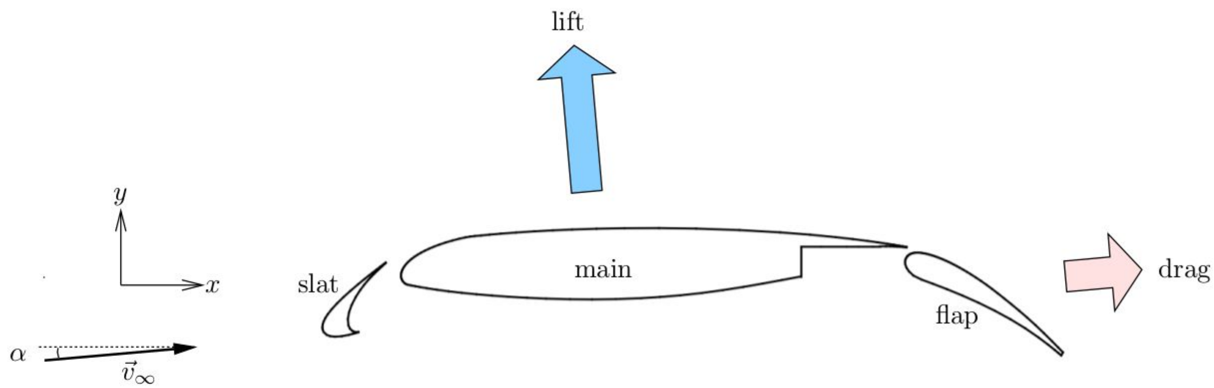
From: Luo, Xiao Kai
To: AE 523
Date: 11-30-2016
Subject: Project 3: Flow over a Three-Element Airfoil

Problem Description

A first and second-order finite volume method (FVM) needs to be implement to solve for the flow properties compressible, subsonic flow over a three-element, high-lift airfoil (see geometry of in Figure 1). Postprocessing calculations include coefficient of forces, field contour of pressure and and Mach, streamlines, and convergence over iteration plots.



Summary

First and second order finite volume methods were implemented on a three element high lift airfoil. Element-wise coefficients of lift was obtained. The other post-processings include: cp plot, pressure contour, Mach contour, streamline contour, and mash flow rate between gaps of the airfoil elements. It is found that the coefficient of lift values increased from first order solution to second order solution. It is with the second order solution that the coefficient of lift seems to convergence with element amount increase. It is found that the pressure is lower at the upper surface and higher on the bottom. Corresponding, the Mach is higher at the top and lower on the bottom. There is also mass flow between the gaps of the airfoil elements, which also increased between first and second order FVM. The streamline, however, looks the similar for all meshes and orders.

PART 1: PROBLEM SETUP, BOUNDARY CONDITIONS AND MESH

There four boundary conditions for the setup in Figure 1. Inviscid walls are used for the boundary of slat, main, and flap, the three components of the airfoil. For the fairfield conditions the following values were used:

Parameter	**Variable**	**Farfield Value (Freestream)**
Flow State	$\overline{n} = [\rho, \rho u, \rho v, \rho E]^T$	$\overline{n_\infty} = [\rho_\infty, \rho_\infty u_\infty, \rho_\infty v_\infty, \rho_\infty E_\infty]^T$
Density	ρ	$\rho_\infty = 1$
X-Velocity	u	$\rho_\infty u_\infty = M_\infty cos(\alpha)$
Y-Velocity	v	$\rho_\infty u_\infty = M_\infty sin(\alpha)$
Specific Energy	E	$\rho_\infty E_\infty = \frac{1}{(\gamma-1)\gamma} + \frac{M_\infty{}^2}{2}$
Mach	M	$M_\infty = 0.25$
Ratio of Specific Heat	γ	$\gamma = 1.4$ (same for freestream and inner flow)
Angle of Attack	α	$\alpha = 5^o$ (same for freestream and inner flow)
Chord	c	$c = 0.5588$ (same for freestream and inner flow)

Table 1. Parameters and their variables and values.

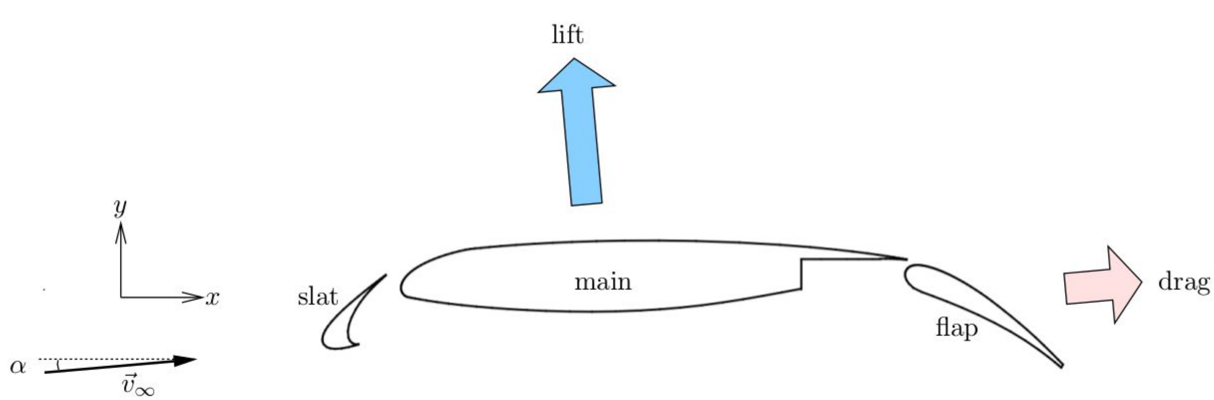


Figure 1. Geometry and setup of the three-element, high-lift airfoil.

