Conceptual Design of Supersonic Mixed-Compression Intake for Ramjet Powered Target Drone

Muhammad Adnan Hutomo[1, a), b)] and Romie Oktovianus Bura[2, c)]

[1]*Student, Aeronautics and Astronautics,*
Faculty of Mechanical and Aerospace Engineering
Bandung Institute of Technology, Ganesha 10, Bandung, Indonesia
[2]*Flight Physics Research Group*
Faculty of Mechanical and Aerospace Engineering
Bandung Institute of Technology, Ganesha 10, Bandung, Indonesia

[c)]Corresponding author: romiebura@ae.itb.ac.id
[a)]adnanhutomo@gmail.com
[b)]adnanhutomo@students.itb.ac.id

Abstract. When designing a vehicle flying at supersonic speed, aerodynamic forces will become the main issue. Beside drag force gets higher, intake must be designed cleverly to provide an adequate air supply for engines. Most engines require high pressure, high total pressure recovery, high temperature, and subsonic airflow. To achieve this, supersonic intakes can take benefits from compressible flow phenomena such as shocks. This paper will cover the design methodology for mixed compression intake type and the CFD analysis as a validation. The intake will be designed at Mach 3 at high altitude and will be used to provide air for solid fuel ramjet. Target drone design reference comes from comparable ramjet powered target drone and missile with equal performance. For the initial sizing step analytic calculation with inviscid assumption will be done. Then geometry adjustment must be conducted to achieve critical condition. Shocks location will be determined at geometry adjustment stage. Finally, validate the design by CFD and analyze it from the results. ANSYS ICEM will be used as grid generation process and ANSYS FLUENT will be used as the solver.

INTRODUCTION

Target drone is the term used for drones which designed to be destroyed. Most target drones which have been designed is similar to missiles without warhead. Instead of removing warhead from missiles, military prefers to use well designed drones at lower price and higher reliability. In some cases, military want target drones with better performance than the most powerful missiles to anticipate future technology.

History of target drone started after the Second World War ended. In 1950, U.S. Navy wanted to train the readiness of its army when faced real combat situation. At that time, NASA has reached supersonic flight and the knowledge in designing supersonic aircraft is well understood. A conceptual design of supersonic target drone was proposed for the first time and it is called Northrop AQM-35. It designed to cruise at Mach 1.5 for several seconds. But soon, the proposal was scrapped because of the technology at that time was not sufficient enough to complete a full mission.

Many design has been proposed until now. But, only few are accepted until manufacturing stage. One of them is GQM-163 Coyote which is expected to be in service in June 2017. GQM-163 Coyote is a supersonic sea skimming target built by Orbital Science. Its main feature is the capability of high supersonic flight (Mach 3.5), with a range of approximately 220 km. To achieve this range and performance in its small body needs a high efficiency propulsion system. Therefore, GQM-163 Coyote is equipped with solid fuel ramjet which has high energy density compared to liquid fuel ramjet. The other reason is solid fuel ramjet is more reliable because of less moving parts in its system.

In order to design a supersonic ramjet powered target drone, it is highly recommended to consider aerodynamic performance first[1]. Aerodynamic performance can be divided into two, external and internal flow. In external flow, aerodynamic forces must be calculated based on the shape and flight conditions. When the Mach number is getting higher, heat transfer on the outer skin must be calculated. As for internal flow, it is mainly focused on how to supply air to propulsion system. Most of propulsion system cannot accept supersonic air flow. Therefore, intakes have a very crucial role in reducing airflow Mach number and increasing static pressure. Intake efficiency can be evaluated based on how the intake maintain energy from free stream ram air. Total pressure loss is an indicator of how much energy is released when the compression process occurs. To maintain total pressure loss low, analytical calculation in predicting shocks must be done precisely. When the intake geometry is not precisely calculated, normal shocks can appear in front of intake. This kind of shock is considered as strong shock.This could reduce intake performance dramatically.

DESIGN REQUIREMENT AND OBJECTIVE

Before designing an intake, we must have Design Requirement and Objective or DRO. Target drone we will use is formulated from comparing to other comparable target drone or missile. There are four target drones and missiles which are GQM-163 Coyote, MA-31, AQM-127 SLAT, and MQM-8. Only two ofthem are especially designed as target drone, the rest are missile-converted target drone. All of them are equipped with ramjet propulsion, but only GQM-163 Coyote is fueled with solid fuel. Therefore GQM-163 Coyote is designated as main design reference.

From these comparable target drones, the designed target drone must be better or at least has similar specification. Then, target drone's design point for its intake can be determined as:

1. The main mission of designed target drone must be able simulated supersonic cruise missiles such as BrahMos. Therefore, the minimum cruise speed at high altitude must be not less than Mach 3.
2. The propulsion system must use solid fuel (Hydroxyl-terminated polybutadiene/HTPB) ramjet for cruising.
3. Maximum diameter must not exceed 350 mm.
4. Cruise altitude at Mach 3 is at least at 11,000 km.

There are some assumption will be used in designing this intake:

1. Ramjet is in ideal condition.
2. Isentropic flow at intake.
3. Analytic calculation is using inviscid flow.
4. Cruise angle of attack is at zero degree.
5. No heat transfer on intake wall.

COMPRESSIBLE FLOW THEORIES

When air speed hits Mach 0.3, air density changes significantly. Then the compressible theories must be applied in calculation. Expressing airvelocity in compressible region must be stated in Mach number which an air speed ratio to the speed of sound. Therefore Mach number can be expressed as this equation.

$$M = \frac{v}{a}$$

$$a = \sqrt{\gamma RT}$$

Shocks can appear in this compressible regime. Shocks happen because air molecule on object's surface cannot transmit information in wave form to the front because it is already exceeded the speed of sound. There must be sudden deceleration in some area which is where shockwaves are located. This sudden deceleration can change fluid properties, such as pressure, temperature, density, and Mach number.

There are several types of shock wave which will be used in intake design.

- Normal Shock is a strong shockwave which can decelerate air speed from supersonic into subsonic. The air properties can change dramaticaly depending on Mach number. As Mach number in supersonic regime gets higher, the normal shockwave gets stronger. The air properties changes can be formulated as:

$$\frac{P_1}{P_0} = \frac{2\gamma M^2 - (\gamma - 1)}{\gamma + 1}$$

$$\frac{P_{t_1}}{P_{t_0}} = \left(\frac{(\gamma + 1)M^2}{(\gamma - 1)M^2 + 2}\right)^{\frac{\gamma}{\gamma - 1}} \left(\frac{(\gamma + 1)}{2\gamma M^2 - (\gamma - 1)}\right)^{\frac{1}{\gamma - 1}}$$

$$\frac{T_1}{T_0} = \frac{(2\gamma M^2 - (\gamma - 1))((\gamma - 1)M^2 + 2)}{(\gamma + 1)^2 M^2}$$

$$\frac{\rho_1}{\rho_0} = \frac{(\gamma + 1)M^2}{(\gamma - 1)M^2 + 2}$$

$$M_1^2 = \frac{(\gamma - 1)M^2 + 2}{2\gamma M^2 - (\gamma - 1)}$$

- Oblique Shock is similar to the normal shock, but since the orientation is not perpendicular to the flow direction, the shock becomes weak. Oblique shock wave capable to change flow direction. To calculate air properties change, we can project the air flow perpendicular to the oblique shock first. Then calculate it like normal shock equations. Theta, Beta, M (θβM) Relation below is a formula for air flow deflection, and shock angle in two dimensional if Mach number is known.

$$\tan\theta = 2\cot\beta \left[\frac{M_1^2 \sin^2\beta - 1}{M_1^2(\gamma + \cos 2\beta) + 2}\right]$$

Since the properties change of oblique is basicly the same, then by normalize the flow to the oblique shock, equations below can e formulated.

