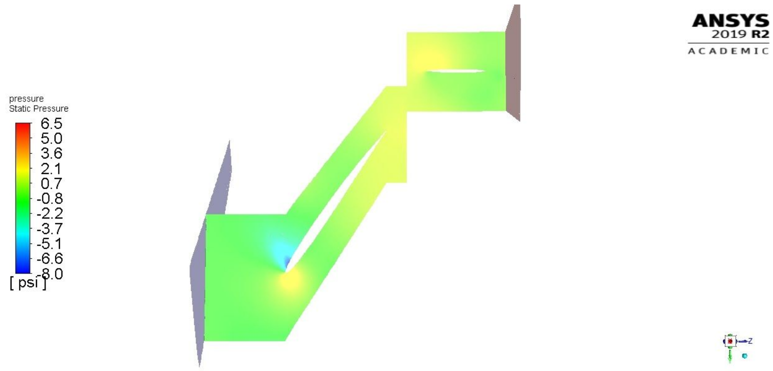
The model was run in multiple steps, starting at slower RPM and gradually increasing until 4,000 RPM was reached. Below shows the residual plot for the simulation at 4,000 RPM. The residuals per time step was shown to converge to 1e-03 or lower for continuity.



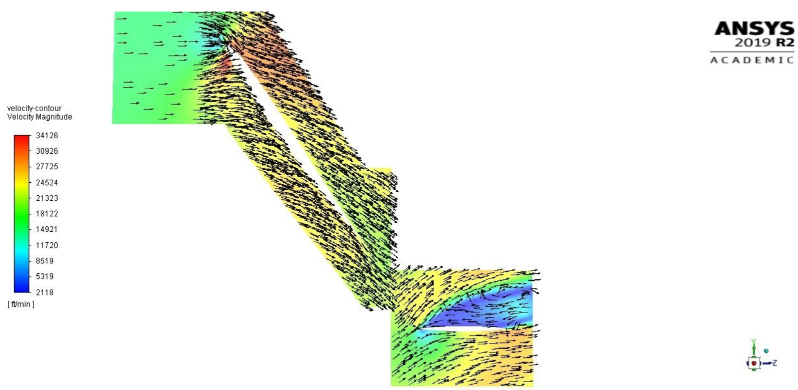
The mass-weight average for pressure at the outlet is shown in the below graph, plotting average static pressure over flow time.



The static pressure contour is shown below. There is a small low pressure bubble at the leading edge of the rotor blade, which is expected. The contour plot uses the global auto range so the difference between the pressure at the inlet compared to the outlet may seem small but is actually significant. The compressor stage pressure ratio is derived from data from the Turbo Topology tool, mentioned later.



The velocity vector and contour plot is shown below, with peak velocities of about 30,000 ft/min. From the contour plot, it is apparent that there is a large recirculation, low velocity region at the stator. In the Multi-stage design, this issue resolved slightly by introducing a 10 deg angle to the stator blade.

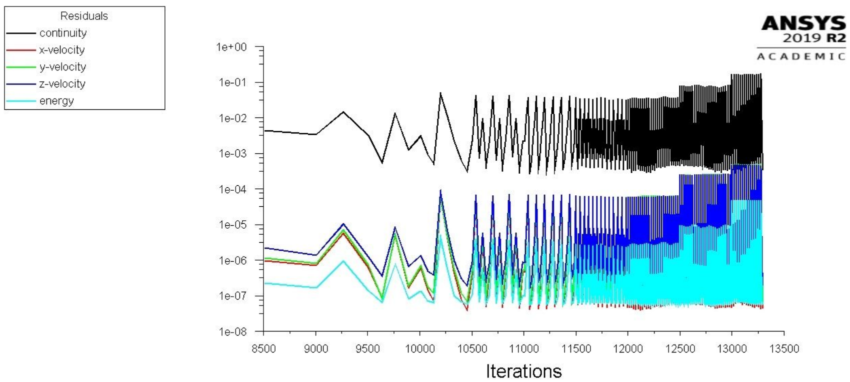


Using the Turbo Topology Tool, results are given in the table below:

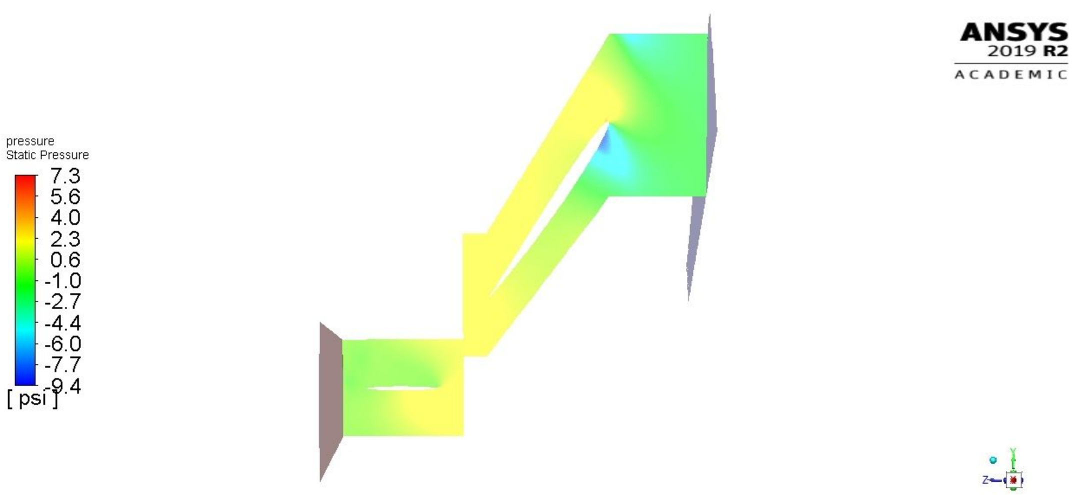
Property	Value
Compressor Stage Pressure Ratio (CPR)	1.72
Torque (lbf-ft)	130