There are four different meshes used and compared with for the FVM solvers. From Mesh 0 to 3, they have correspondingly 1149, 2116, 4124, and 8031 volume elements:

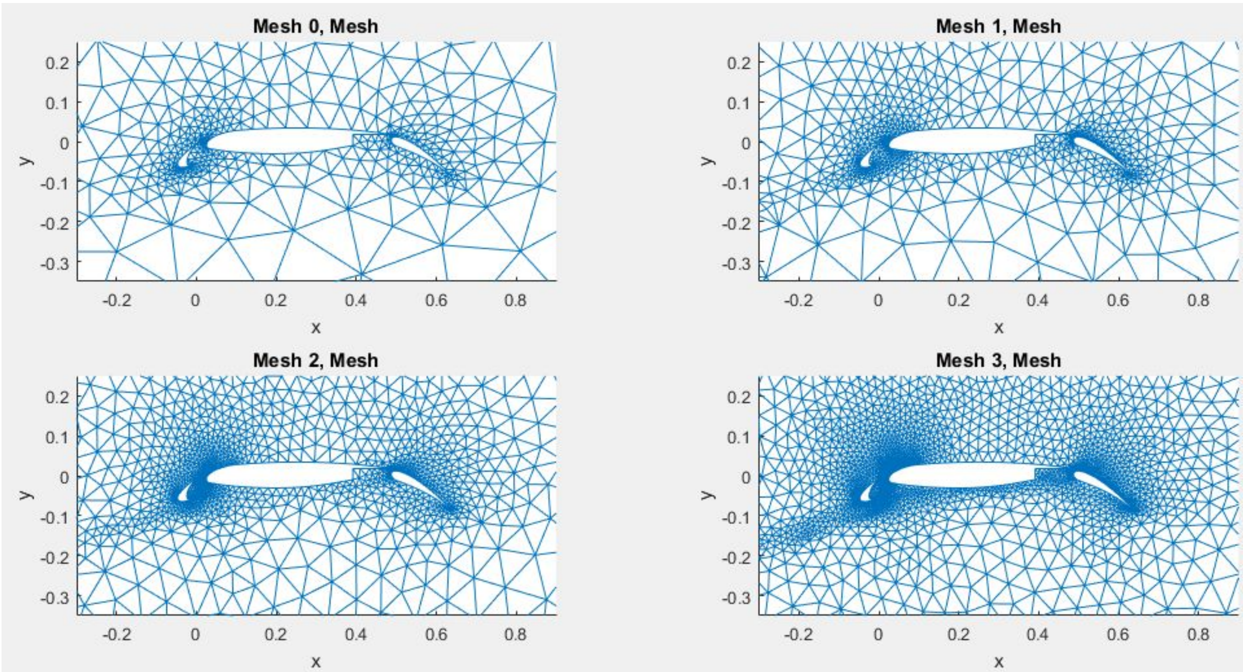


Figure 2. Mesh 0, 1, 2 3 with correspondingly 1149, 2116, 4124, and 8031 volume elements.

The Matlab code main1.m was used to read the mesh data given in the “meshes” folder after unzipping the givens.zip provided for this project. Then postproc.m was used to plot out the meshes shown above.

PART 2: FIRST-ORDER FVM

Flow State

FVM uses the follow general equations to update the flow state value, where A is the area, F is the flux given by flux.m, n is the normal, and dl is the edge length.:

$$A_i \frac{du_i}{dt} + \underbrace{\oint_{\partial A_i} \vec{F} \cdot \vec{n}\, dl}_{R_i} = 0,$$

(Eqn. 1)

$$R_i = \sum_{e=1}^{3} \hat{F}(u_i, u_{N(i,e)}, \vec{n}_{i,e})\, \Delta l_{i,e},$$

(Eqn. 2)

$$\begin{aligned} \Delta l_{i,e} &= \sqrt{(x_A - x_B)^2 + (y_A - y_B)^2}, \\ \vec{n}_{i,e} &= \left[(y_B - y_A)\hat{x} + (x_A - x_B)\hat{y}\right] / \Delta l_{i,e}. \end{aligned}$$

(Eqn. 3)

In flux.m, three inputs are required: left element edge state, right element edge state, and normal. The normal used in residual1.m was from the equation above. Since this is a first order method, the left and element state used for input to flux.m were the left and right cell averages.

For the First-Order FVM, the forward Euler (FE) was used for time stepping resulting the follow full equations:

$$A_i \frac{u_i^{n+1} - u_i^n}{\Delta t^n} + \underbrace{\sum_{e=1}^{3} \hat{F}(u_i, u_{N(i,e)}, \vec{n}_{i,e})\, \Delta l_{i,e}}_{R_i} = 0.$$

(Eqn. 4)

$$u_i^{n+1} = u_i^n - \frac{\Delta t^n}{A_i} R_i.$$

(Eqn. 5)

Local time step was used to acceleration the convergence. The CFL number used in main1.m was CFL = 1. This is chosen by trial and error. It is around the value of CFL = 1 which the flux.m didn't given the "unphysical state" error. The equations for the time stepping is follows:

$$\text{local } \Delta t_i = \frac{\text{CFL}\, d_i}{|\bar{s}|_i} = \frac{2A_i\, \text{CFL}}{\sum_{e=1}^{3} |s|_{i,e} \Delta l_{i,e}},$$

(Eqn. 6)

The freestream farfield boundary condition equations is as follows:

$$\hat{\mathbf{F}}^b = \hat{\mathbf{F}}(\mathbf{u}^+, \mathbf{u}^b, \vec{n}).$$

(Eqn. 7)

This is basically saying the left and right element states used involve the freestream state. Since in BE.m the left element is the one in the computational grid, the right element is assumed to be at freestream conditions. So in conclusion the left element state used the cell average of the element in the computational grid, whereas the right element state used the cell value of freestream.