$$\frac{P_1}{P_0} = \frac{2\gamma(M\sin\beta)^2 - (\gamma - 1)}{\gamma + 1}$$

$$\frac{P_{t_1}}{P_{t_0}} = \left(\frac{(\gamma + 1)(M\sin\beta)^2}{(\gamma - 1)(M\sin\beta)^2 + 2}\right)^{\frac{\gamma}{\gamma - 1}} \left(\frac{(\gamma + 1)}{2\gamma(M\sin\beta)^2 - (\gamma - 1)}\right)^{\frac{1}{\gamma - 1}}$$

$$\frac{T_1}{T_0} = \frac{\left(2\gamma(M\sin\beta)^2 - (\gamma-1)\right)\left((\gamma-1)(M\sin\beta)^2 + 2\right)}{(\gamma+1)^2(M\sin\beta)^2}$$
$$\frac{\rho_1}{\rho_0} = \frac{(\gamma+1)(M\sin\beta)^2}{(\gamma-1)(M\sin\beta)^2 + 2}$$

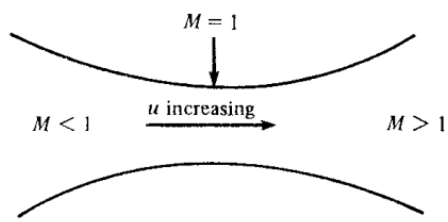
- Conical shock is basicly an oblique shock that occupies in three dimensional spaces. Therefore air can be deflected in three dimensional spaces. The properties change can be treated as an oblique shock, but it is different when determining the shocks angle. The equation was first formulated by Taylor dan Maccoll using linearized flow approach.

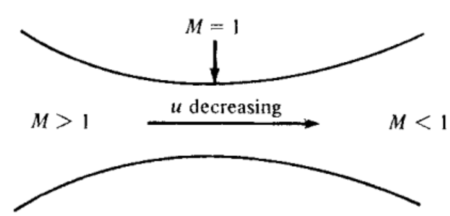
$$\frac{\gamma-1}{2}\left[V_{max}^2 - V_r^2 - \left(\frac{dV_r}{d\theta}\right)^2\right]\left[2V_r + \frac{dV_r}{d\theta}\cot\theta + \frac{d^2V_r}{d\theta^2}\right] - \frac{dV_r}{d\theta}\left[V_r\frac{dV_r}{d\theta} + \frac{dV_r}{d\theta}\left(\frac{d^2V_r}{d\theta^2}\right)\right] = 0$$

This equation has no closed solution. Therefore numerical calculation must be done.

Compressible Flow in Duct

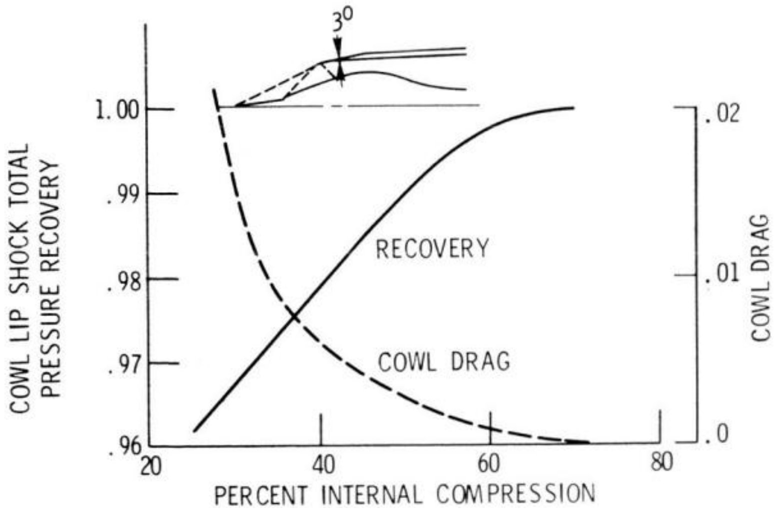
Compressible flow through duct behave differently compared to incompressible flow. It is the opposite from the incompressible flow. In incompressible flow, the density will always be constant to conserve the mass flow. But, in compressible flow, the air can be expanded or compresed. Therefore to obey the conservation of mass law, when the cross section area decreasing then the velocity must be decreasing too. It is the exact opposite when the cross section area is increasing.



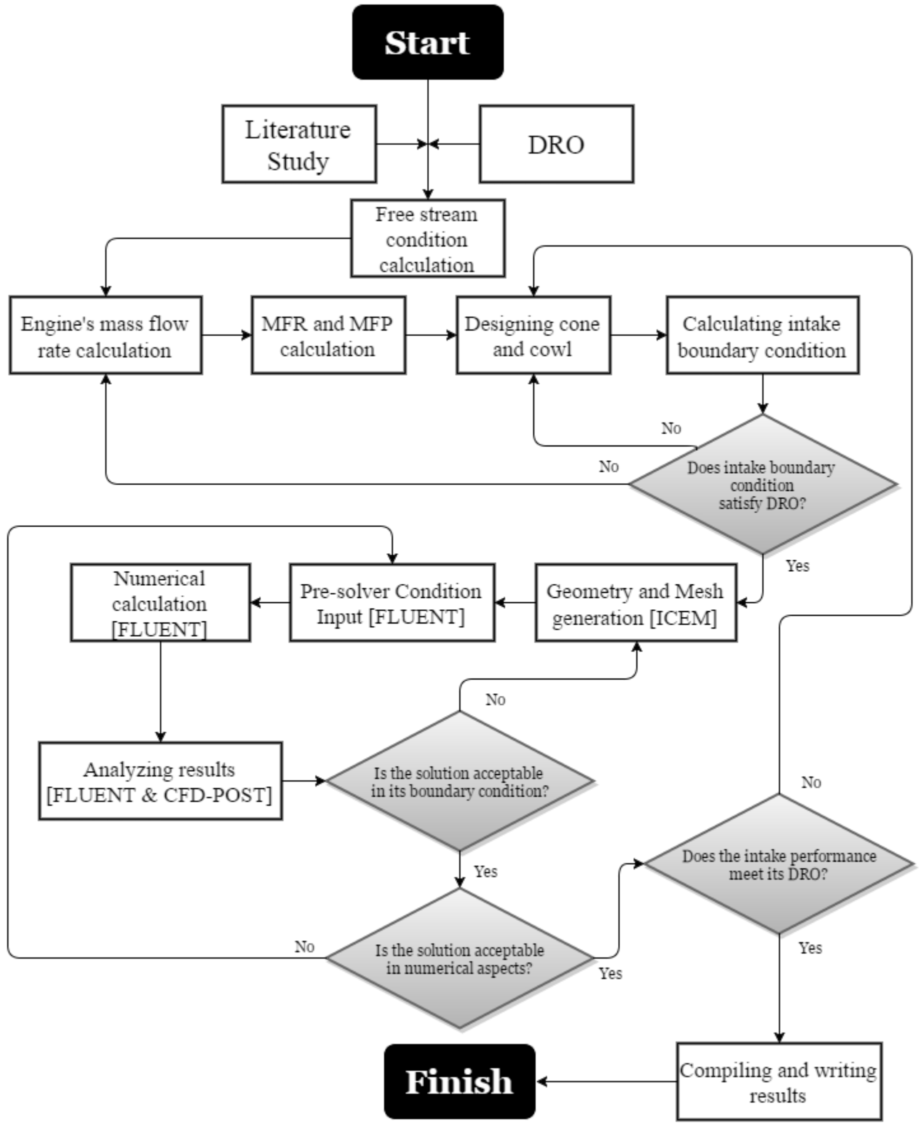


SUPERSONIC INTAKE DESIGN PROCESS

The designed intake must be working at Mach 3. When cruise speed exceeds Mach 2.5, external compression intake design is not effective and efficient even for multi-ramp configuration. Therefore the mixed compression intake design is chosen. The main reason why external compression alone cannot work effectively is the cowl drag produced. When many oblique shocks meet at cowl it may produce normal shocks at the tip. The shock itself is considered as strong shock. The mixed compression intake design can decrease the cowl lip drag and total pressure recovery. This curve visualize how much cowl lip drag to the portion of internal compression.



Curve above can be a reference for determining the portion of internal compression should happen. The work flow intake design process done in this paper is shown below.



