Plasma Skimming Through Branched Microchannel Using OpenFOAM

Sahil Deepak Kukian

B.Tech in Aeronautical Engineering

Manipal Institute of Technology, Manipal

Abstract

Blood plasma extraction plays a big part in the medical industry as the need for fresh blood and also its plasma are always in huge demand. The plasma extracted can be used for various plasma therapy and disease testing procedures. Due to conventional methods being difficult to transport, new novel methods need to be developed for efficient extraction. One such method is the use of branched microchannels. Blood flowing through a microchannel can be considered a two-phase flow consisting of plasma and red blood cells (RBC's). Branching the microchannel allows plasma to be extracted from the blood in a process called Plasma Skimming. This effect utilizes the Zweifach-Fung effect, also known as the bifurcation law. Some factors that affect this function are going to be studied using single phase "nonNewtonianIcoFoam" solver in OpenFOAM. Once verified we can then move onto a more complex two-phase solver such as the " twoPhaseEulerFoam" to capture the plasma moving into the branch channel.

1. Introduction

The need for plasma skimming arises due to requirement of pure plasma for plasma therapy and testing of anti-bodies. Standard plasma separating equipment tend to be large, heavy and cannot be transported easily to other locations. This further increases the need for an easy to manufacture and simple to use plasma separator. The microchannel separator is not bulky and can be transported easily when required. The main principle that results in separation is called the Zweifach-Fung effect and was experimentally demonstrated in simple microchannels. The Zweifach-Fung effect describes that in microchannels, red blood cells tend to flow into the branch with the higher flow rate. This effect is also accompanied by a separation occurring at the wall that results in a plasma layer being formed that is free of red blood cells as the red blood cells tend to follow streamlines at the center Core region as shown in fig.1 below.

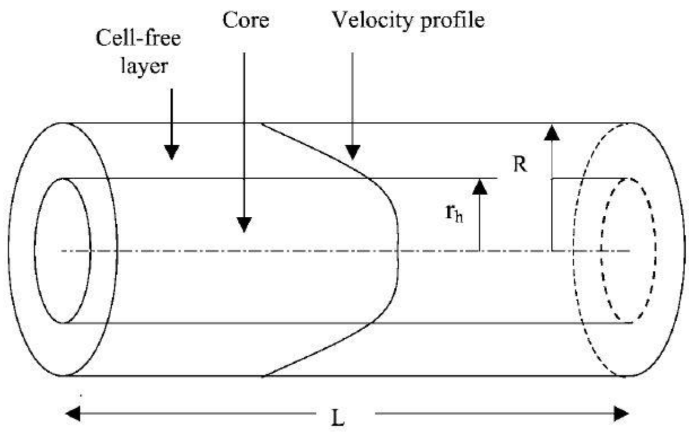


Fig 1 Diagram showing RBC core and Plasma Layer at Wall [3].

Some factors like change in the Plasma layer thickness due to change in Flow ratio, change in velocity after the bifurcation etc. will also be investigated.

Fig. 2 below will give us a better understanding on how the plasma separates from blood and enters into the branch, the blue colored balls represent the plasma and the red colored balls represent the red blood cells.

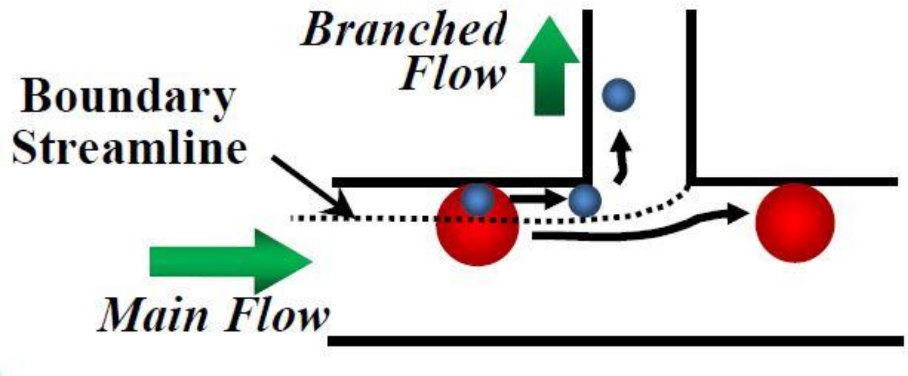


Fig 2 Plasma Separation in the Branch[1].

2. Problem Statement

First, we are going to simulate the behavior of blood as a single phase by using a solver that accounts for the non-Newtonian behavior of blood and apply a model called Casson model to sufficiently capture the behavior. We are going to be using the 'nonNewtonianIcoFoam' solver to try and study the behavior of blood as a single fluid and see if it behaves similarly to what has been observed in the paper by K. Morimoto et al [1]. Once the case is verified, we are then going to study the behavior in a two-phase simulation using "twoPhaseEulerFoam".

3. Governing Equations

The flow model that is applied to the fluid in the single phase solver is the Casson model and this is represented by the equation shown below

$$v = \left(\sqrt{{\tau_0}/{\gamma}} + \sqrt{m}\right)^2 \quad v_{min} \leq v \leq v_{max} \tag{1}$$

where γ is the shear rate

τ_0 is the strain rate corresponding to threshold stress

m is the consistency index

v_{min} and v_{max} are the minimum and maximum viscosities.