The equations for the inviscid wall are as following:

$$\hat{\mathbf{F}}^b = [0, p^b n_x, p^b n_y, 0]^T, \qquad p^b = (\gamma - 1)\left[\rho E^+ - \frac{1}{2}\rho^+ |\vec{v}^b|^2\right], \qquad \text{(Eqn. 8)}$$

Instead of using flux.m, the inviscid flux was calculated direction. The pressure value is obtained from the density, specific energy, and velocity magnitude of the left element. Density and specific energy can be obtained directly from the cell flow state vector. The velocity magnitude is just the magnitude of the velocity vector made from the x-velocity and y-velocity magnitudes.

Force

The force calculation from an airfoil element is follows (in this example the slate):

$$\vec{F}'_{\text{slat}} = \int_{\text{slat}} p\vec{n}\, dl, \qquad \text{(Eqn. 9)}$$

This is basically saying the force is the integral sum of pressure over the arclength of the airfoil element. Notice this results a vector, so the force has an x and y component. The total force for the whole airfoil would then be the combination for the slate, main, and flap.

$$\vec{F}'_{\text{airfoil}} = \vec{F}'_{\text{slat}} + \vec{F}'_{\text{main}} + \vec{F}'_{\text{flap}}. \qquad \text{(Eqn. 10)}$$

Coefficients of Lift and Drag

The lift L is the y component of the force and the drag D is the x component. They can be normalized by the freestream values to calculate the coefficients of lift and drag. The pressure coefficient can also be calculated from the pressure obtained in Eqn. 8, which can be generalized for all elements.

$$c_\ell = \frac{L'}{\frac{1}{2}\rho_\infty |\vec{v}_\infty|^2 c}, \qquad c_d = \frac{D'}{\frac{1}{2}\rho_\infty |\vec{v}_\infty|^2 c}, \qquad c_p = \frac{p - p_\infty}{\frac{1}{2}\rho_\infty |\vec{v}_\infty|^2}, \qquad \text{(Eqn. 11)}$$

Streamline
The equation for streamline is follows:

$$\psi_2 = \psi_1 + \int_1^2 \rho \vec{v} \cdot \vec{n}\, dl = \psi_1 + \Delta l\, \hat{F}_\rho, \qquad \text{(Eqn. 11)}$$

Mach
The formula for Mach is:

$$\begin{aligned} M &= |\vec{v}|/c = \sqrt{u^2 + v^2}/c \\ c &= \sqrt{\gamma R T} = \sqrt{\gamma p/\rho} \end{aligned} \qquad \text{(Eqn. 12)}$$

The pressure can be found using Eqn. 8's pressure formula. And u, v, and density can be obtained from the volume element.

Results

The Matlab script main1.m refreshes the convergence plot every 10 iterations. There are four individual residual curves because there are total for four flow states. The convergence criterion is set that when the highest residual is below 10^-7 the iteration ends. The following figures shows the result:

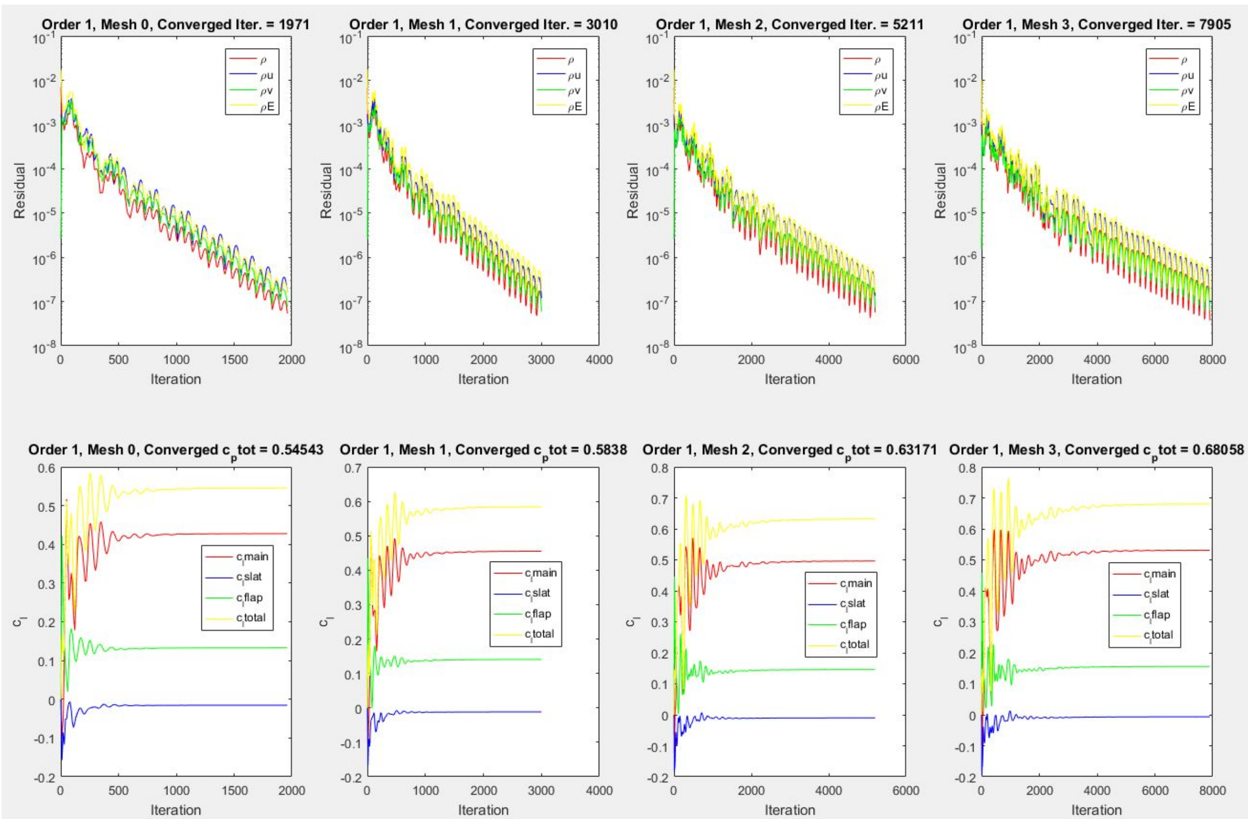


Figure 2. First-order residual plot and coefficient of lift plot over iterations for Mesh 0, 1, 2, 3.