Free Stream Condition

The DRO has been designated in the previous section. From the DRO we can define air properties when the target drone cruising. The table below is the summary of air properties would be.

Flight Requirement and Free Stream Condition			
Mach number (max)	M_{max}	3.0	
Altitude	H	11,000	m
Density	ρ	0.3403	Kg/m^3
Static Temperature	T	216	K

Total Temperature (@M3.0)	T_0	606	K
Static Pressure	P	22,700	N/m^2
Total Pressure (@M3.0)	P_0	833,832	N/m^2
Gamma	γ	1.4	
Gravity	g	9,785	m/s^2
Ideal Gas Constant	R	287.058	J/Kg. K

These properties will be our reference in designing the intake later. It is necessary to know the air properties after air flow through the intake. It will determine if the intake is suitable to the engine specification. Total pressure will be our main focus here because it can represent the efficiency of the intake.

Efficiency Specification

As mentioned before, one of the most important parameter at intake design is pressure recovery. Pressure recovery (η) can be defined as the ratio of total pressure inside the intake (at engine's face) divided by total pressure at free-stream.

$$\eta = \frac{(Po)engine}{(Po)\infty}$$

Intake minimum specification refers to military's total pressure recovery reference, MIL-E-5008B. This specification define the minimum total pressure recovery for intakes in every Mach number.

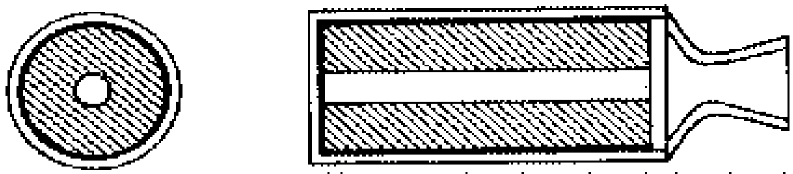
$$\eta_{spec} = 1 - 0.075(\mathrm{M} - 1)^{1.75},\ \ for\ 1 < M < 5$$

Then, from the equation above we can determine the minimum total pressure recovery for designed intake. When flying at Mach 3.0, the minimum total pressure recovery is **80.8%**.

Mass Flow Required Calculation

This target drone will be equipped with solid fuel ramjet or known as SFRJ for cruising. One main reason using SFRJ is its energy density is higher compared to liquid fuel. Other reasons are its reliability and requires less moving parts. In this paper, we assume to use pure hydroxyl-terminated polybutadiene (HTPB). The fuel is already known its burning properties and its performance expressed in equation.

The main feature of solid fuel ramjet is we can choose the grain shape to manipulate the thrust over time. When the drone is getting faster, then the drag will increase exponentially too. When the drag gets higher we need more thrust to maintain the velocity. Thus, we need higher thrust at the end of flight. Therefore we use internal burning tube shape for our solid fuel. The idea is as the time progresses, the burning area is getting larger thus will increase the thrust.



The mass flow required for solid fuel is different from liquid fuel. Where the liquid fuel ramjets can be controlled depending on how much the air mass flow is. However, in SFRJ case, it cannot be controlled in the middle of flight. But, it still can be engineered to meet the mission needs. Since the largest mass flow needed is in the end of flight, we can assume the SFRJ needs the maximum mass flow required. There have been a study of regression rate of pure HTPB as a function of air mass flux and chamber static pressure.

$$r_f = 0.073\ {G_{ox}}^{0.43} {P_c}^{-0.039}$$

Where the G_{ox} is the oxygen mass flux and P_c is the chamber static pressure. The equation above can be solved to get air mass flow needed by using air to fuel burning ratio, geometry of combustion chamber, and fuel properties. Then the mass flow needed can be expressed as equation below.

$$\dot{m}_a = 17.0546\ {P_c}^{-0.039}$$

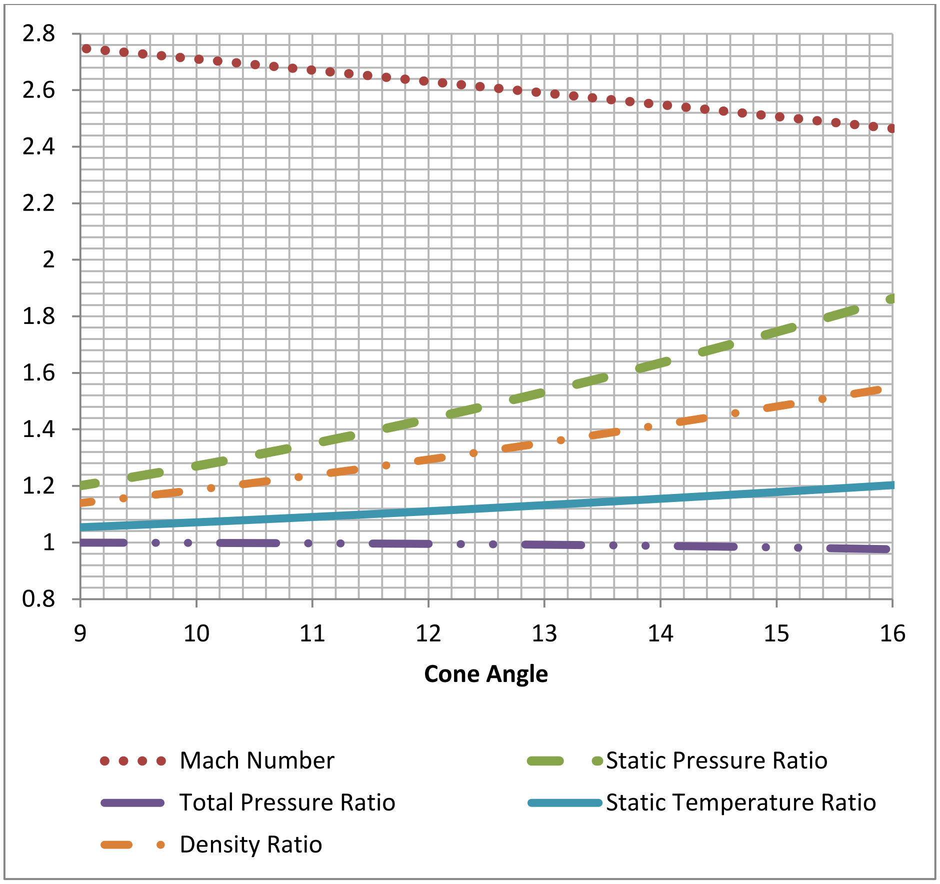
The predicted chamber pressure is around 350,000 Pa. Using the equation above, the mass flow required at stationary is **10.4 kg**. Other than mass flow rate, we need another parameter to determine capture area. According to Aircraft Engine Design by Mattingly, Mass flow parameter which a Mach number and air characteristic function, can be used. How we calculate capture area will be discussed later.

$$MFP = \frac{\dot{m}\sqrt{T_t}}{pA} = M\sqrt{\frac{\gamma g_c}{R}}\left(1 + \frac{\gamma - 1}{2}M^2\right)$$

Bicone Design

The Bicone can be defined as a cone that consists of two cone in serial. To put it simply, a cone is a single ramp in axissymmetrical and bicone is double ramp in axissymmetrical. External compression has its advantages such as it does not create oblique shock that would interfere each other. When there is shock interference, boundary layer can thicken that would increase energy loss.

Designing this bicone can be calculated in two dimensional except when defining shock angle and deflection angle. From the equation (8) can be concluded, as the deflection angle incerease, the shock angle would increase too. This would affect the shock strength. Oblique shocks generated should be as weak as possible to minimize total pressure loss. But, the weaker the shock is, the compression is weaker too. The graph below illustrates the compression performance as the cone angle increases.



Our main ojective is to achieve high compression ratio while maintaning total pressure loss as low as possible. Therefore we can choose the first cone angle with 1% total pressure loss which is 12.7 degrees. The compression ratio for the first oblique shock reach 151% which is good enough.