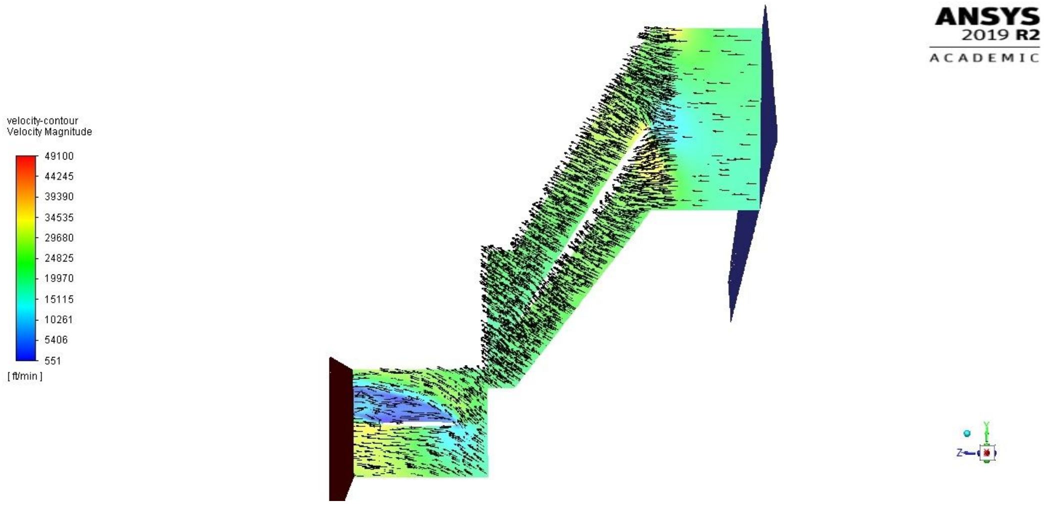
Results of the Sea Level condition with the Inviscid Method

The simulation was repeated for sea level altitude. The residuals, shown below, also show good convergence. There was some concern for the upper peaks of the residuals gradually increasing but all the residuals converged to acceptable values after each time step.



The following two images are the pressure contour plot and the velocity vector & contour plot, respectively. Results are similar to the results from the 40,000 ft simulation.





Using the Turbo Topology Tool, results are given in the table below. The CPR and Torque values are higher than that of the higher altitude, which was expected. A CPR of 2.15 is high for a single compressor stage but it is likely because turbulent and viscous effects are not accounted for in this inviscid model.

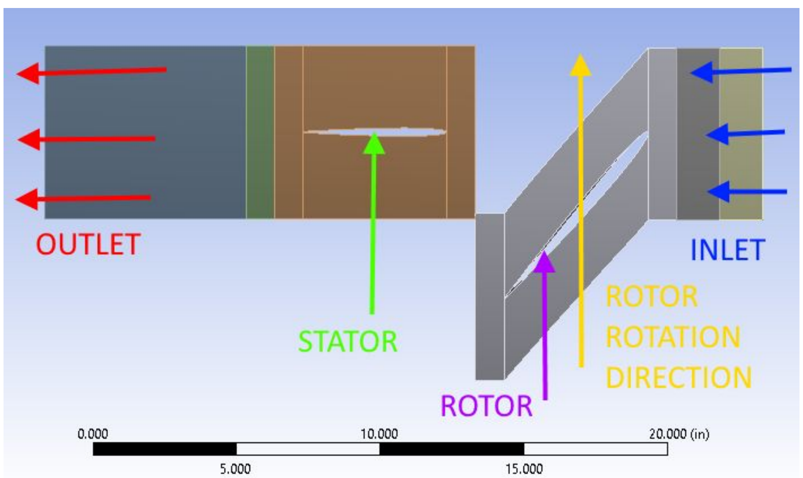
Property	**Value**
Compressor Stage Pressure Ratio (CPR)	2.15
Torque (lbf-ft)	183

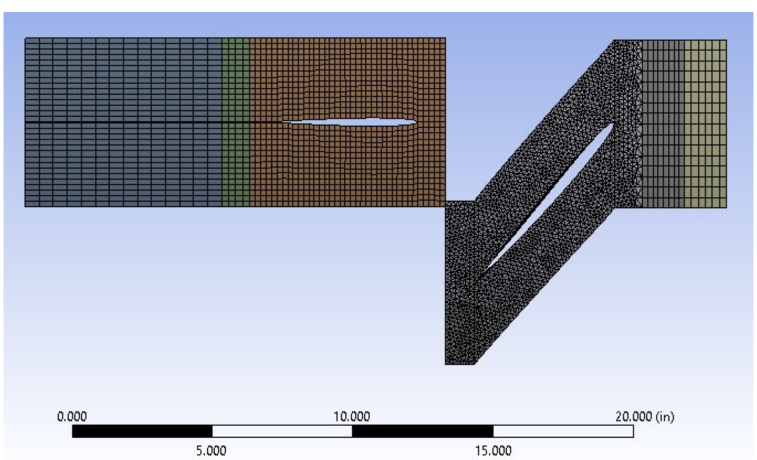
Turbulent Single Stage Design

The pressure increase through a single rotor-stator pair was modeled using the turbulent k-epsilon, realizable model. The goal of this design was to compare the inviscid model to the turbulent model which would be run next.

Geometry and Mesh

The same blades were used as in the inviscid model. The outlet was extended in order to prevent reverse flow at the pressure outlet. This was not a problem for the inviscid model, but it was a prevalent problem for the turbulent model. This was one indicator that suggested that the two models would provide significantly different results.





The following table details the parameters used for meshing.

Property	Details
Inlet Sizing	0.25 in
Adaptive Sizing	0.2 in
Rotor Blade Region Sizing	0.15 in
Rotor Method	Automatic
Stator Blade Region Sizing	0.10 in
Stator Method	Multizone
Outlet Sizing	0.25 in

Boundary Conditions & Solver Setting

The following table lists the boundary conditions.