The converged coefficients are:

Mesh:	0	1	2	3
cl_main	0.4277	0.45493	0.49608	0.53097
cl_slate	-0.015353	-0.011767	-0.010129	-0.0065916
cl_flap	0.13309	0.14063	0.14576	0.1562
cl_total	0.54544	0.5838	0.63171	0.68058
cd	0.047439	0.03903	0.02796	0.020143
Mass flow between [slat,main] (kg/(s-m))	0.0020823	0.0026453	0.0033793	0.0039888
Mass flow between [main,flap] (kg/(s-m))	0.0036989	0.0044334	0.0052742	0.0059823

Table 2. First-order converged coefficients and mass flow rate between airfoil elements.

As per Eqn. 12 the sectional mass flow rate is the difference between the streamline values. Thus, the mass flow between the gaps of two airfoil element was find by finding the stream function value difference between the streamline values of each element piece. Note that there were small difference between the streamline values on the surface of each element piece.

The coefficient of pressure value was also calculated at the midpoint of each edge on the airfoil. The value used to calculate the coefficient of pressure is the cell average of the volume element that contains that inviscid wall boundary edge. The cp plot is cut off at max cp of 4 because there are few points where the cp is too large and will flatten the entire cp plot if plotted. The recorded max for each of the three elements are record in the subplot titles.

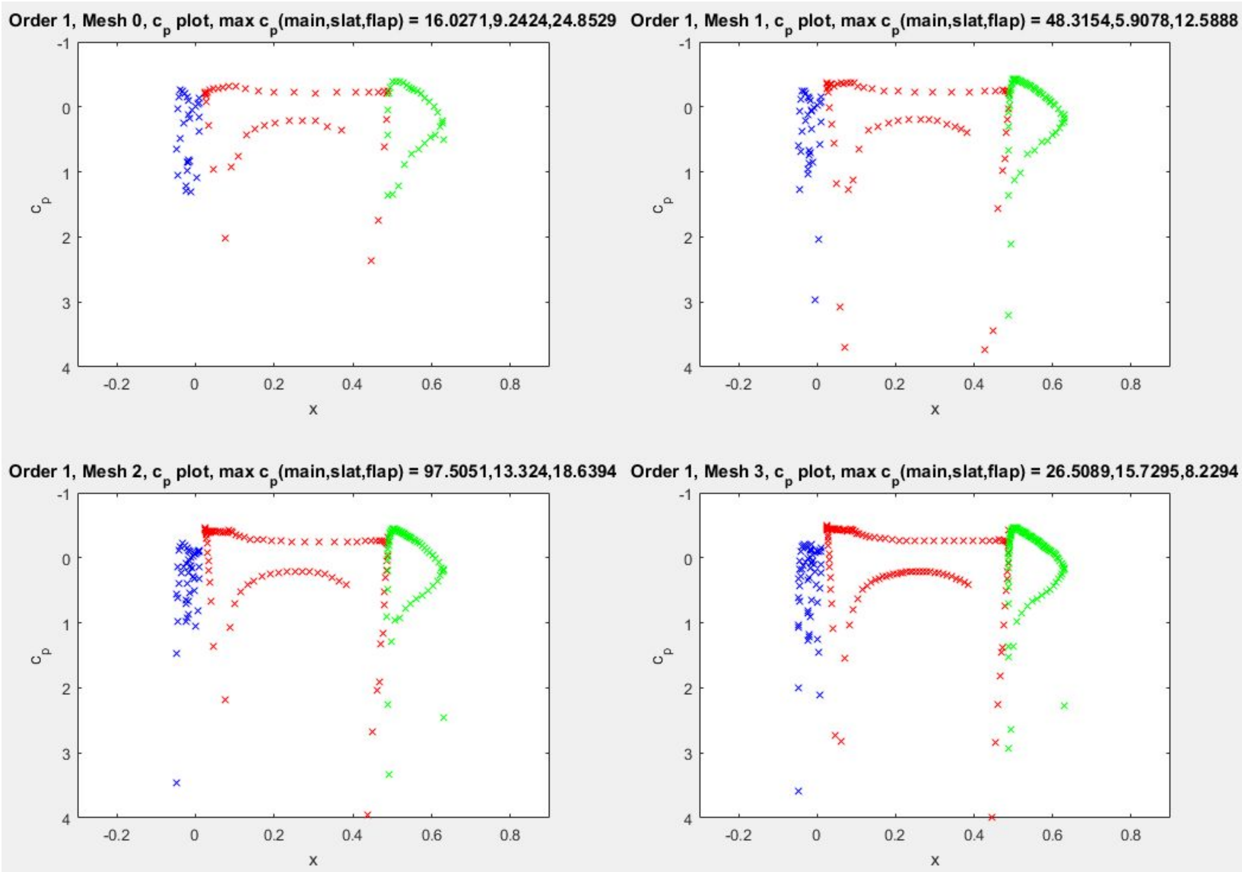


Figure 3. First-order residual plot and coefficient of lift plot over iterations for Mesh 0, 1, 2, 3. Blue is slat, red is main, and green is flap.

Figure 4 and 5 show the cell average contours for first order pressure and Mach. Cell average is plotted because in first order we assume the cell properties are constant. Note the pressure is higher at the top of the surface and Mach is also higher at the top. This is how airfoil give lift.