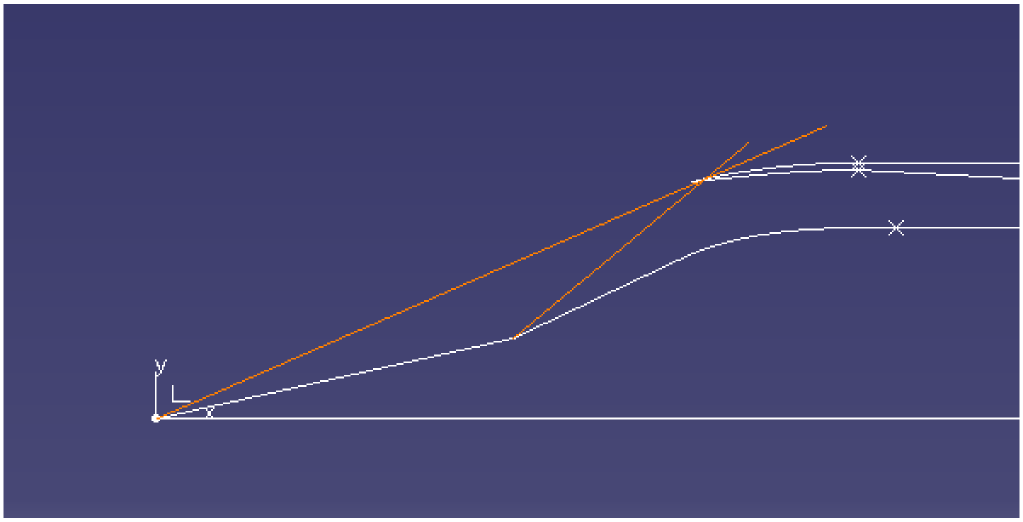
For the second ramp or the next ramp if available, there is an optimization criterion where the total pressure loss is minimum but still can achieve high compression ratio. It is called Oswatitsch (1944) criterion. The equation is expressed as a function of oblique shock angle and Mach number:

$$M_\infty \sin \beta_1 = M_1 \sin \beta_2 = \cdots = M_{n-1} \sin \beta_n$$

From equation above, with known freestream and after the first oblique shock Mach number, then the second shock angle is 26.4 degrees. Using equation (13), then the second ramp angle is 12.7 degrees. The summary for bicone geometry is listed below.

	Oblique Shock 1	Oblique Shock 2	Total
Cone Angle	12.7^o	12.7^o	-
Oblique Shock Angle	23.6^o	26.1^o	-
Mach Number	2.598	2.27	-
Total Pressure Ratio	0.9928	0.9971	0.9899
Static Pressure Ratio	1.5125	1.3554	2.0500
Static Temperature Ratio	1.1278	1.0916	1.2311
Density Ratio	1.3411	1.2416	1.6651

Using the data above, we can draw the cone and cone angle generated by using CAD. When drawing the cone, make sure that the shocks meet right at the tip of cowl because we are designing the intake in critical condition. Below is the initial geometry in 2D complete with cone shock represented as the orange line.



Capture Area Calculation

When calculating capture area, we assume the intake operates in critical condition. Therefore there is no spillage flow through the cowl lip. If we refer MFP in the previous section, the equation below determines the capture area needed to achieve the desired mass flow.

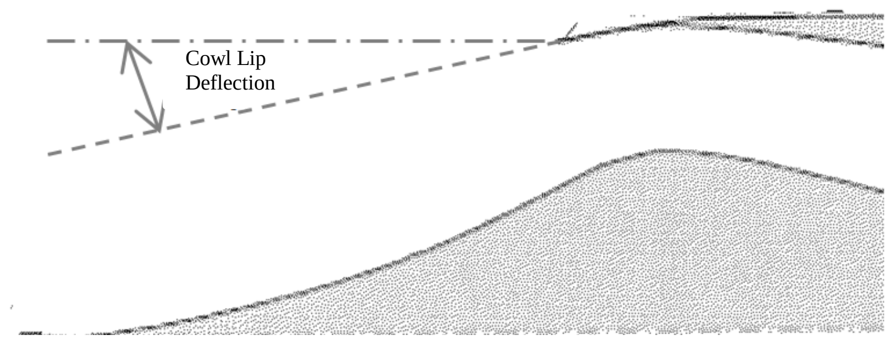
$$A_{CA} = \frac{MFR\ \sqrt{T_0}/P_0}{MFP}$$

The total pressure and total temperature used in this equation are for the flow right before the cowl lip. Thus, it is after the second oblique shock. From above equation, the minimal capture area is 0.0672 m^2. From Aircraft Engine Design reference, there should be area addition because of boundary layer around 4%. Therefore the minimum capture are is 0.0648 m^2.

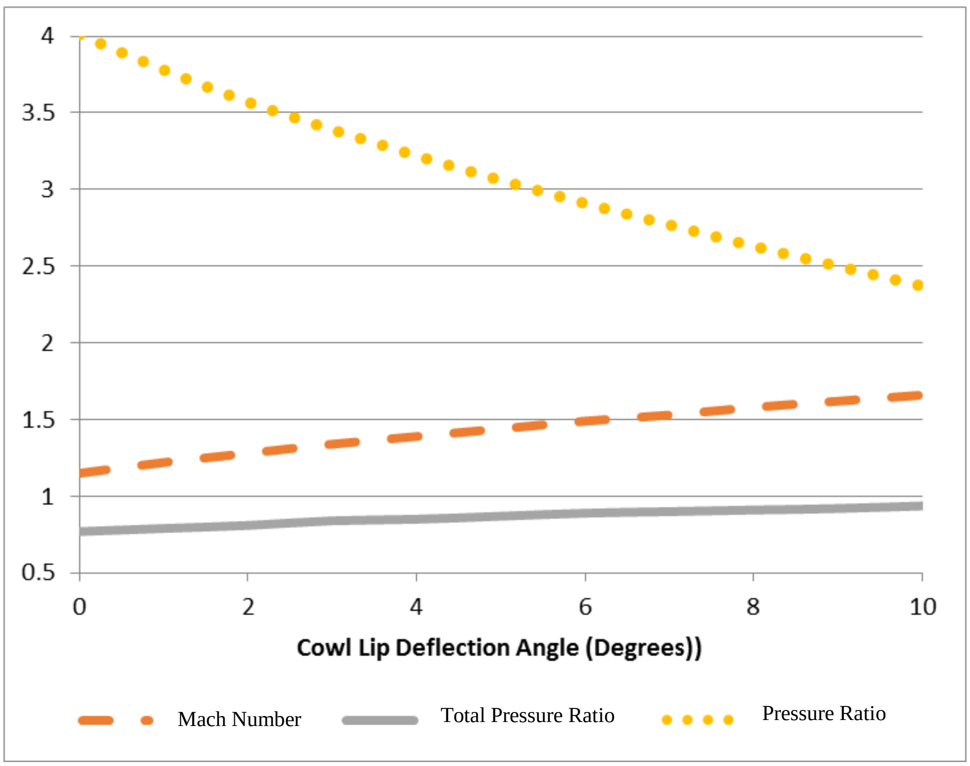
Cowl Lip Deflection Design

Cowl lip must be engineered as accurate as possible because wrong cowl lip design can produce normal shock at the intake entrance. The normal shocks would be very strong because the entrance flow is as high as Mach 2. It would lose a lot of total pressure and create high drag. The cowl lip also intitate the oblique shock in the internal compression system (Kopasakis, et al.).

Calculating shock in duct can be solved using 2D shock equation (equation 8) even though it is an axissymmetrical. It is because the movement of fluid is restricted and not as freely as at the cone section. The definition of cowl lip itself is shown below.



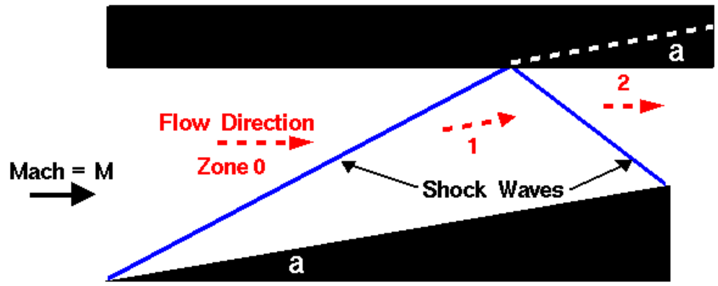
We vary our deflection angle first to see its performance trend. Graph below represents the cowl lip performance to the cowl lip deflection angle.