The Velocity values have been calculated from the 0.06 µl/min flow rate condition (main channel flow rate) that has been considered in all situations using the flow rate equation 2 shown below.

$$Q = \rho \times A \times v \tag{2}$$

where ρ is density

v is velocity

A is the cross sectional area

Using equation 3 we can back calculate out the velocity at the inlet and plasma outlet respectively. This will be used for the two flow ratio conditions that we have considered

$$Flow\ Ratio = {Q_{branch}}/{Q_{Main}} \tag{3}$$

where Q_{branch} is the flow rate at the branch

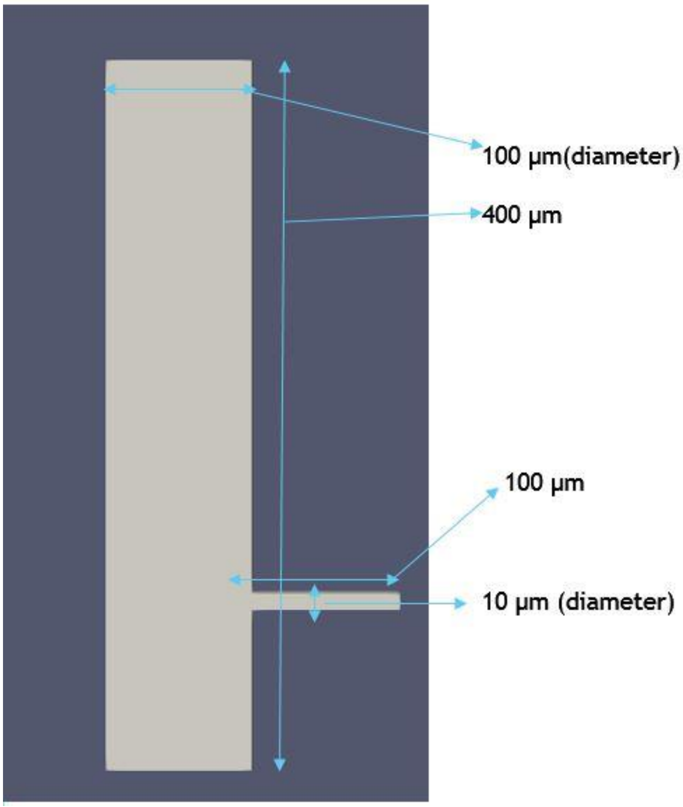
Q_{Main} is the flow rate at the main channel

We are going to be comparing the flow ratios 3 and 10 with the results from the K. Morimoto et al [1] and also from Yang Sung et al [2].

4. Case Setup

4.1 Geometry and Mesh

The geometry for the first case study consists of blood_inlet, plasma_outlet, blood_outlet, wall and frontAndBackPlanes. The geometry is modelled in ANSYS workbench in the 'µm' dimension as shown in fig 3. For meshing the model, ANSYS Meshing was used and a structured mesh was made using the face Meshing option. The mesh can be seen in fig 3. It consists of 6560 hexahedral elements. For the two-phase simulation, we are going to be using similar geometry as in fig 3, with the change that the inlet is split into two, consisting of blood_inlet and plasma_inlet. Also since the two-phase solver is more complex, the mesh is slightly coarser consisting of 1332 hexahedral cells.



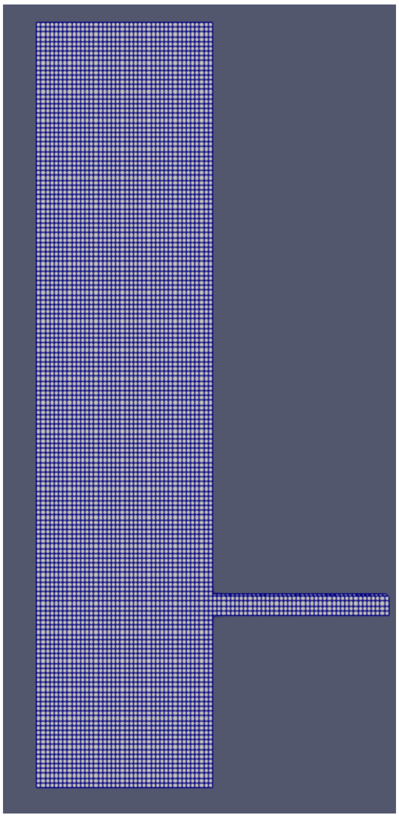


Fig 3 Geometry (left) and Mesh (right)

4.2 Boundary Conditions

The mesh needs to be converted from the .msh file to a format that is readable by OpenFOAM. To do this, the "fluentMeshToFoam" command is used. This will create the polyMesh folder. In the boundary dictionary inside the polymesh folder, the named selections are changed to patch type, walls to wall type and frontAndBackPlanes to empty type.

The boundary conditions used for the patches are as shown below in Table 1.

Boundary Name	U	p
inlet_blood	fixedValue	fixedValue
outlet_blood	fixedValue	zeroGradient
outlet_plasma	fixedValue	totalPressure
wall	noSlip	zeroGradient
frontAndBackPlanes	empty	empty

Table 1 Boundary conditions nonNewtonianIcoFoam

For the two-phase case (Case 3), named selection are changed to patch type, walls to wall type and frontAndBackPlanes to empty type after converting mesh to OpenFOAM format. Also for the boundary conditions for the 'twoPhaseEulerFoam' solver, since there are 8 different property files, we are going to be focusing on the 4 main ones that have most impact on the results.

Boundary Name	U.air	U.water	Alpha.air	P_rgh
inlet_blood	fixedValue	fixedValue	fixedValue	fixedFluxPressure
Inlet_plasma	fixedValue	fixedValue	fixedValue	fixedFluxPressure
outlet_blood	zeroGradient	FixedValue	zeroGradient	prghPressure
outlet_plasma	fixedValue	zeroGradient	inletOutlet	prghPressure
wall	zeroGradient	fixedValue	zeroGradient	fixedFluxPressure
frontAndBackPlanes	empty	empty	empty	empty

Table 2 Boundary conditions twoPhaseEulerFoam

4.3 Solver and Simulation Controls

For the first test 'nonNewtonianIcoFoam' solver is going to be used with the Casson model which is set in the thermophysical properties dictionary. The simulation is a laminar simulation. The time step selected for the single phase model is 1e-7.

For the second test the "twoPhaseEulerFoam" is using the laminar type with no special model applied. The time step selected for the two-phase model is 2e-5.

5. Result and Analysis

5.1 Case 1 – Flow ratio 3

In fig.4 we can see the velocity magnitude streamlines, from this plot we can see the streamlines are entering into the branch channel and the thickness of the layer that enters into the channel is approximately 20.5 µm. Also from fig.4 we can see the change in U magnitude which is measured across the main pipe before the bifurcation and after the bifurcation.The curve on the top (brown color) represents the velocity magnitude before the bifurcation and the bottom curve (blue color) is representing the velocity magnitude after the bifurcation. A significant drop in velocity is observed as the fluid flows past the bifurcation, this suggests that fluid is flowing into the branch channel.