Property	Detail
Turbulence Model	K-epsilon, realizable
Air Density	Ideal Gas Model
40,000 ft Operating Pressure	2.7 psi
Sea Level Operating Pressure	14.7 psi
40,000 ft Air Temperature	-69.7°F
Sea Level Air Temperature	77°F
Rotational Velocity	4,000 RPM

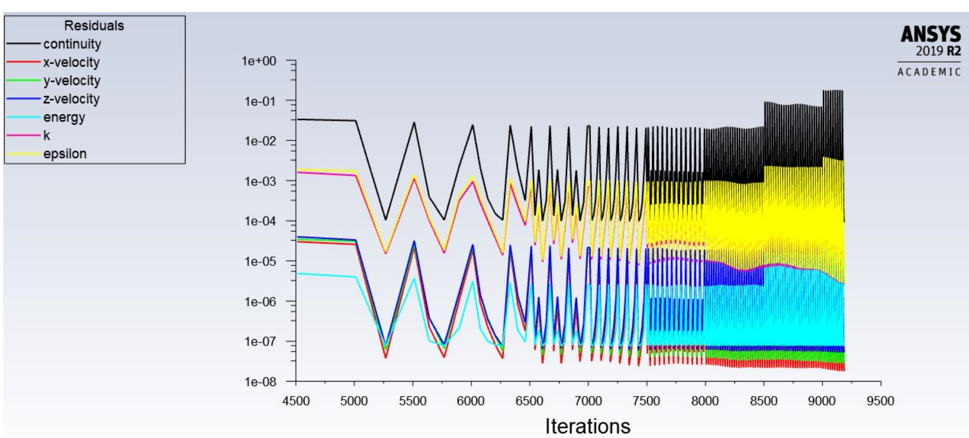
The following table outlines the solver parameters used for the 2D manifold simulations.

Property	Detail
Scheme	Coupled
Gradient	Least Squares Cells Based
Pressure	Second Order
Momentum	First Order Upwind
Turbulent Kinetic Energy	First Order Upwind
Turbulent Dissipation Rate	First Order Upwind

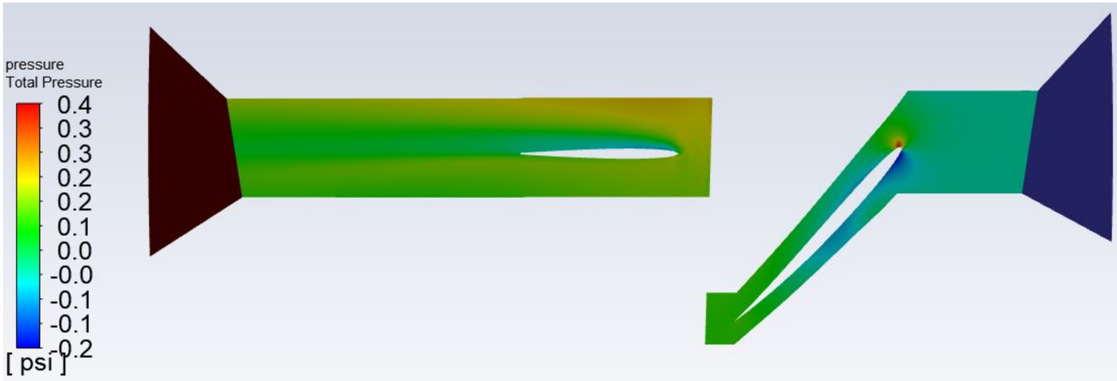
Table 5.2. The solver settings used for the first inlet stage.

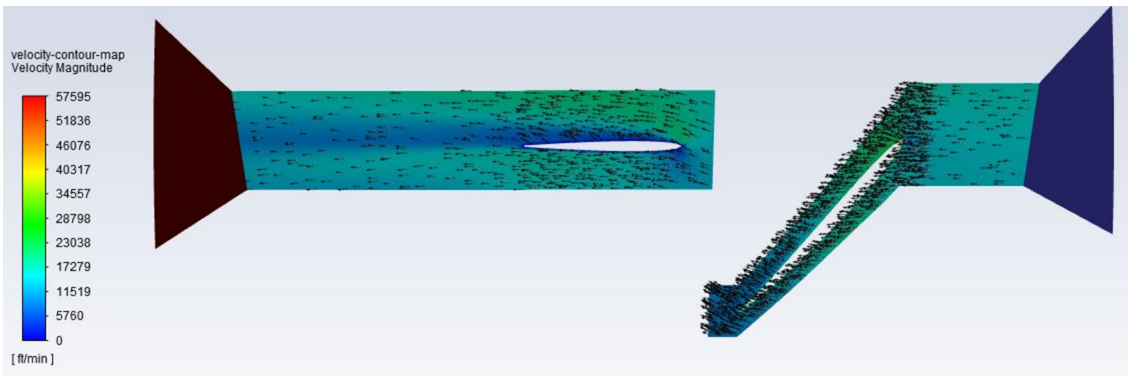
Results of the 40,000 ft condition with the Turbulent Model

The simulation approach for this model was similar to the Inviscid Single Stage simulation. Initially, the RPM was set low to start the solution and was gradually increased to 4,000 RPM. The residuals shown below are of the simulation starting at 4,000 RPM.



A total pressure contour is shown below. The scale was adjusted to show that the pressure ratio from this model is significantly less than the pressure ratio of the inviscid model.





In the above figure, the velocity vectors & contour is shown. There are peak velocities up to ~57,000 ft/min which is high and approaching sonic conditions, but most of the flow exists in the 10,000 - 20,000 ft/min range. From the Turbo Topology report, the results are the following:

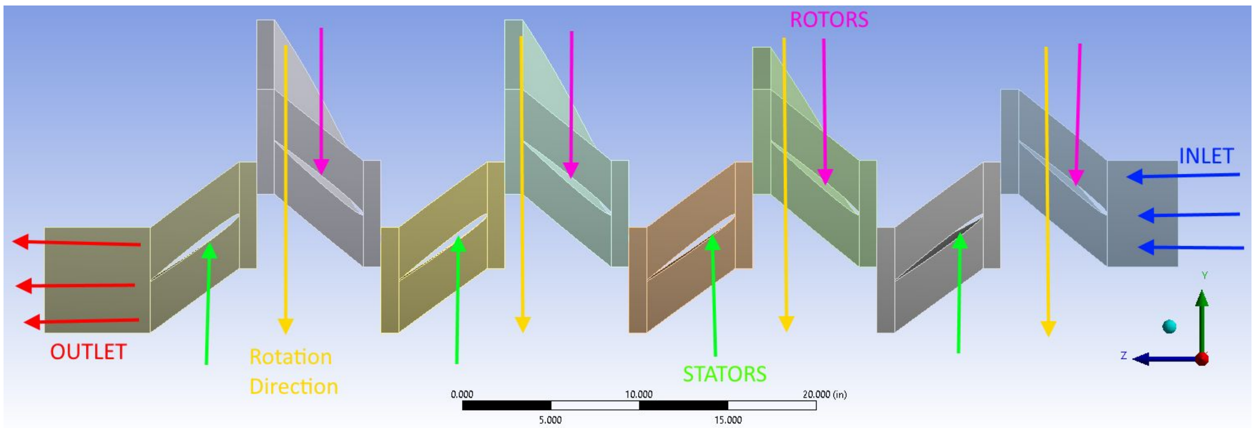
Property	Value
Compressor Stage Pressure Ratio (CPR)	1.26
Torque (lbf-ft)	95