Deflecting cowl lip has its limit regarding intake geometry. It will increase intake diameter and it is very limited if we refer the DRO. There must be an iterative process after finding intake capture area. Therefore it must be assumed at first then adjust the deflection as needed. Here, we skipped the iterative process and got the optimum deflection angle at 6 degrees. At this step do not forget to adjust oblique shocks to meet at the cowl lip in the CAD drawing.

Duct Design

The internal compression occurs inside the duct. It implements shock reflection to create shocks system. It only occurs when the flow is deflected repeatedly. Therefore the duct cross sectional area is decreasing which also helps slowing down the flow. The picture below si the visualization of shock reflection.



Flow in Zone "1" is parallel to Wedge "a".
Flow in Zone "2" is parallel to wall.

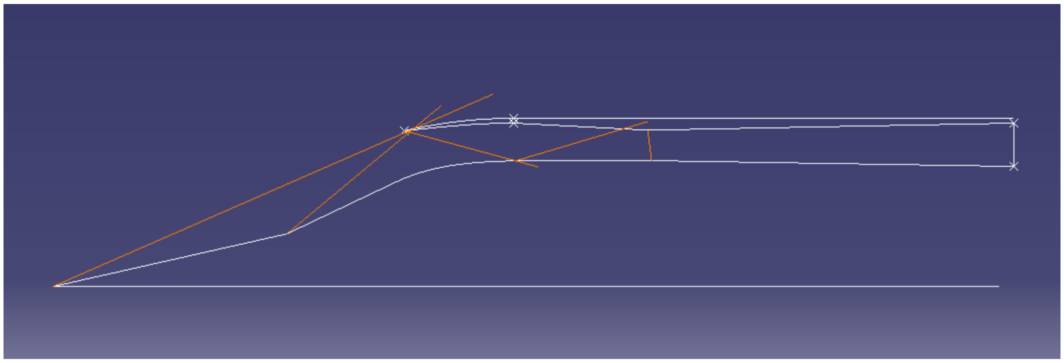
When the flow has passed the shock reflection system or when the mach number has become as low as Mach 1, there should be a terminal shock. The flow needs to be subsonic therefore we should create a normal shock in throat area. The downside is the terminal shock is considered as strong shock. Therefore we need the Mach number before terminal shock as close as possible to Mach 1 to minimize the total pressure loss. The equation below helps us to determine throat area (A*).

$$\left(\frac{A}{A^*}\right)^2 = \frac{1}{M^2}\left[\frac{2}{\gamma+1}\left(1+\frac{\gamma-1}{2}M^2\right)\right]^{\frac{\gamma+1}{\gamma-1}}$$

After the throat, we need a diffuser to keep the terminal shock in place and also to slow down the flow even further. The only requirement in designing the diffuser is to avoid separation. Table below is the summary of what we have designed from the cowl lip to the throat.

	After external compression properties	Cowl Lip	Reflected Shocks	Terminal Shock	Total
Deflection Angle	-	19.4^o	9^o	-	-
Oblique Shock Angle	-	40.51^o	46.78^o	-	-
Mach Number	-	1.6713	1.3566	0.7587	-
Total Pressure Ratio	0.9899	0.9368	0.9909	0.9683	0.8899
Static Pressure Ratio	2.0500	2.3792	1.5640	1.9805	15.1103
Static Temperature Ratio	1.2311	1.3051	1.1392	1.2268	2.2459
Density Ratio	1.6651	1.8230	1.3728	1.6143	6.7277

If we take a look at the final total pressure ratio, it still satisfies the MIL-E-5008B specification. The final static pressure ratio is quite high and the Mach number is still allowable. This design is accepted. Picture below is the final design of mixed compression intake complete with the predicted shock location.



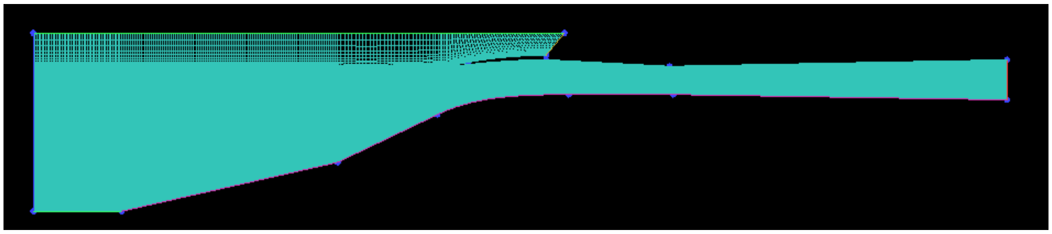
CFD ANALYSIS PROCESS

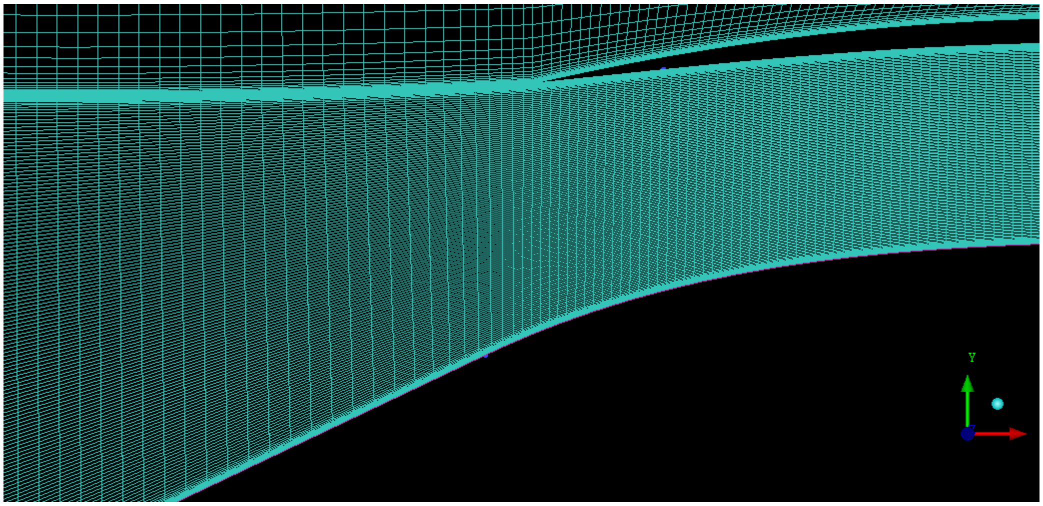
The design methodology must be validated through the CFD analysis. The CFD will show the actual flow condition around the intake. While the design methodology written above is still assumed as inviscid flow, CFD analysis will model the viscous flow. RANS method is used because it requires low computational cost and enough to represent the actual flow. The separation here is not the main focus, therefore the turbulent model can be modeled for this case.

Grid Generation

Analyzing performance for this intake will be conducted using ANSYS FLUENT Solver. Mesh generation otherwise, is created using ANSYS ICEM. The imported geometry to the ICEM must be given the boundary parts. Picture below illustrates each boundary part and its position.

The mesh for this analysis is using structured mesh to maximize mesh fidelity. The mesh size required for this simulation must be small enough to capture shocks and boundary layer. Shock boundary layer interaction is very important which might change the shock angle. To capture boundary layer, the first mesh on the wall must be calculated. Using boundary layer theories and which model will be used, the mesh size can be calculated and its first mesh thickness is 0.005938 mm. Picture below is the generated mesh which will be used for CFD analysis.





The total elements for the analysis are 250,000 elements. The mesh size for the internal flow is smaller than the external flow. It is because of shock reflection is expected to fill the duct and must be captured.