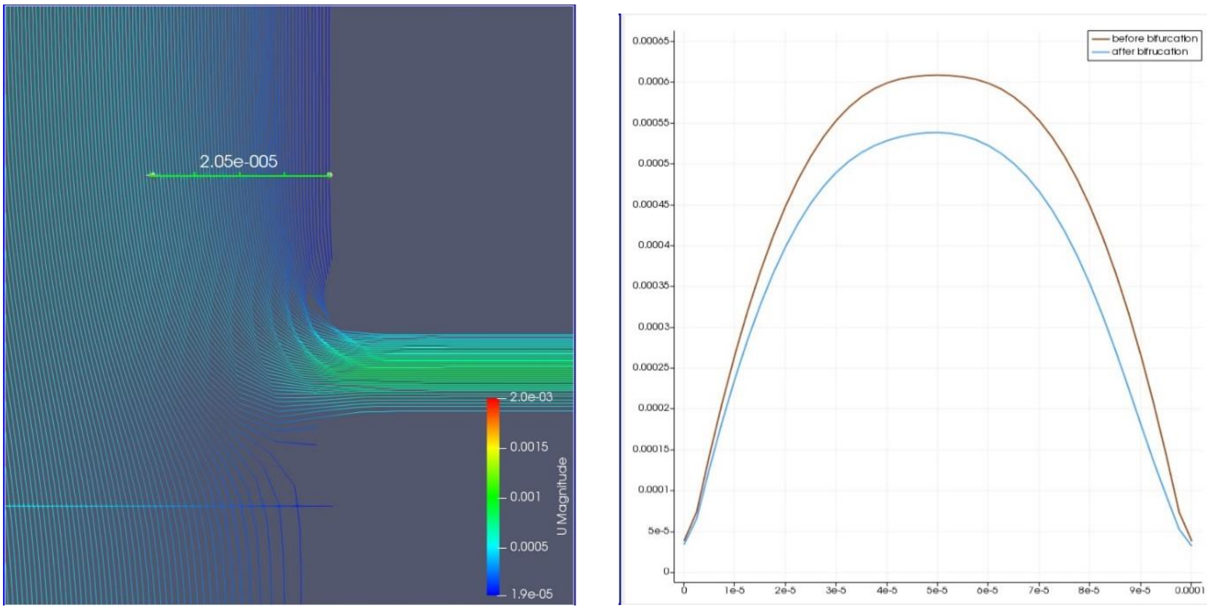


Fig 4. Flow ratio 3 U magnitude Streamlines with plasma layer thickness (left), Change in U magnitude before and after the bifurcation (right)

5.3 Case 2 – Flow ratio 10

In fig. 5 we can see the velocity magnitude streamlines, from this plot we can see the streamlines are entering into the branch channel and the thickness of the layer that enters into the channel is approximately 11.9 µm. Also from the U magnitude plot, we can see the change in U magnitude which is measured across the main pipe before the bifurcation and after the bifurcation. Similar behavior is seen here as the flow ratio 3 case, wherein there is a reduction in the velocity magnitude after the bifurcation.

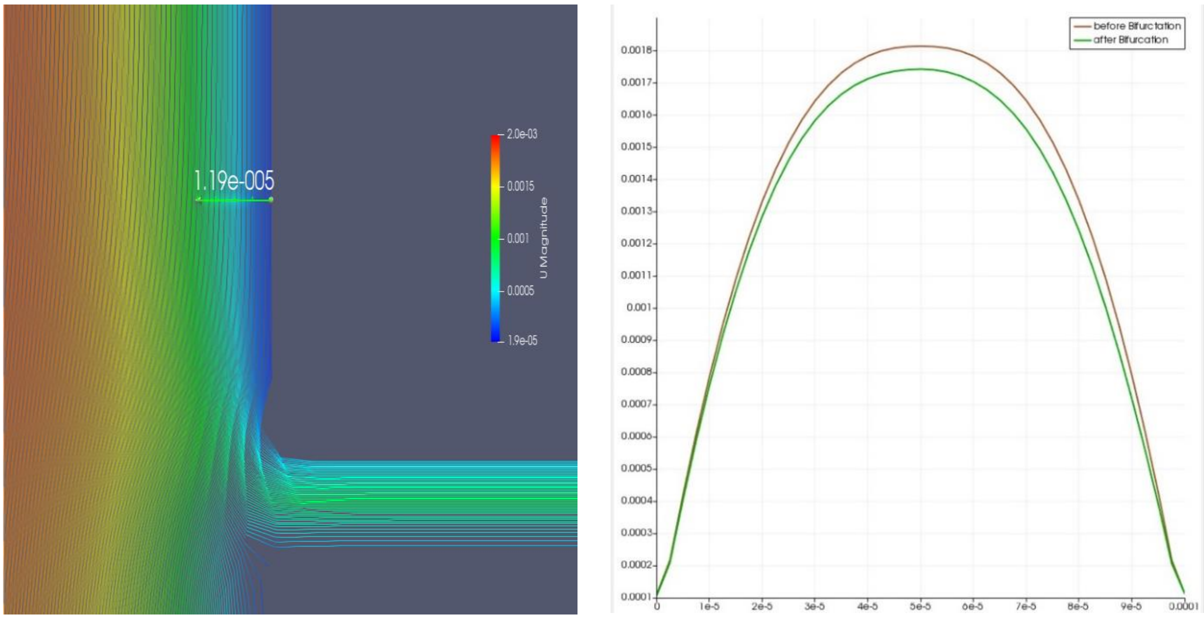


Fig 5. Flow Ratio 10 - U magnitude Streamlines with plasma Layer thickness (left), Change in U magnitude before and after the bifurcation (right)

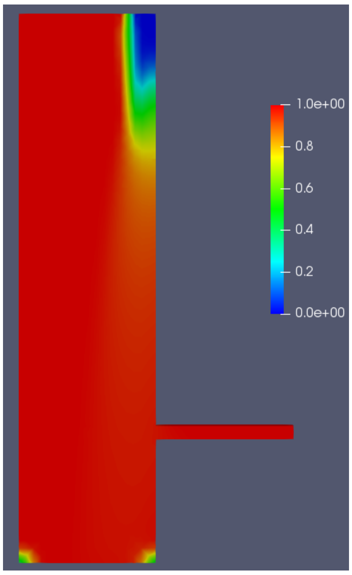
From the above contours and plots we can see that there is a reduction in the plasma layer thickness that is flowing into the branches as the Flow ratio is increased, this is similar to what was seen in experimental and simulation results conducted by K. Morimoto et al [1], although the values may not be an exact match. Another observation is that reduction in the velocity magnitude after the fluid has flowed past the bifurcation. This behavior has been also been observed in Yang Sung et al [2].