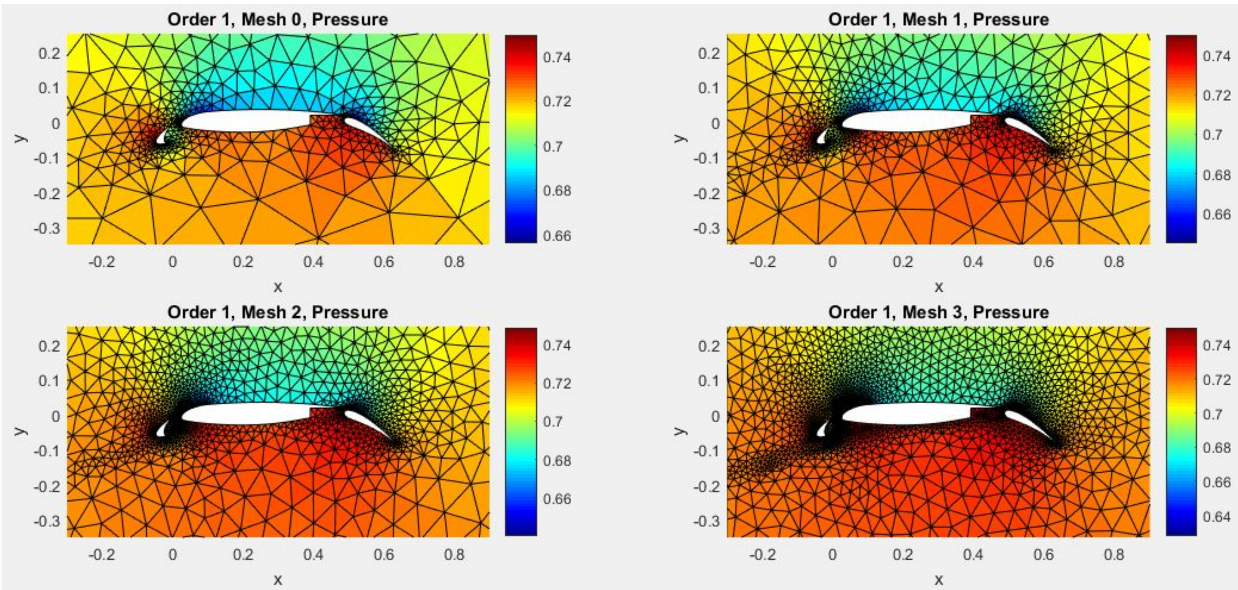


Figure 4. First-order pressure contour for Mesh 0, 1, 2, 3.

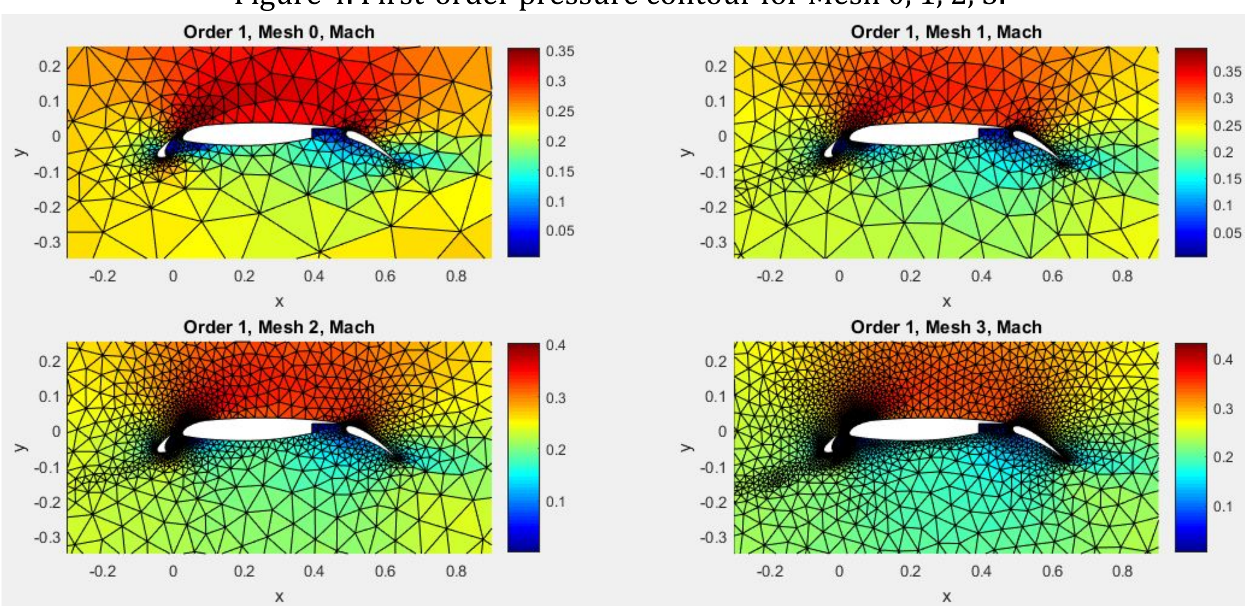


Figure 5. First-order Mach contour for Mesh 0, 1, 2, 3. .

The streamline values are found as follows: It is assume the main airfoil piece to have 0 streamline value. Then using Eqn. 12, if one of the know of an edge nodes have known value and the other unknown, the other can be found using Eqn. 12. All the edges from inner and boundary are looped through multiple times until all node streamline values are found. The streamlines with 50 levels can be seen below:

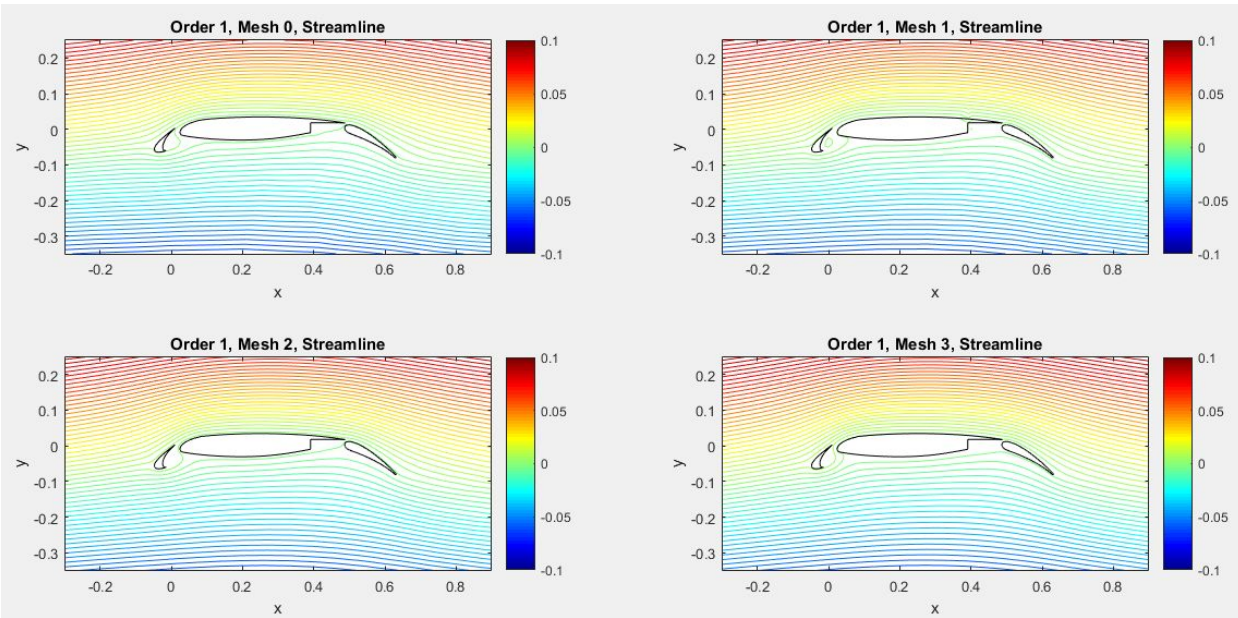


Figure 6. First-order streamline contour for Mesh 0, 1, 2, 3. .

PART 3: SECOND-ORDER FVM

Flow State

The force, coefficients, and streamline equations for the Second-Order FVM are the same in physical sense. But the implementation is different. The First-Order assume the edge state used to solve for the flux is the same as cell average. But the Second-Order discretization assumes the variation of value inside a cell is linear. Thus a modified set of equations are needed.

Since it is assume the value inside a cell is linearly varied, an average gradient that defines the linear variation is needed. The equation is follows:

$$\nabla \mathbf{u}\big|_i = \frac{1}{A_i}\int_{\partial A_i} \hat{\mathbf{u}}\vec{n}\,dl \tag{Eqn. 13}$$

In the equation û is the edge average state. It is assume to be the average between the cell average of the left and right volume element.

Then the gradient is used to find the flow state at midpoint of the edge. The midpoint value is calculated because since the cell is linearly varied the midpoint would provide the edge average. Then edge average from seperatedly the left and right cell will be inputed into flux.m to calculate the second-order flux.

The equation for finding the midpoint value is follows, x is the vector position of the midpoint, and xi is the vector position of the cell centroid (which calculated using polyarea() in Matlab using the three vortices):

$$\mathbf{u}(\vec{x}) = \mathbf{u}_i + \nabla \mathbf{u}\big|_i \cdot (\vec{x} - \vec{x}_i), \tag{Eqn. 14}$$

A second-order time stepping is needed. The one chosen was TVD RK2. It has the following equations:

$$\begin{aligned} \mathbf{u}_i^{FE} &= \mathbf{u}_i^n - \frac{\Delta t_i^n}{A_i}\mathbf{R}_i(\mathbf{U}^n) \\ \mathbf{u}_i^{n+1} &= \frac{1}{2}\left[\mathbf{u}_i^n + \mathbf{u}_i^{FE} - \frac{\Delta t_i^n}{A_i}\mathbf{R}_i(\mathbf{U}^{FE})\right]. \end{aligned} \tag{Eqn. 15}$$

For the inviscid wall boundary condition, the boundary edge's flow state assumed to be the cell centroid state. And this is used to contribute to the state gradient within the cell. To obtain the surface pressure for the force, the boundary surface pressure was also used: Meaning, it is after using the state gradient to find the wall flow state, then wall flow state is used to calculate pressure. The midpoint was used as edge average, once again, because the cell state is linearly variated.

Results

The second order residual convergence plots and coefficient of lift for the three different element plots are in Figure 7. Overall the convergence iteration for second order is much higher than the first order. And the coefficient of lift increased.