The CPR is significantly lower than the inviscid solution which was anticipated. The concern is the lower torque value. With the viscous interactions of the system now accounted for in the model, it would be expected for the torque required to be increased. Perhaps the viscous interactions make it easier to move the blades but that seems unlikely.

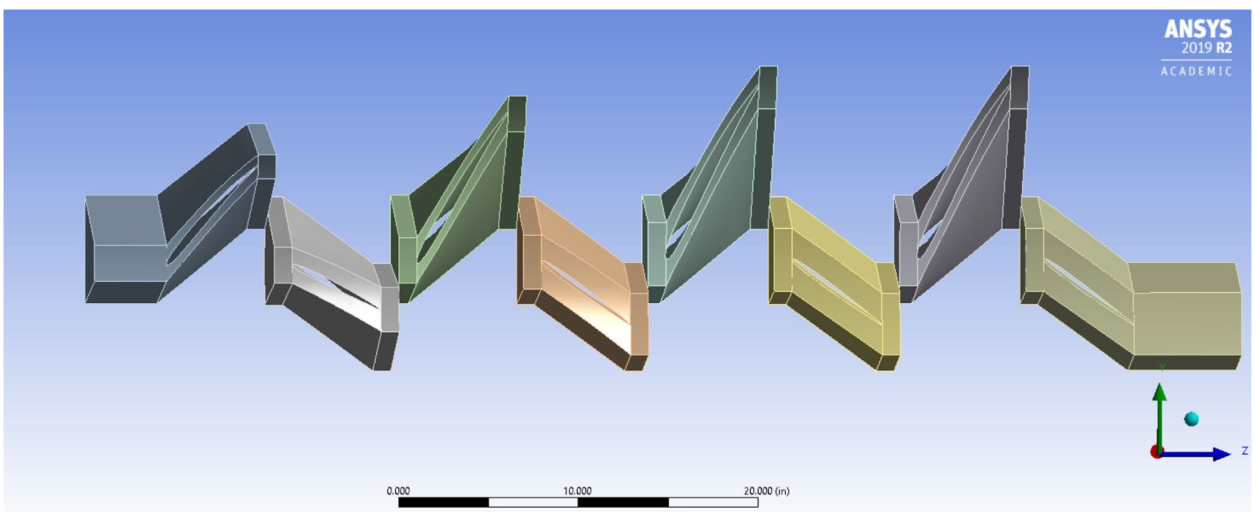
Multistage Design

Geometry

The following figures show the geometry of the rotor blades, stator blades, fluid regions, and rotor rotation direction. After some research, a stator angle of -15 deg was deemed to be more efficient and better for improving pressure ratio by reducing the recirculation bubble.



Geometry seen from shroud



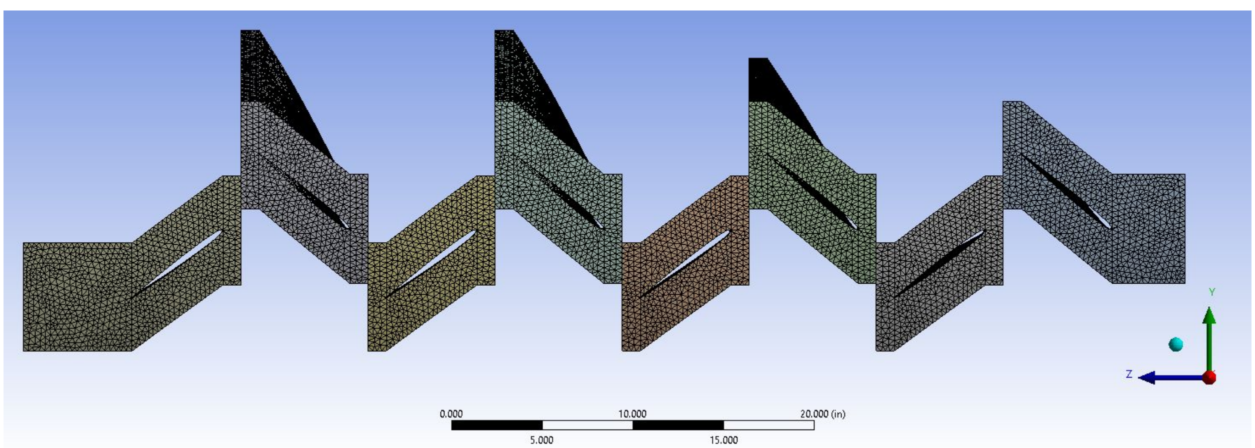
Geometry viewed from hub

The following table lists the details of the geometry used.

Inlet length	4 in
Horizontal Rotor Blade Length	5 in
Horizontal Stator Blade Length	5 in
Rotor-Stator Gap	2 in
Outlet Length	4 in
Blade Shapes	Bladegen NACA 0006-83
Rotor angle	45 deg
Stator angle	-15 deg

Mesh

The following figure shows the mesh used for the geometry above.



The following table lists the mesh details

Body Sizing	0.3 in
Periodic Matching	True

Method	Automatic
Element Count	2,629,204

Boundary Conditions and Solver Settings

The following table lists the boundary conditions.