5.3 Case 3 – Two-Phase Case

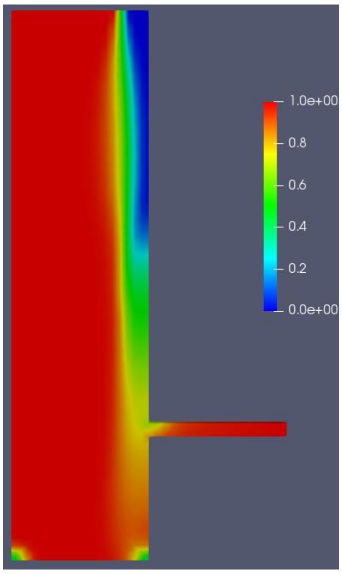
In fig. 6 and fig. 7 we can see the alpha contour (i.e. Phase fraction) where blue is represented by the plasma and the red is represented by blood. The case was run for 18.5 sec by which steady state was achieved. The case was run with the same velocity values as Case 2.

This case is meant to be a proof of concept, to simulate the flow behavior of the plasma going into the branch without much modification to the tutorial case files, therefore the properties of the two-phase have not been changed from air and water to blood and plasma.

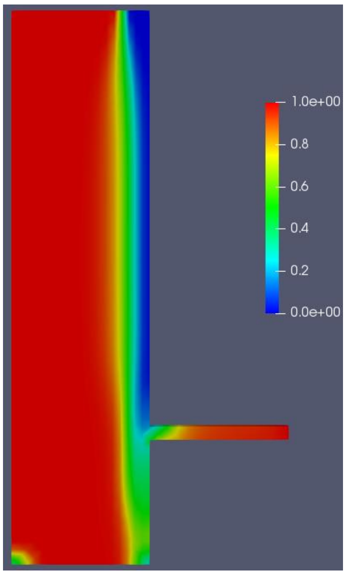
Some problems encountered during the simulation is that once the plasma reached the branch channel (at 0.5s) the simulation speed slowed down significantly such that it took more than 24hrs of computational time to reach 18.5 sec.



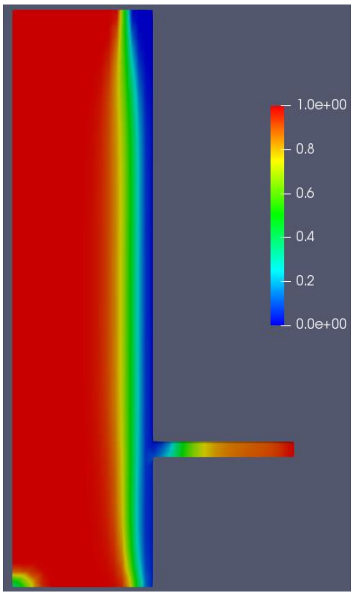
Alpha at 0.1s



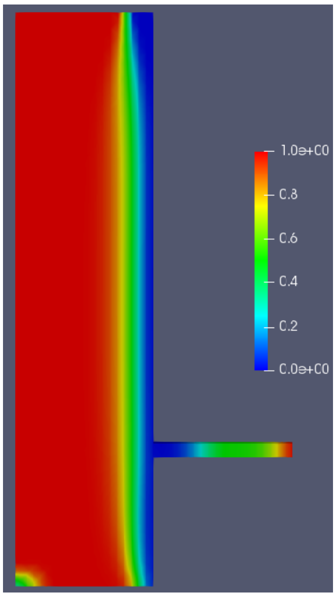
Alpha at 0.3s



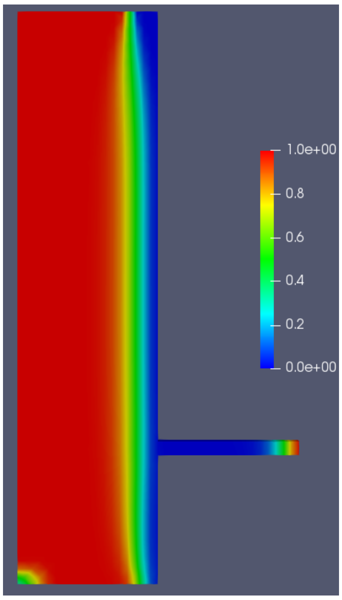
Alpha at 0.5s



Alpha at 1s



Alpha at 3s



Alpha at 10s

Fig 6. Alpha Distribution (Phase fraction) at various time-steps

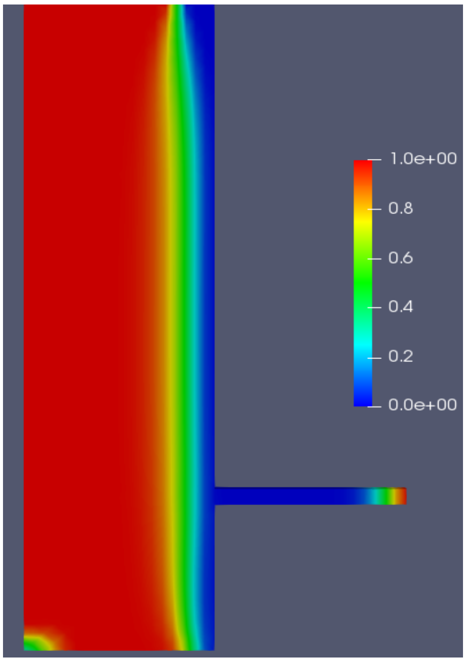


Fig 7. Alpha Distribution (Phase fraction) at Final time-step 18.5s

6. Conclusion

This case study has explored the project with the help of OpenSource solvers nonNewtonianIcoFoam and twoPhaseEulerFoam. Exactly matching results were not obtained but comparable observations like the reduction of the plasma layer with variation of flow ratio when compared with K. Morimoto et al[1.] and reduction of velocity magnitude before and after the bifurcation when compared with Yang Sung et al[2] . One similarity can be observed when we compare the thickness of the plasma layer in the Case 3 to Case 2, the plasma layer seems to have approximately the same thickness in both cases.