Solver Setup

For the solver configuration, two dimensional axissymmetrical is used. Double precision is used to increase its accuracy. Density based solver is selected because compressible cannot be neglected. It is set to be steady state solution. Since the solver using RANS, the turbulence model used is SST k-ω. This turbulence model can switch automatically whether to use k-ω or k-ε. The k-ω turbulence model is useful to model eddies where the k-ε ise useful to model turbulence in boundary layer. Energy equation must be turned on since we expect shocks to appear. The boundary conditions in critical condition are given below:

- Inlet (pressure inlet):
  - Gauge total pressure: 833832 Pa
  - Initial gauge pressure: 22700 Pa
  - Turbulent intensity: 1%
  - Total temperature: 606 K

- Outlet (pressure outlet):
  - Gauge pressure: 22700 Pa
  - Backflow total temperature: 606 K

- FF (pressure far-field):
  - Gauge pressure: 22700 Pa
  - Mach number: 3
  - Temperature: 223.25 K

- Bpressure (pressure outlet):
  - Gauge pressure: 343000 Pa (will be varied)
  - Backflow total temperature: 606 K

- Axis (axis)

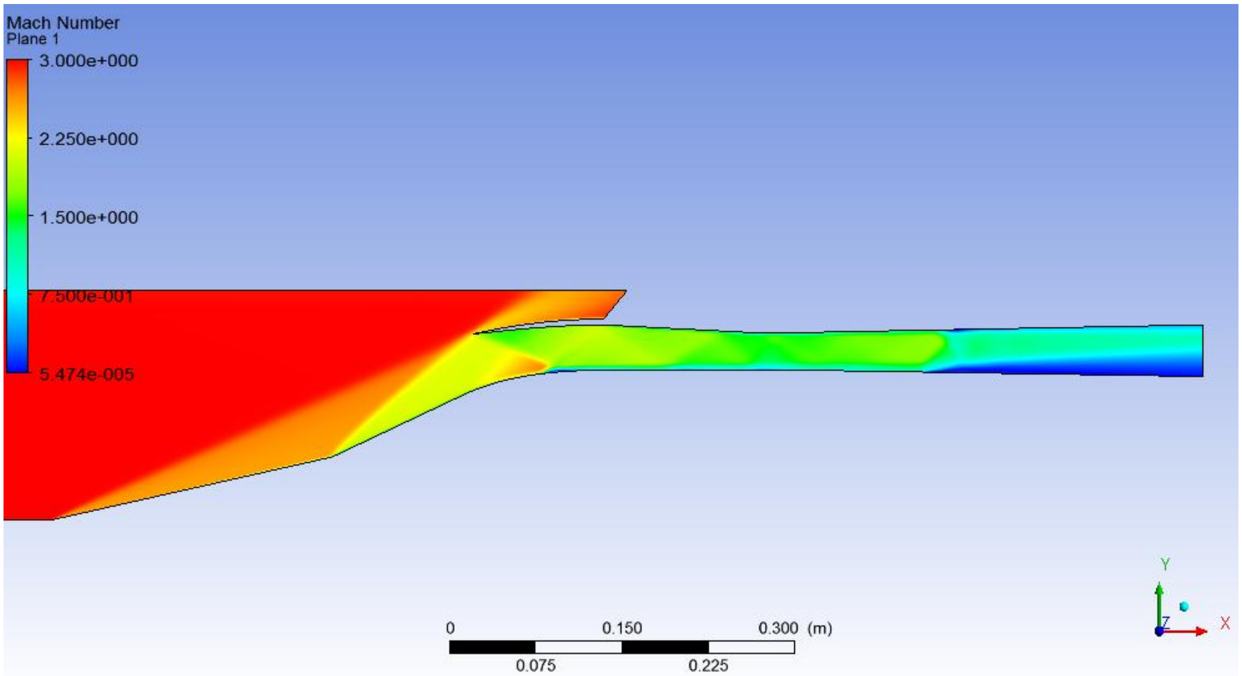
- Center & cowl (wall): no slip wall and smooth surface

Since the anlysis is in steady state and terminal shock is not steady over time, the simulation must be varied for pressure at engine face. Therefore the gauge pressure at bpressure boundary condition will be varied between 337,000 Pa to 350,000 Pa in this steady state simulation.

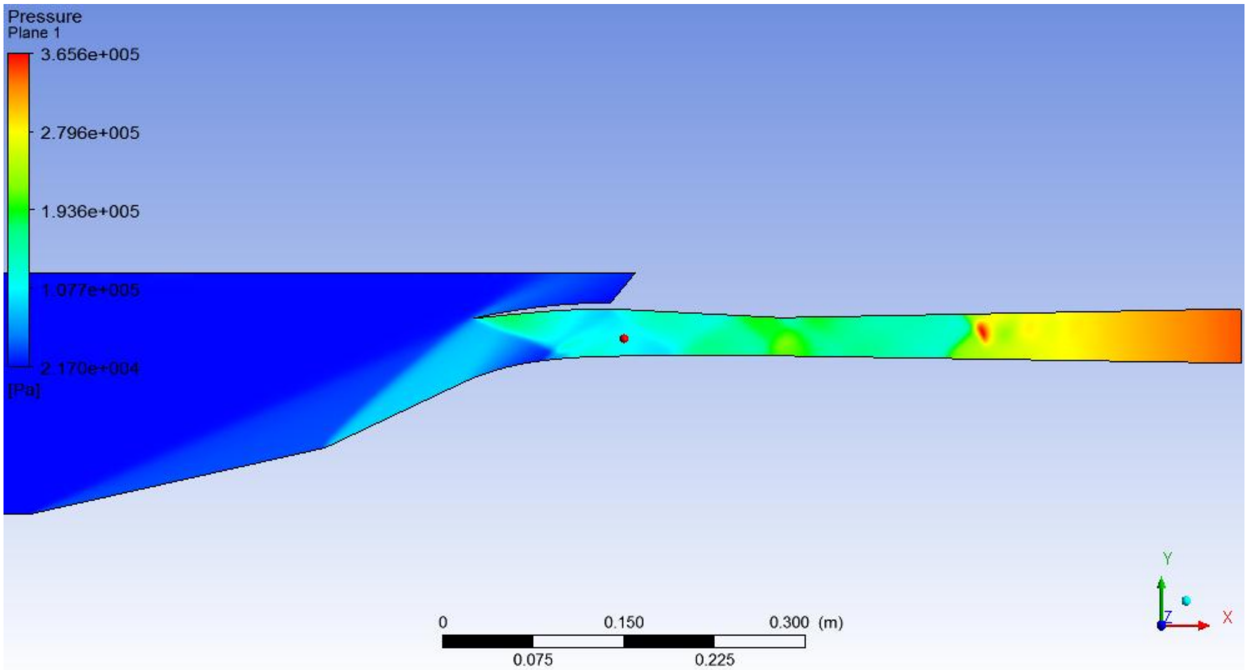
The convergence criterias are when all scaled residual RMS reach less than 0.0001 and the average Mach number does not change for 500 iterations. Solution steering is used to shorten iteration time therefore the Courant number is controlled. Where the Courant number is set less than 0.5 at the beginning of iteration and increase it to 0.99 when the scaled residuals become more stable.