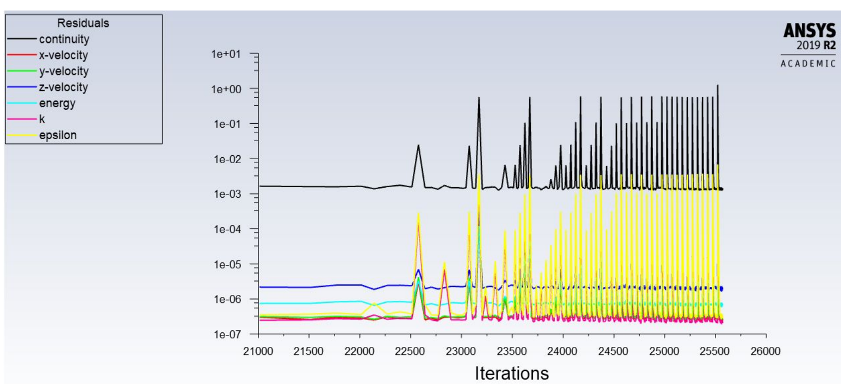
Property	**Detail**
Turbulence Model	k-epsilon with enhanced wall treatment
Air Density	Ideal Gas Model
40,000 ft Operating Pressure	2.7 psi
Sea Level Operating Pressure	14.7 psi
40,000 ft Air Temperature	-69.7°F
Sea Level Air Temperature	77°F
Rotational Velocity	10,000 RPM

The following table lists the solution parameters.

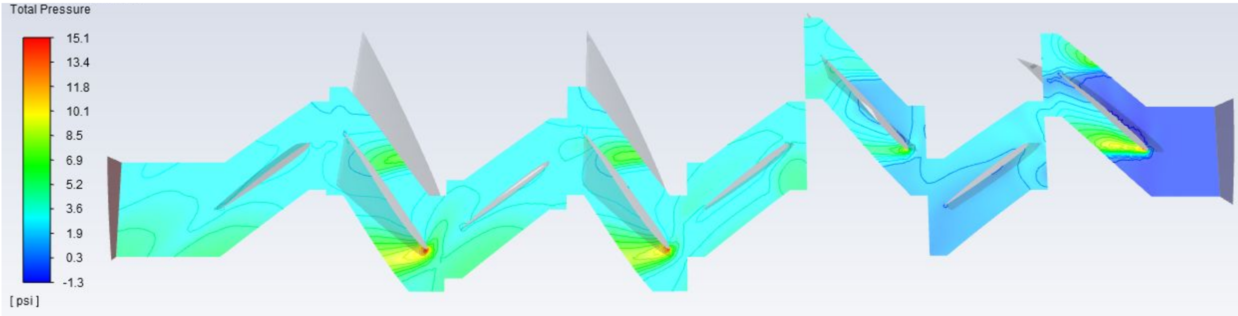
Property	Detail
Scheme	Coupled
Gradient	Least Squares Cells Based
Pressure	Second Order
Momentum	First Order Upwind
Turbulent Kinetic Energy	First Order Upwind
Turbulent Dissipation Rate	First Order Upwind

Results of the 40,000 ft condition

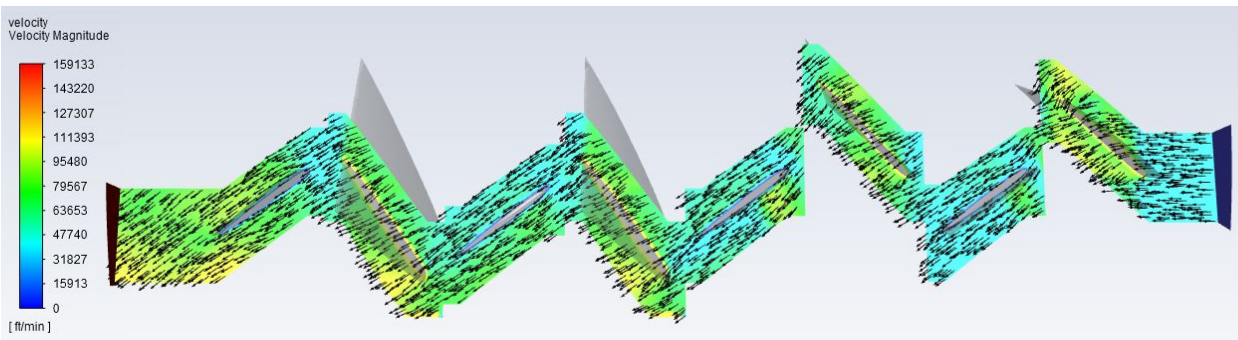
The residual graph of the 40,000 ft condition



The pressure contour for the 4 stages with the inlet on the right and outlet on the left.



The velocity map for the 4 stages with the inlet on the right and outlet on the left.

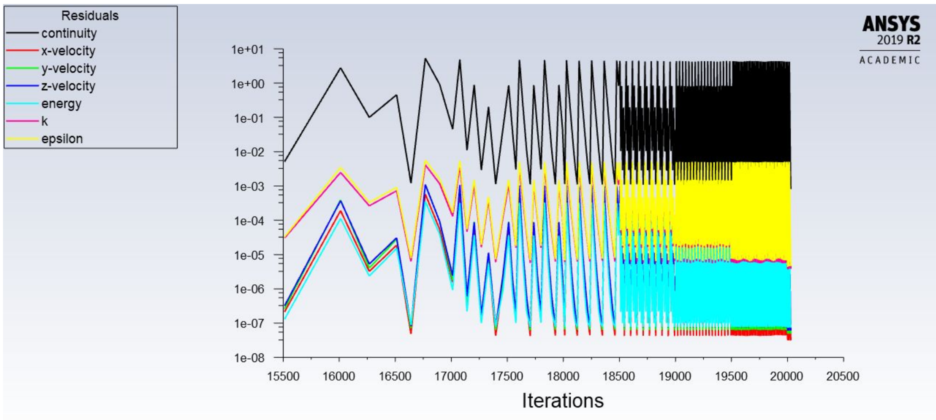


The turbo topology report data is summarized in the table below:

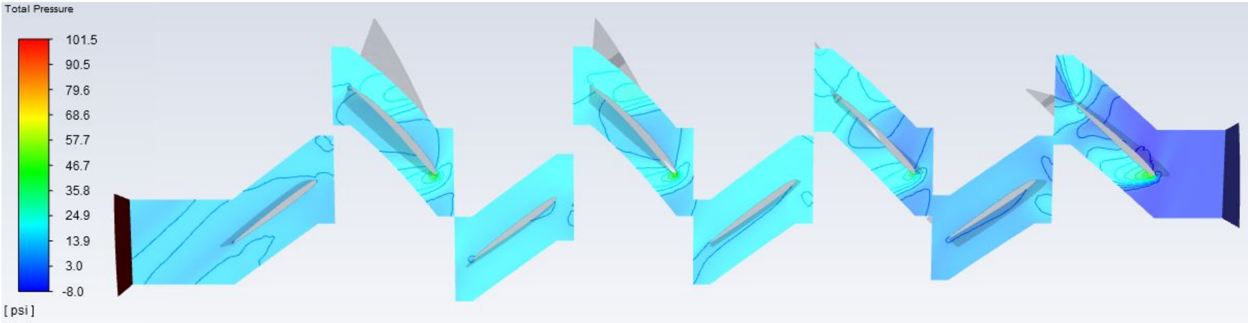
Stage	Total Temp (deg F)	CPR	Torque (lbf-ft)
1	91.1	3.33	119
2	220.7	2.10	96
3	278.3	1.33	38
4	316.1	1.19	34
Total	**-**	**11.0**	**287**

Results of the Sea Level condition

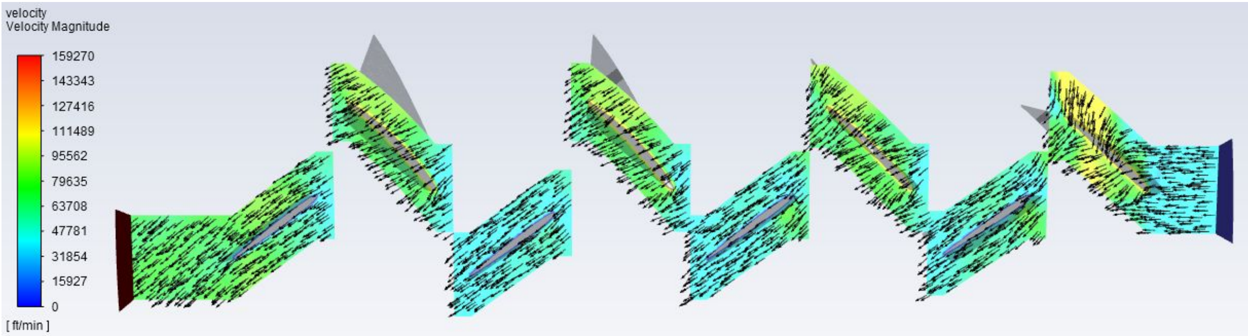
The residual graph for the sea level condition



The pressure contour for the 4 stages with the inlet on the right and outlet on the left.



The velocity map for the 4 stages with the inlet on the right and outlet on the left.



The turbo topology report data is summarized below:

Stage	Total Temp (deg F)	CPR	Torque (lbf-ft)
1	249.5	2.82	543
2	354.0	1.62	327
3	406.1	1.24	152
4	447.5	1.18	143
Total	**-**	**6.7**	**1165**

Operating conditions of design:

Based on the Turbo Topology Results, the number of stages can be calculated. Since the sea level and 40,000 ft altitudes resulted in different CPRs for 4 stages, the calculated number of stages will be for the lower CPR value. **Due to time constraints, other stages were not modeled so for the remainder of the stages needed, a CPR value of the 4th stage will be assumed with a 1.5% estimated reduction in CPR with each additional stage**. The torque of the 4th stage was also used for the remainder of stages as a conversative estimate.

Altitude	Total Number of Stages for 20:1 CPR	Total Torque (lbf-ft)
40,000 ft	9	457
Sea Level	15	2738

The number of stages needed is likely significantly higher since the CPR should start diminishing with each additional stage past stage 4. Based on the table above, **a total number of stages of 15 was selected.** The resulting CPR, torque, and power are the following:

Altitude	CPR (with 15 stages)	Total Torque (lbf-ft)	Power (hp)
40,000 ft	27.6	661	1258
Sea Level	20.1	2738	~5200

At altitude, the CPR is 27.6 for 15 stages, which provides significant margin compared to the requirement. **For all the stages, the operating speed is set to 10,000 RPM.** At sea level, the power consumption at max operating speed is ~5200 hp, while at 40,000 ft, the power is reduced to 1258 hp. What is unexpected is that the CPR at sea level is lower than the CPR at 40,000 ft, but results have been consistent throughout our simulations.

One type of compressor efficiency that can be calculated is the input power compared to the compression ratio, which is Efficiency = Power / CPR.

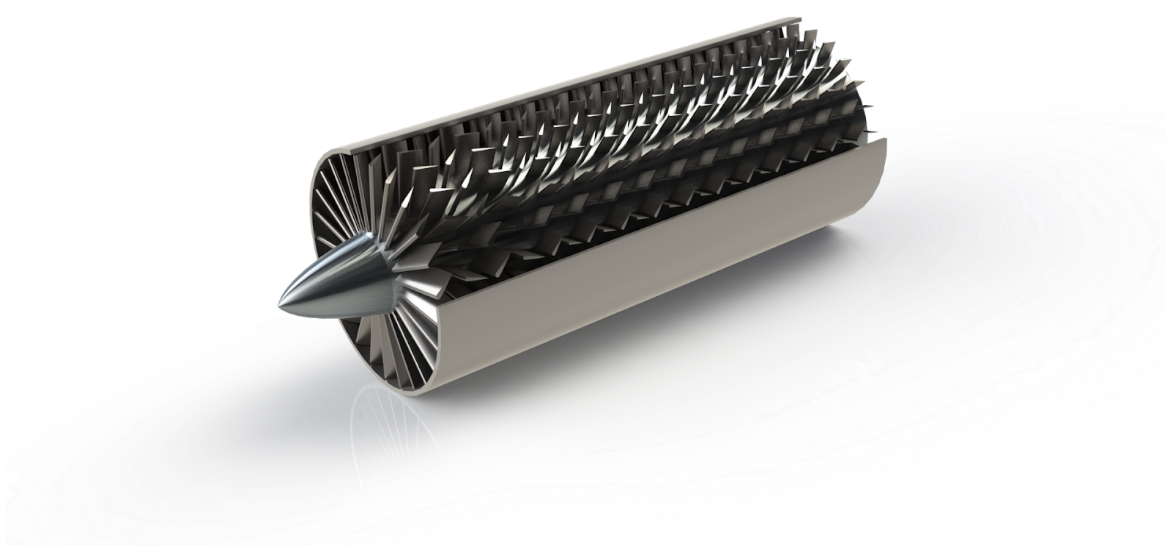
Altitude	Power / CPR (hp/unit CPR)
40,000 ft	45.6
Sea Level	258.7