References

1. K. Morimoto et al (2007) in the paper titled “Numerical estimation of plasma layer thickness in branched micro channel using a multi-Layer model of Blood Flow”

2. Yang Sung et al (2006) in the paper titled “A Microfluidic device for continuous, real time blood plasma separation”

3. Maithili Sharan et al (2001) in the paper titled “A two-phase model for flow of blood in narrow tubes with increased effective viscosity near the wall”

FOSSEE, IIT Bombay
FOSSEE Fellowship 2020
June 20, 2020

A CFD study on 2D SCRAM jet intake using OpenFOAM

Sahil Deepak Kukian

B.Tech in Aeronautical Engineering

Manipal Institute of Technology, Manipal

Abstract

SCRAM jet engines are external compression engines used for hypersonic flight vehicles. They comprise of an inlet spike over which most of the compression takes place due to the formation of shockwaves and cowl that deflects shocks into the engine. Now that space exploration has matured, there is a need to study and develop faster methods of propulsion. In this study we are going to validate the results from K. Sinha et al. (2016), simulate the case at on-design Mach No. for Different Angles of Attack, and Compare the variation of pressure in the isolator region at different Angles of Attack.

1. Introduction

Supersonic flow is characterized as flow that is above 1.2 Mach. For this project we are going to study the shock interaction at High supersonic flows with Hypersonic Intake that is designed for optimum Operation at Mach 6.5. For this we use the rhoCentralFoam Solver and ParaView for the visualization. One of the major issues with solving solutions at such high velocities is that we then need to consider the viscous interaction effects and high Temperature effects which add another level of complexity to the solution. Although we will not be considering those effects in this study, it does play a major role in real world hypersonic aerodynamics. The Intake design we are going to be considering is similar to the Design from K.Sinha et al (2016)[1]. This design is known as mixed compression intake which is also the most widely used design due to the shorter length and lower Drag. The other intake types are External Compression intake and Internal Compression Intake.

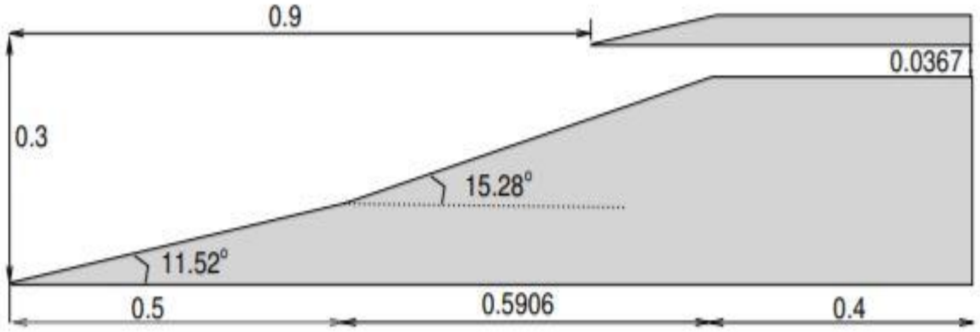


Fig 1. Hypersonic Intake Geometry [1]

2. Problem Statement

To Study a 2D SCRAMJET intake design at various Mach Numbers and Angles of Attack (AOA) using the compressible OpenFOAM solver “rhoCentralFoam”. The case is simulated at 26km altitude with temperature 219.3k and air density of 0.03436 kg/m^3.

3. Governing Equations

Conservation of Mass equation follows directly from the control volume equation, by applying Gauss Divergence theorem, we can transform the surface integral into a volume integral finally becoming the Equation shown below

$$\frac{\partial \rho}{\partial t} + DIV(\rho v) = 0 \tag{1}$$

The Inviscid Euler equation is given below

$$\frac{\partial(\rho u)}{\partial t} + \nabla(\rho u u^t) + \nabla p = F \tag{2}$$

Where ρ is density, p is Pressure

u is velocity

F is the volume Force

The energy equation is given below

$$\frac{\partial e}{\partial t} + \nabla.\big((e+p)u\big) = Q \tag{3}$$

Where e is the total energy per unit volume

u is velocity

p is the pressure

Q is the heat source

Using equation 4 we get pressure as 2130 pa. These parameters will remain the same for all the cases that are going to be run.

$$P = \rho \times R \times T \tag{4}$$

Where ρ is the density of air

R is the ideal Gas constant

T is the temperature

We are going to be evaluating the flow at various different Mach No. and comparing the performance parameters with the on-design parameter (i.e. Mach No 6.5).

Also to calculate the Velocity values at various Mach numbers we use the equations shown below. From equation 5, we can calculate the speed of sound.

$$a = \sqrt{\gamma \times R \times T} \tag{5}$$

Where R is gas constant (287 J/kgK)

T is temperature (K)

γ is Specific Heat ratio (assumed 1.3)

Once the Speed of sound is calculated, we use Equation 6, shown below to calculate the velocities at their respective Mach numbers.

$$V = M \times a \tag{6}$$

Where M is Mach number

a is speed of sound (m/s)

4. Case Setup

4.1 Geometry and Mesh

The SCRAM jet intake geometry consists of inlet, outlet, top, spike, cowl and oulet_spike. The total length of the model is 1.4906 m and 0.3 m in breadth. The angle of the first wedge is 11.52^0 and second wedge is 15.28^0 as shown in Fig 1. Since we want to simulate only the 2D simulation for this case but OpenFOAM operates only in 3D, so we assign a thickness of 0.035 m. The mesh can be seen in Fig 2. It consists of 110000 hexahedral Mesh elements made using Ansys meshing tool and the mesh has been designed in such a way to capture the oblique shocks and also the region inside the cowl. The mesh was exported into .msh format and then converted into OpenFOAM readable mesh by using the built-in function “fluentMeshToFoam “.