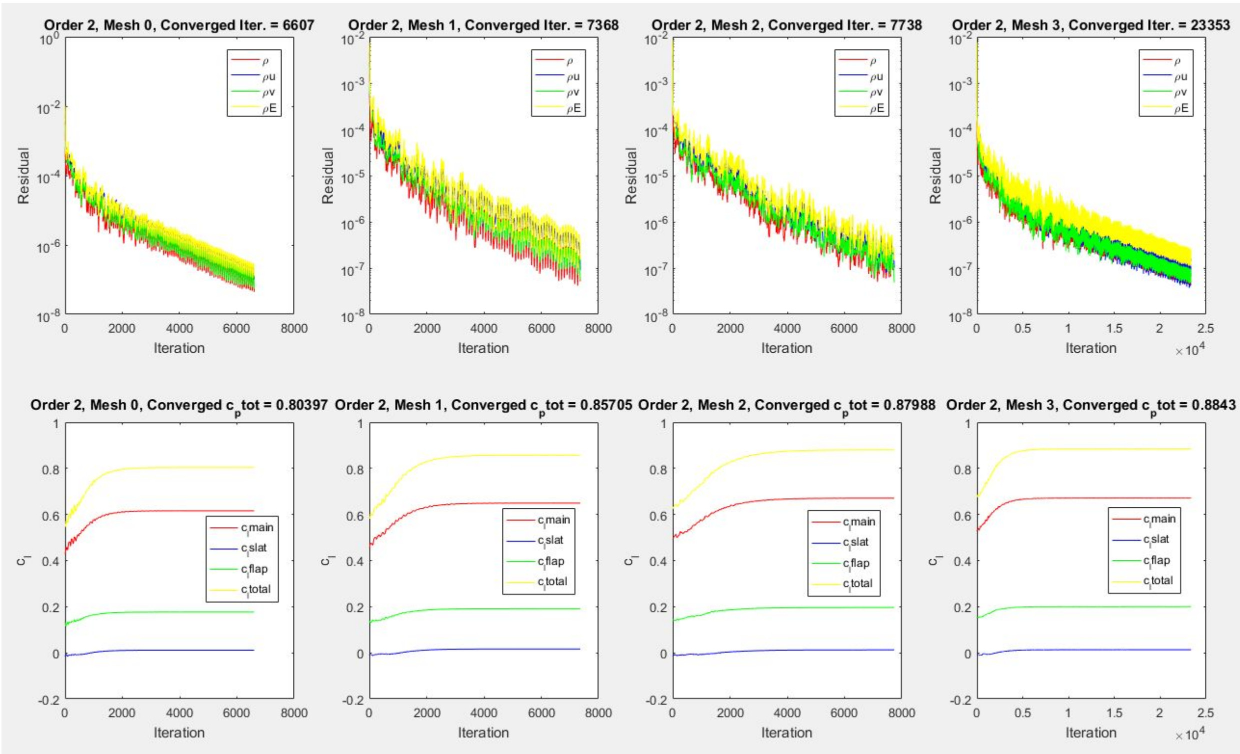


Figure 7. Second-order residual plot and coefficient of lift plot over iterations for Mesh 0, 1, 2, 3.

The converged coefficients are:

Mesh:	0	1	2	3
cl_main	0.61598	0.64936	0.67089	0.67113
cl_slate	0.01129	0.016432	0.012485	0.013077
cl_flap	0.17671	0.19125	0.1965	0.20009
cl_total	0.80397	0.85705	0.87988	0.8843
cd	-0.0032858	-0.0055701	0.005295	0.00058391
Mass flow between [slat,main] (kg/(s-m))	0.0052444	0.0056228	0.006013	0.0060639
Mass flow between [main,flap] (kg/(s-m))	0.0074907	0.0080361	0.0084523	0.0085377

Table 3. Second-order converged coefficients and mass flow rate between airfoil elements.

Note the cd in second order is much lower than the cd in first order. This fits the D'Alembert paradox for 2D potential flow analysis. The drag for a 2D potential flow airfoil is always zero.

Note that the mass flow between the airfoil elements increased from first order to second order. Then it stabilizes between mesh 2 and 3, as in the increase in value between meshes is slowing down. This effect is also similar in the cl values. This suggests the second order is need for an accurate solution (compared to exact solution of the Euler's equations).

The coefficient of pressure is shown in Figure 8. Once again, the cp maximum is capped at cp = 4 in the plots because otherwise the cp plot will be flattened by the high cp caused by sharp edges. The change from the first order cp plots includes a lower upper surface pressure.

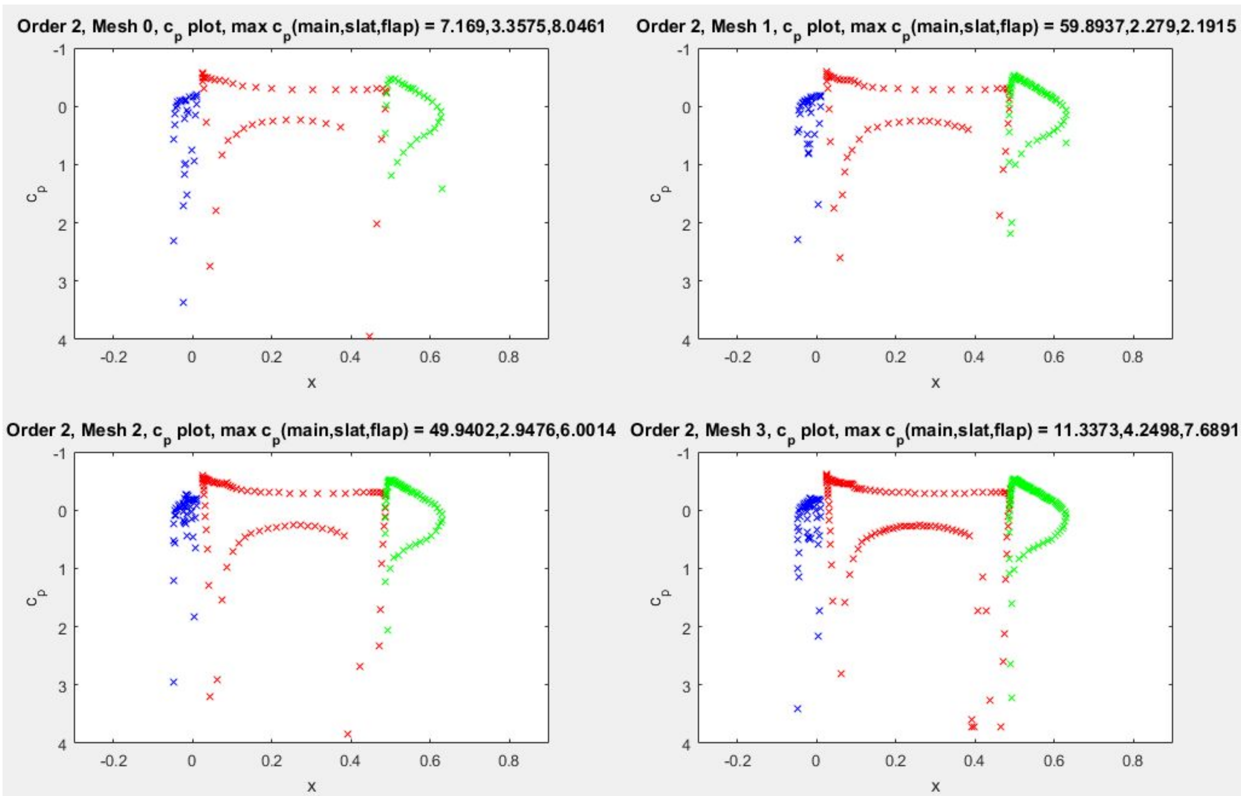


Figure 8. Second-order residual plot and coefficient of lift plot over iterations for Mesh 0, 1, 2, 3. Blue is slat, red is main, and green is flap.