Conclusion

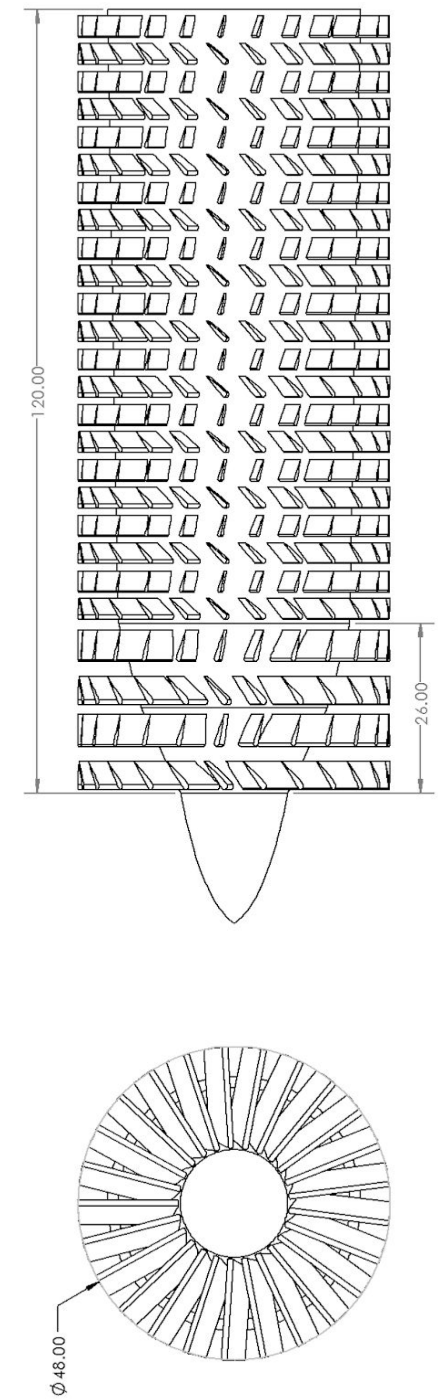
In summary, the results are provided in the table below. The compressor is designed to have 15 stages with the sea level altitude being the limiting factor in terms of number of stages. At 40,000 ft, there is a significant margin in CPR to satisfy the requirement.

Altitude	CPR (with 15 stages)	Total Torque (lbf-ft)	Power (hp)	Power / CPR (hp/unit CPR)
40,000 ft	27.6	661	1258	45.6
Sea Level	20.1	2738	~5200	258.7

Using the results from CFD, some hand calculations, and assumptions made for CPR per stage following the 4th stage, a CAD model was generated with SolidWorks, shown below.



A 2D simple drawing is also shown below. The outer diameter of the compressor is 4 ft. The hub diameter tapers to a larger diameter in the first two stages. Afterwards, the hub diameter tapers slightly more up to the 15th stage. The additional tapering is to promote additional pressure and temperature rise.

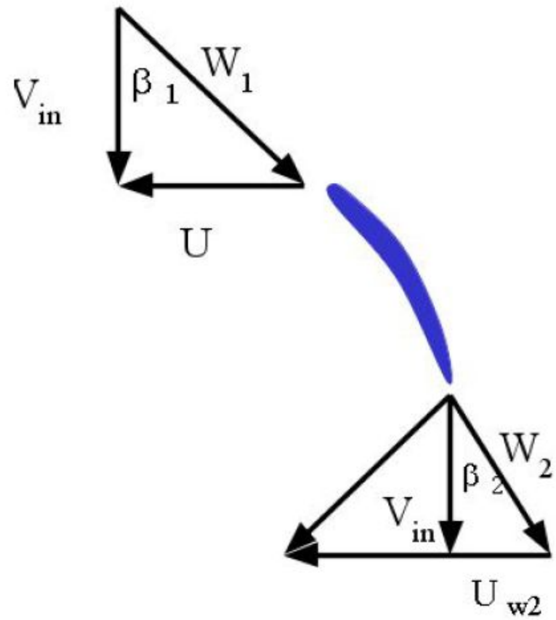


Hand verification calculations

For hand calculations, we focused on understanding the resulting exit temperatures across each stage from thermodynamic theory as well as the operating efficiency of each stage to ultimately get an engine efficiency. The calculations shown in this section use the given properties in the table below for 40,000 ft altitude. Specific heat and specific heat ratio are nominal values that will not vary significantly with 40,000 ft altitude compared to sea level.

Property	**Value**
Pressure	2.70 psia
Temperature	-69.7 deg F
Density	~ 0.3 kg/m^3
Cp	0.24 Btu/lb-F
γ (specific heat ratio)	1.40

To calculate the exit temperature across a stage, the velocity triangle must be visited, shown below. V_{in} is the inlet velocity of the air and U is the tangential velocity of the rotor blade. However, for this design activity, the compressor is assumed to be placed on a test stand (not in a wind tunnel) so the incoming air is not from the speed of a moving aircraft but rather just the rotor blades drawing air in from the inlet.



It is difficult to approximate V_{in} with this approach so CFD was used to understand the mass capture of air into the compressor. By knowing the area of the stage (which is the cross sectional area between the shroud diameter and the hub diameter) and the density of the air, the velocity can be calculated ($M_{dot} = \varrho V_{in} A$). U can be calculated by just knowing the rotational speed of the blades and the diameter of the blades.