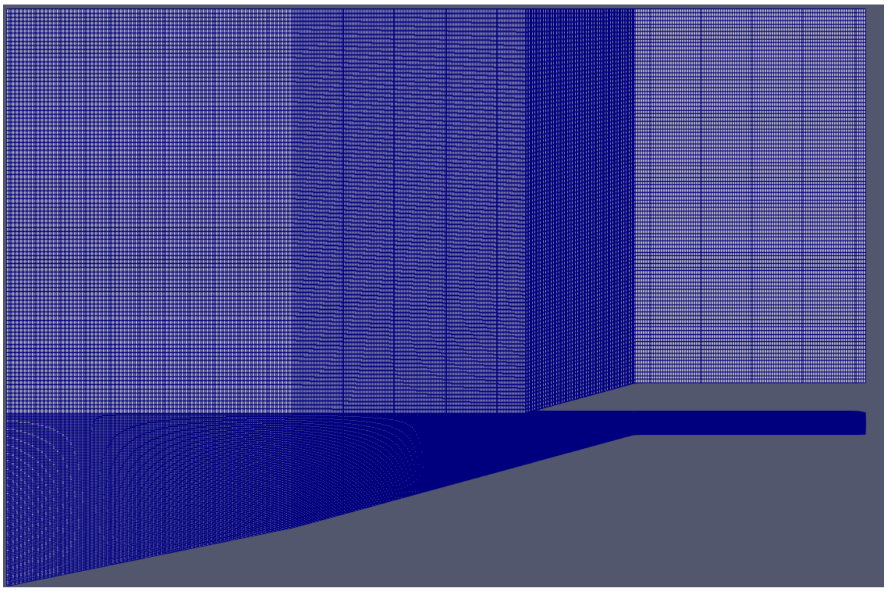


Fig 2. Mesh Region

4.2 Boundary Conditions

The boundary conditions used for the patches are as shown below in Table 1.

Selecting boundary conditions was one of the most difficult part of the simulation as incorrect selection will lead to the model diverging and not giving a good result.

Before being able to make the changes in the boundary conditions the necessary changes to the top, spike, outlet, spike_outlet and cowl must be done in the polyMesh folder after importing the mesh into OpenFOAM format. The frontAndBackPlanes must be changed into empty. All the others should be changed to patch. Inlet velocities are changed according to the Mach No to be simulated.

The Temperature at inlet is 219.3K and the Pressure is 2162pa.

Boundary Name	U	T	P
inlet	fixedValue	fixedValue	fixedValue
outlet	supersonicFreeStream	inletOutlet	waveTransmissive
top	supersonicFreeStream	inletOutlet	zeroGradient
outlet_spike	zeroGradient	zeroGradient	zeroGradient
cowl	slip	zeroGradient	zeroGradient
spike	slip	zeroGradient	zeroGradient
frontAndBackPlanes	empty	empty	empty

Table 1 Boundary Conditions

4.3 Solver and Simulation Controls

There is no special Turbulence model applied to this simulation. So in the turbulence type dictionary it is set to laminar.

As for the thermophysical properties, we are going to be using a mixture model with the properties as set in the dict file.

5. Result and Analysis

5.1 Pressure Contours at various Mach No.

In Fig 3 we can see the pressure contour comparing the result with literature. From Fig 3 we can see the two oblique shocks intersect at the tip of the cowl and gets reflected into the isolator region of the engine. This condition is known as the Shock-on-lip condition. This shows the pressure contours at Mach 6.5 which is the On-Design condition.