Post-processing and CFD Results

Backflow Static Pressure 343,000 Pa



Mach number contour for M 3.0 freestream and 343,000 Pa of pressure at the engine face condition



Static pressure contour for M 3.0 freestream and 343,000 Pa of pressure at the engine face condition

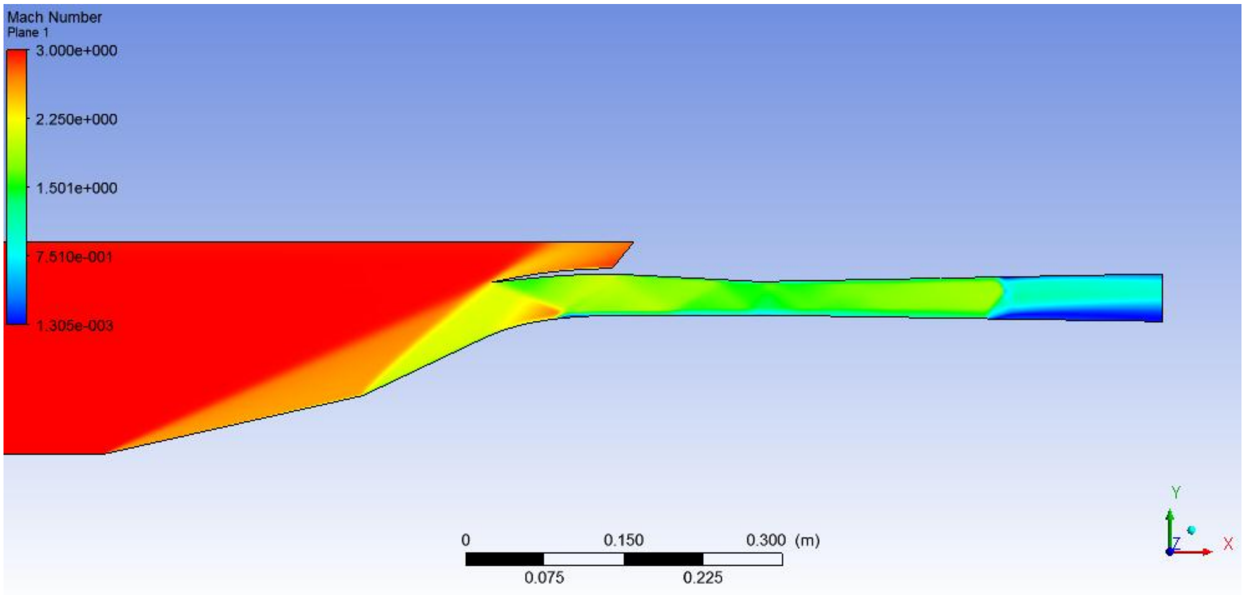
At the engine face	Mach Number, M	Static Pressure, P (Pa)	Total Pressure, Po (Pa)	Static Temp. , T (K)	Total Temp. , To (K)	Density (kg/m^3)	MFR (kg/s)
Results	0.49	343,000	694,600	564.5	602.9	2.123	26.15

Table of flow properties at the engine face for M 3.0 freestream and 343,000 Pa of pressure

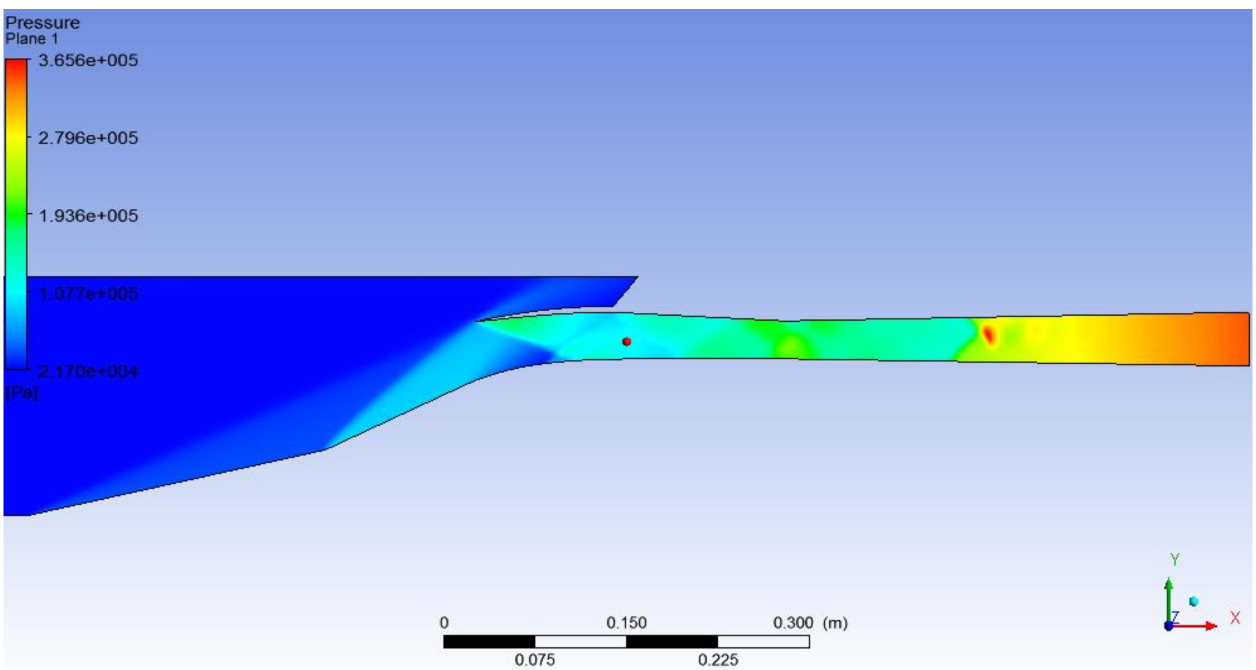
The boundary condition in this simulation case is the same as the boundary condition calculated from analytical solution. At the external flow, or at the bicone, the shock generated is as predicted before. Both shocks met at he cowl lip which create the critical condition.

At the internal compression area, it is not the exact same as the analytical solution. There are boundary layer shock interaction and expansion fan inside. Oblique shock generated from the cowl lip is the same with analytical solution, but the reflected shock is missed. The reflected shocks disturbed by boundary layer generated mainly from expansion fan. Therefore the reflected shocks here is more than what have been predicted before and the terminal shocks appear after the throat. Since the terminal shock appea at the diffuser which the cross sectional area increasing, the mach number is also increasing which makes the terminal shock stronger.

Backflow Static Pressure 350,000 Pa



Mach number contour for freestream M 3.0 and 350,000 Pa of pressure at the engine face condition



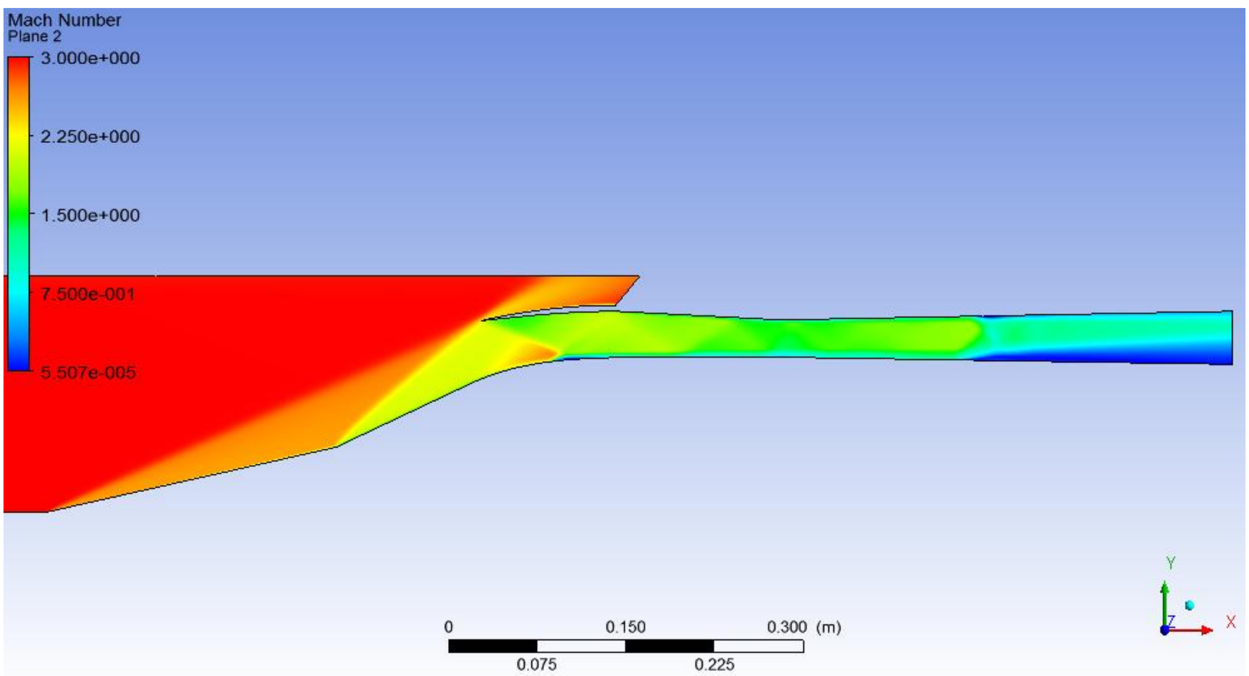
Static pressure contour for M 3.0 at freestream and 350,000 Pa of pressure at the engine face condition