Using V_{in} and U values, the air to blade relative velocity, W_1, can be solved as well as β_1, which is the angle between V_{in} and W_1.

$$W_1 = \sqrt{U^2 + \quad V_{in}^2} \quad \text{and} \quad \beta_1 = \arctan\left(\frac{U}{V_{in}}\right)$$

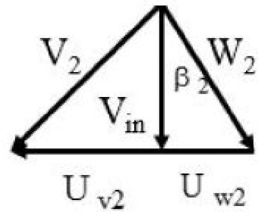
In a given stage, the area does not change, so for the exit velocity triangle, the axial velocity V_{in} should remain the same. The exit angle β_2, the relative blade speed U_{W2}, and the relative air speed W_2 can be calculated using the following equations:

$$\beta_2 = \beta_1 + \Delta\beta \ ,$$

where $\Delta\beta$ is the change in angle of the blade, and

$$U_{W2} = V_{in} \tan(\beta_2) \quad W_2 = \frac{V_{in}}{\cos(\beta_2)}$$

The actual air velocity, V_2, can then be calculated as well as the relative blade speed associated with the actual air velocity, U_{V2}.



$$U_{V2} = W_x - U_{W2} \quad V_2 = \sqrt{V_{in^2} + U_{V2}^2}$$

Where Wx is the is actually U (tangential blade speed).

The equations to then calculate the temperature rise across a stage is given below:

$$T_1 = T_{in} + \frac{V_{in}^2}{2Cp}$$

Where T1 is the temperature entering the stage accounting for velocity.

$$\text{Shaft Torque} = T_{shaft} = \dot{M} \times \frac{D_{blade}}{2} \times (U_{v1} - U_{v2})$$

But U_{v1} is the tangential component of the inlet velocity which is equal to 0.

Shaft power could be calculated using the following equation:

$$P_{shaft} = T_{shaft} \times 2\pi \times \omega$$

Where ω is the rotational speed in revolution per second.

$$\text{Specific Work} = W_{stage} = \frac{P}{\dot{M}} \quad \text{Exit Temperature} = T_2 = T1 + (W_{stage}/Cp)$$

The compressor pressure ratio can be calculated from isentropic relations:

$$\frac{P_2}{P_1} = CPR = \frac{T_2}{T_1}^{(\gamma / (\gamma - 1))}$$

The efficiency, which can be specified as the input power over the pressure ratio can then be calculated by dividing shaft power by the compression ratio of P/CPR. This would give a value representing how much power is needed to achieve unit CPR.

The calculations laid out above are what can be used to calculate the compression ratio and efficiency of each stage. The inlet temperature and inlet pressure changes with each stage as well as the velocity (since there is tapering in the area) so the exit temperatures and pressure from the previous is inputted into the next as the inlet conditions.

Using the equations above, a spreadsheet was made to streamline the calculation process and is shown in the Figure below.

Description	Symbol	Value	Units
Environment Conditions			
Nominal Altitude of Flight	altitude	12192	m
Gravity constant	g	9.81	m/s2
Specific Gas Constant	r_gas	286.7	J/kg/K
Temperature at Sea Level	T_0	288.15	K
Temperature at Altitude	T_1	216.9	K
Total Pressure at Sea Level	Pt_0	101325	Pa
Total Pressure at Altitude	Pt_1	18649	Pa
Density at Altitude	rho	0.300	kg/m3
Specific Heat	cp	1003	J/kg/K
Specific Heat Ratio	gamma	1.4	-
Sound speed	a_speed	295.1	m/s
Mach number at inlet	**mach**	**0.35**	**-**
Velocity at inlet	v_in	103.3	m/s
Compressor Inlet and Single Stage Blade Geometry			
Outer Diameter of Inlet	**D_outer**	**1.22**	**m**
Height of Blades at Inlet	**height**	**0.4**	**m**
Hub Diameter of Inlet	**D_hub**	**0.42**	**m**
Area of Compressor Inlet	A_inlet	1.03	m2
Mass Capture of Inlet	mdot	31.9	kg/s
RPM of Compressor	**RPM**	**5000**	**rpm**
Blade Mean Velocity	v_blade	-214	m/s
Air to blade relative velocity	w_relative	238	m/s
Blade Relative Inlet Angle	**beta_1**	**-64.3**	**deg**
Blade Turning Angle	**beta_delta**	**25**	**deg**
Blade Relative Exit Angle	**beta_2**	**-39.3**	**deg**
Relative Exit Blade Speed	u_w2	-84	m/s
Relative Exit Air Speed	w_2	133	m/s
Relative Blade Speed associated with actual air velocity	u_v2	-130	m/s
Actual Exit Air Velocity	v_2	166	m/s
Main Power and Pressure Ratio Calcs			
Adjusted Temperature	T_1_update	222.2	K
Compressor Shaft Torque	T_shaft	1697	N*m
Power from shaft torque	Power	8.9E+05	J
Work from single stage	W_stage	2.8E+04	W
Isentropic Efficiency	**n_eff**	**0.90**	**-**
Exit Temperature	T_2	247.2	K
Pressure Ratio (P2/P1)	**PR**	**1.45**	

The main inputs of the spreadsheet are: altitude, inlet velocity, shroud diameter, blade height, hub diameter, RPM, desired blade turning angle, and assumption on isentropic efficiency.

By performing the various CFD simulations, the mass flow rates were noted and inputted into the spreadsheet to obtain velocities. The pressure ratios and efficiencies are shown in the tables below:

From Hand Calculations

Altitude	**CPR (with 15 stages)**	**Total Torque (lbf-ft)**	**Power (hp)**
40,000 ft	33.1	540	1028
Sea Level	25.2	2243	4271

Using the hand calculations, the CPR is estimated to be significantly more, which is expected. The total torque is less which was also anticipated. But the values are within the ballpark range compared to the CFD results.

Design Time Estimate

Total time around 120 hrs, a majority of the time was spent to familiarize with bladegen, meshing, fluent setup, the simulations took around 50 hours in total, and report, presentation and other miscellaneous parts took around 20-30 hours.