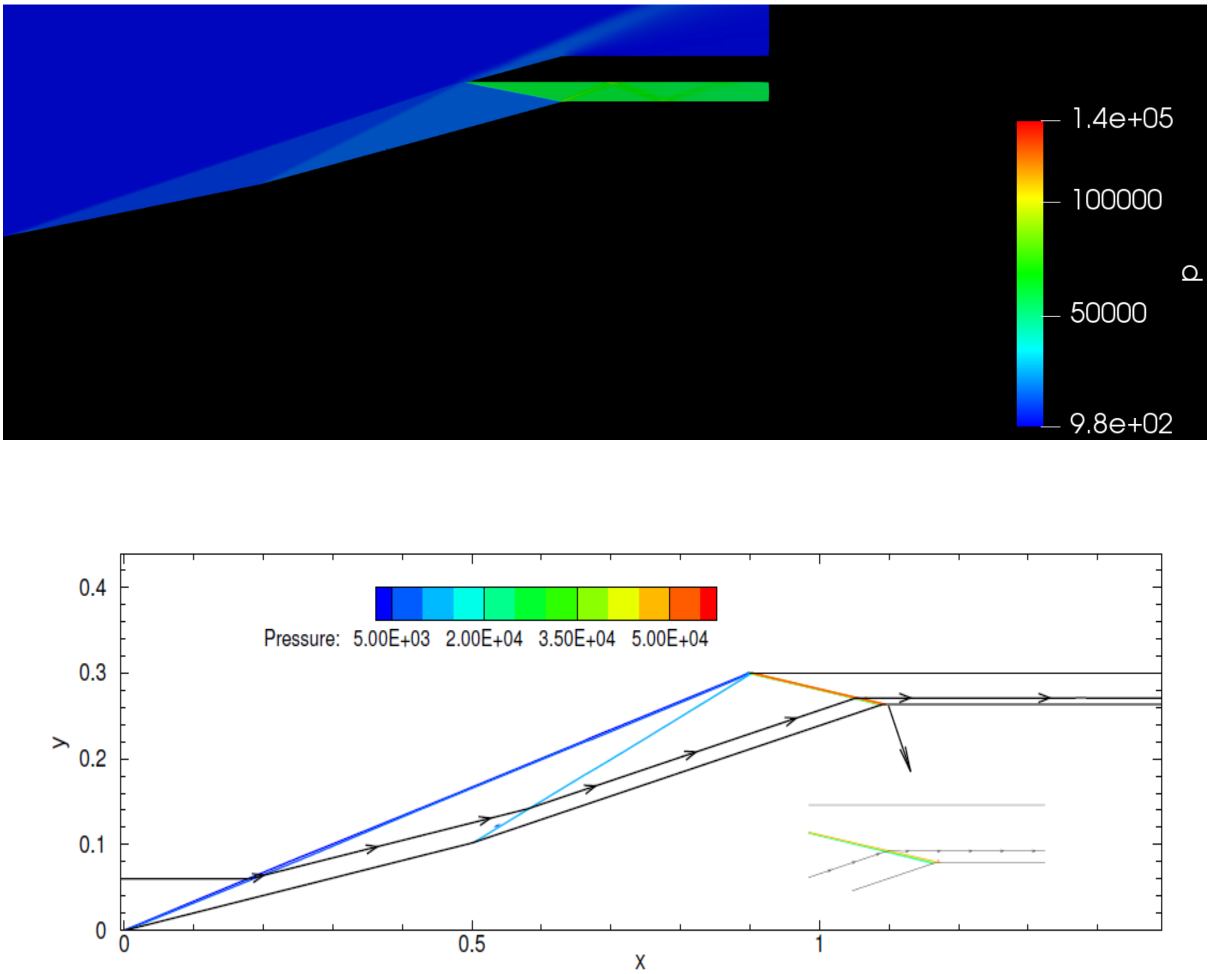


Fig 3. Simulated Pressure contour at Mach 6.5 (top)

Pressure contour at Mach 6.5 from literature [1] (bottom)

The pressure contours for the off-design conditions are given in fig 4. The comparisons will be clearer if we view the results in a table with the values of Mach No. as shown in the table 2.

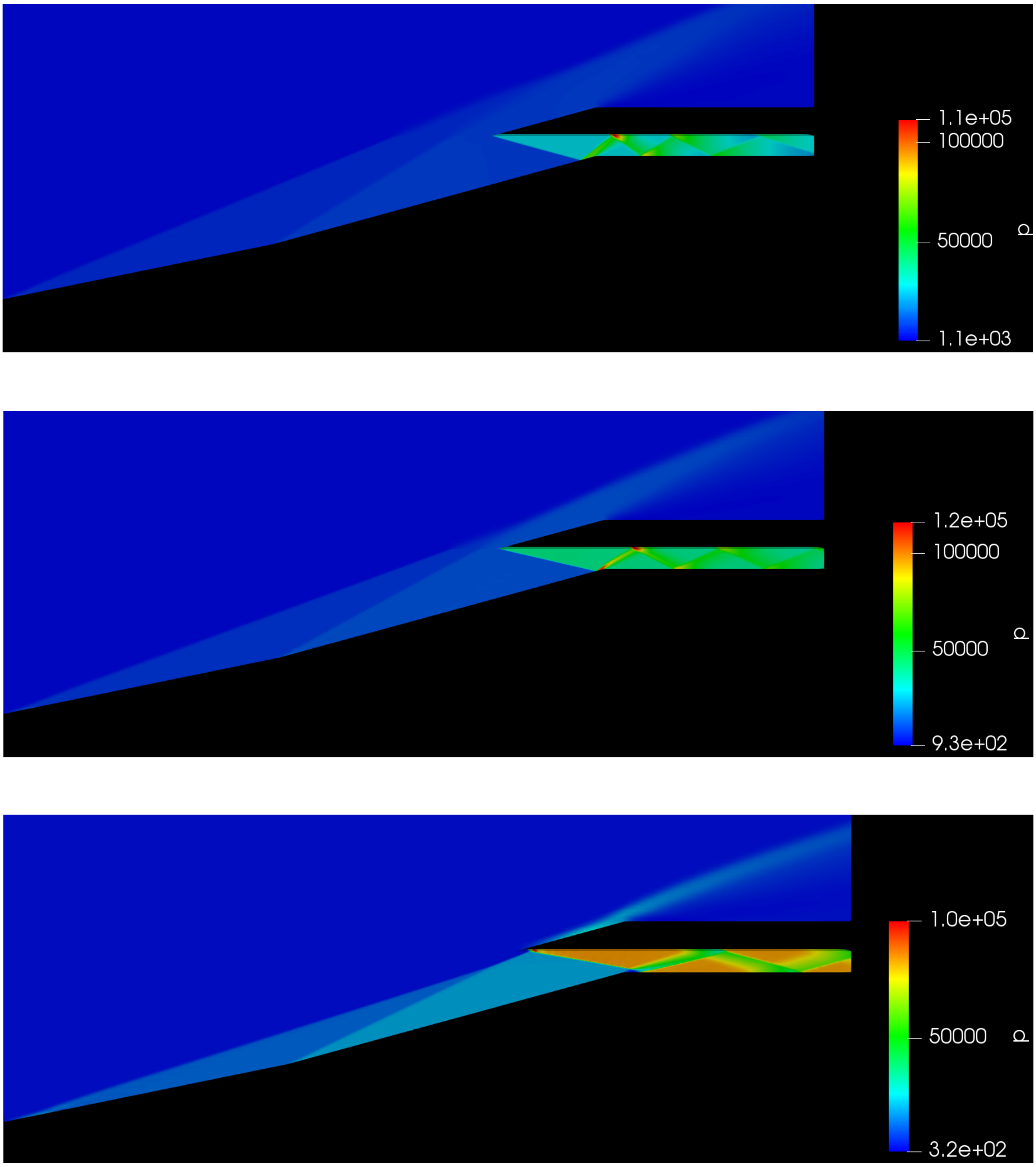


Fig 4. Pressure contour at various off-design Mach No.