At the engine face	Mach Number, M	Static Pressure, P (Pa)	Total Pressure, Po (Pa)	Static Temp. , T (K)	Total Temp. , To (K)	Density (kg/m^3)	MFR (kg/s)
Results	0.45	350,000	645,200	577.6	613.5	2.123	24.96

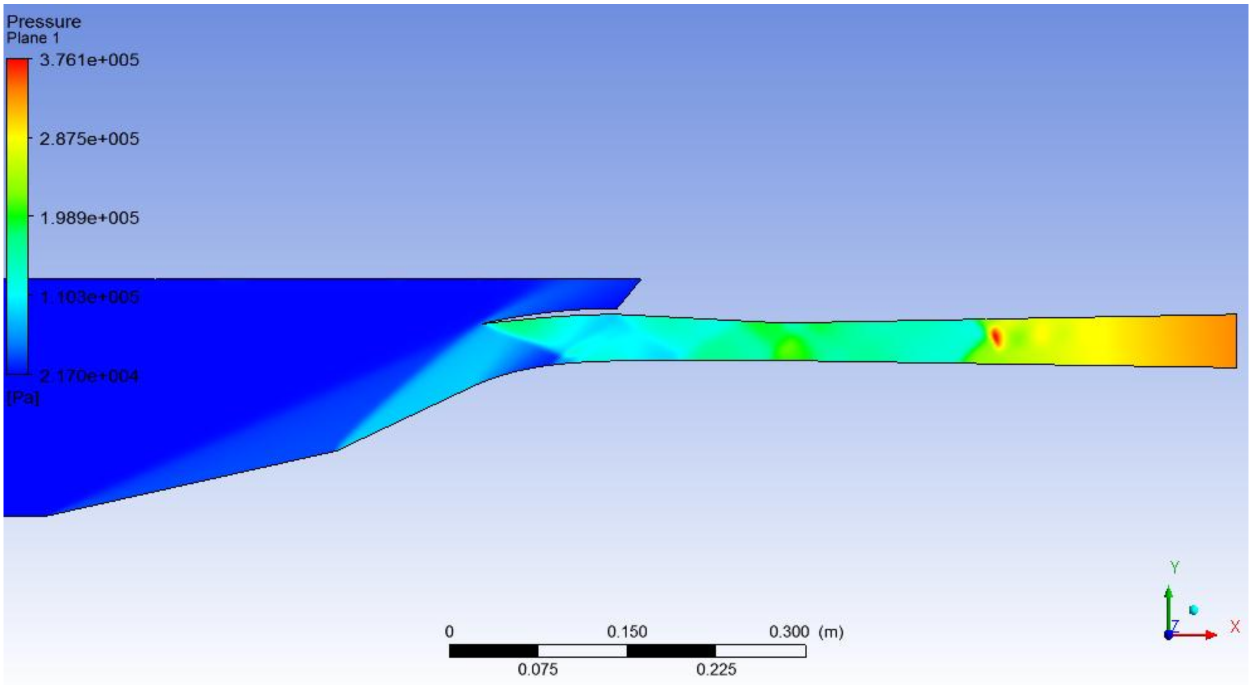
Table of flow properties at the engine face for M 3.0 freestream and 350,000 Pa of pressure

If compared to the 343,000 Pa case, there is a visible difference at the terminal shock position. The terminal shock is getting close to the engine face. It is accepted because when the terminal shock is getting further from the throat, the mach number before the terminal shock is getting higher. Therefore the terminal shock becom stronger which will decrease total pressure recovery. Then, this is not a better condition compared to the 343,000 Pa one. There are no difference at the external compression area.

Backflow Static Pressure 337,000 Pa



Mach number contour for freestream M 3.0 and 337,000 Pa of pressure at the engine face condition



Static pressure contour for M 3.0 at freestream and 337,000 Pa of pressure at the engine face condition

At the engine face	Mach Number, M	Static Pressure, P (Pa)	Total Pressure, Po (Pa)	Static Temp. , T (K)	Total Temp. , To (K)	Density (kg/m^3)	MFR (kg/s)
Results	0.49	337,000	692,600	563.3	602.9	2.091	26.16

Table of flow properties at the engine face for M 3.0 freestream and 337,000 Pa of pressure

For the lower pressure relative to the optimum condition at the engine face, the terminal shock is slightly getting further compared to the optimum condition. Therefore the total recovery is slightly lower. It proves that as the terminal shock getting further then the total pressure recovery will get lower. As for the external compression, it is as good as the optimum condition. Then, the pressure change at the engine face doesn't affect the external compression performance.

CONCLUSION

The design mehodology has been made and proven to be valid. From the CFD analysis we can concluded the shocks generated is similar to the predicted shocks. The analytical solution is accurate enough except for the total pressure. It is because of the analytical solution still assumed as inviscid solution while the simulation is already using viscous model. The static pressure variation at the back pressure shows us that the terminal shock is not always in one place. It's moving over time. Therefore, transient simulation must be conducted for better results and shows the shocks vibration. Also, these results show us that the analytical calculation gives the optimum condition where the total pressure recovery is the highest.

Since the design methodology is still in inviscid condition, the shock boundary layer interaction cannot be predicted. The impact from this interaction is quite significant especially for a design that does not require a supporting system. There must be a research or experiment for creating a formula to adjust the total pressure loss due to boundary layer and separation. As for diffuser, it needs a further research because diffuser's role is to keep the terminal shock in place. Diffuser must distribute the mass flow evenly at the engine face to increase combustion chamber efficiency.

REFERENCES

1. Anderson, J D. Jr.: "Fundamental of Aerodynamics", 2nd ed; McGraw-Hill Book Company, New York, 1991.
2. Anderson, J D. Jr.:"Computational Fluid Dynamics – The Basic with Application", McGraw-Hill Inc, New York, 1995.
3. Anderson, J D. Jr.:"Modern Compressible Flow with Historical Perspective", McGraw-Hill Inc, New York, 1986.
4. Seddon, J.: "Intake Aerodynamics", Collins Professional and Technical Books, Great Britain, 1985.
5. Seddon, J.: "Practical Intake Aerodynamic Design", Setrite Typesetters Ltd, Great Britain, 1993.
6. Mattingly, Jack D.: "Aircraft Engine Design", 2nd ed; American Institute of Aeronautics and Astronautics Inc, Virginia, 2002.
7. Bhat, V.K, et. al.:"Performance of Fuel-Rich Propellant for Ramjet Applications", Defence Science Journal, 1996.
8. Walrup, Paul J., et al.:"History of Ramjet and Scramjet Propulsion Development for U.S. Navy Missiles", John Hopkins APL Technical Digest, 1997.
9. "GQM-163A Coyote Fact Sheet", Orbital Sciences Corporation, 2014.
10. Das, S., et al.,"Cowl Deflection Angle in a Supersonic Air Intake", Defence Science Journal, 2009.
11. Chyu, W. J., et al., "Calculation of External-Internal Flow Fields for Mixed Compression Inlets", National Aeronautics and Space Administration, 1986.
12. Bowditch, David N., "Some Design Considerations for Supersonic Cruise Mixed Compression Inlets", National Aeronautics and Space Administration, 1973.
13. Timnat, Y. M., "Recent Developments in Ramjets, Ducted Rockets and Scramjets", Pergamon Press, 1990.
14. Kopasakis, George, et al., " Quasi 1D Modeling of Mixed Compression Supersonic Inlets", America Institute of Aeronatics and Astronautics.
15. Oh, J. Y., Ma, F. H., Hsieh, S. Y., and Yang, V., "Interactions Between Shock and Acoustic Waves in a Supersonic Inlet Diffuser", Journal of Propulsion and Power, Vol. 21, No.3, pp. 486-495, 2005.
16. FLUENT Online Help Manual