Mach 4.5 (top), Mach 5.5 (middle), Mach 7.5 (bottom)

From Fig 4, we can see the variation of the pressure contour inside the isolator region as Mach No. is varied. This leads to uneven distribution inside due to the reflected shock waves. At lower Mach Numbers(Mach 4.5,Mach 5.5), the shocks formed by the two compression wedges does hit the cowl wall and thus leads to reduction in capture area and at higher Mach number, the shocks intersect and hit the cowl resulting in reflected shock waves continuing throughout the isolator region.

In table 2 below, the Mach No. inside the isolator region is compared with the results obtained in literature [1] and the error percentage between the simulated results and literature is calculated (given in brackets).

Mach No.	Mach No. $_{isolator}$ (Error %)
4.5	2.5650 (4.69 %)
5.5	2.8376 (0.267 %)
6.5	3.1613 (1.324 %)
7.5	3.3521 (0.531 %)

Table 2 Mach No. inside isolator at various Free-stream Mach No.

5.2 Pressure Contours at different Angles Of Attack

The pressure contours at -2° and 2° Angle of Attack are shown below in Fig 5. The pressure at the isolator outlet is going to be viewed in table 3.

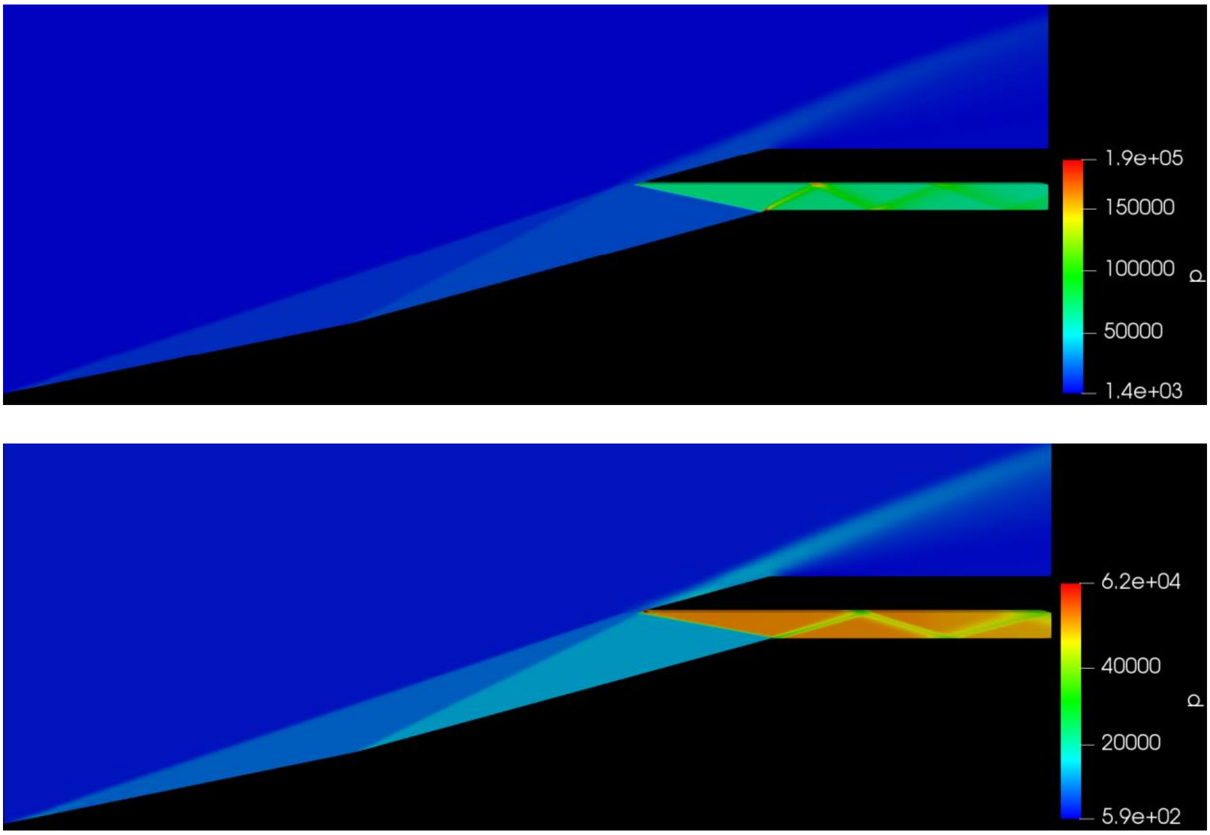


Fig 5 Pressure contour at -2 AOA (top), 2 AOA (bottom)

Angles of Attack	-2	0	2
Pressure(pa) isolator	67433.73	56570.59	47101.24

Table 3 Pressure variation (Isolator region) at different AOA

As Angle of attack increases the intersection points of the two shock formed by the wedges moves upstream and away from the cowl leading edge. This causes a reduced capture area resulting in a drop in the pressure in the isolator as shown in Table 3.

6. Conclusion

This case study has explored the SCRAM jet intake design that has been validated from K.Sinha et al (2016) at the Different Mach No. At Mach No. 6.5 and at an Angle of attack 0^0, the shock-on-lip condition is achieved that results in optimal Air capture Area. When the results obtained by ‘rhoCentralFoam’ was compared with the results from literature, we found that errors percentages were below 5%. This means that rhoCentralFoam was able to accurately simulate complex flows at high velocities. This error percentage can be further reduced by using a refined mesh, turbulence models etc. Also we have simulated the case at Different Angles of Attack (AOA) and from the result we can infer that increasing the AOA results in a reduction in the pressure inside the isolator region(as shown in table 3) thus reducing the efficiency of the intake.

References

1. Krishnendu Sinha, V. Jagadish Babu, Rachit Singh, Subhajit Roy, Pratikkumar Raje, Parametric Study of the performance of two-Dimensional Scramjet Intake, , 18th Annual CFD Symposium, August 10-11, 2016, Bangalore

2. Anderson, J.D., Modern Compressible Flow, McGraw Hill Inc., New York, 1984.

3. J. H Perziger , M .Peric , Computational Methods Of Fluid Dynamics ,Springer, ISBN 3-540-42074-6 Springer-Verlag Berlin Heidelberg NewYork