Figure 9 and 10 show the second order pressure and Mach contour. Matlab pdeplot's "Interp" XYStyle was used be in a second order cell, the state variates. The difference from the first order contours are the pressure and Mach range is much higher. The top surface has lower cp and bottom surface higher cp than their first order counterparts. Thus high coefficient of lift is recorded.

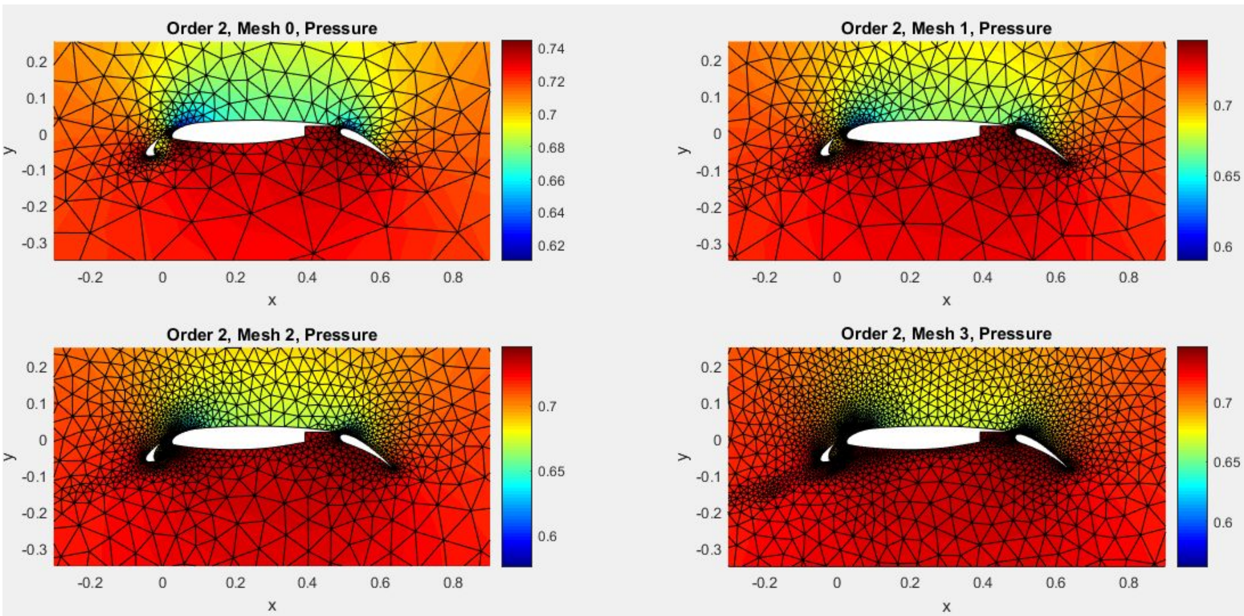


Figure 9. Second-order pressure contour for Mesh 0, 1, 2, 3.

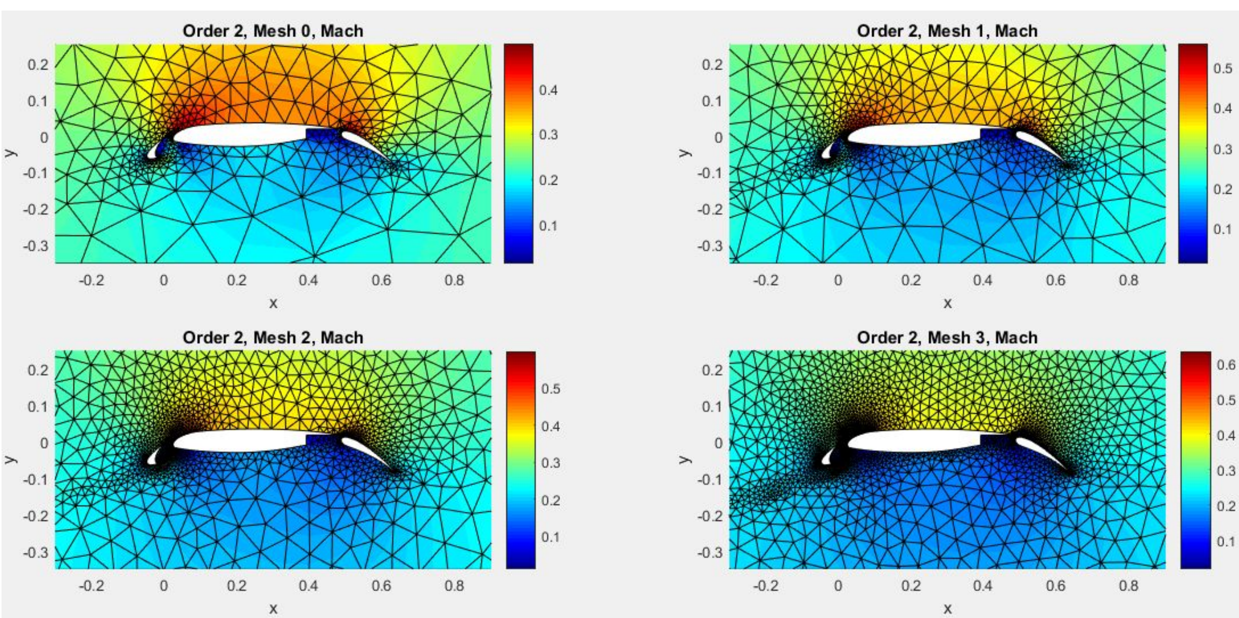


Figure 10. Second-order Mach contour for Mesh 0, 1, 2, 3. .

The streamlines with 50 levels can be seen below:

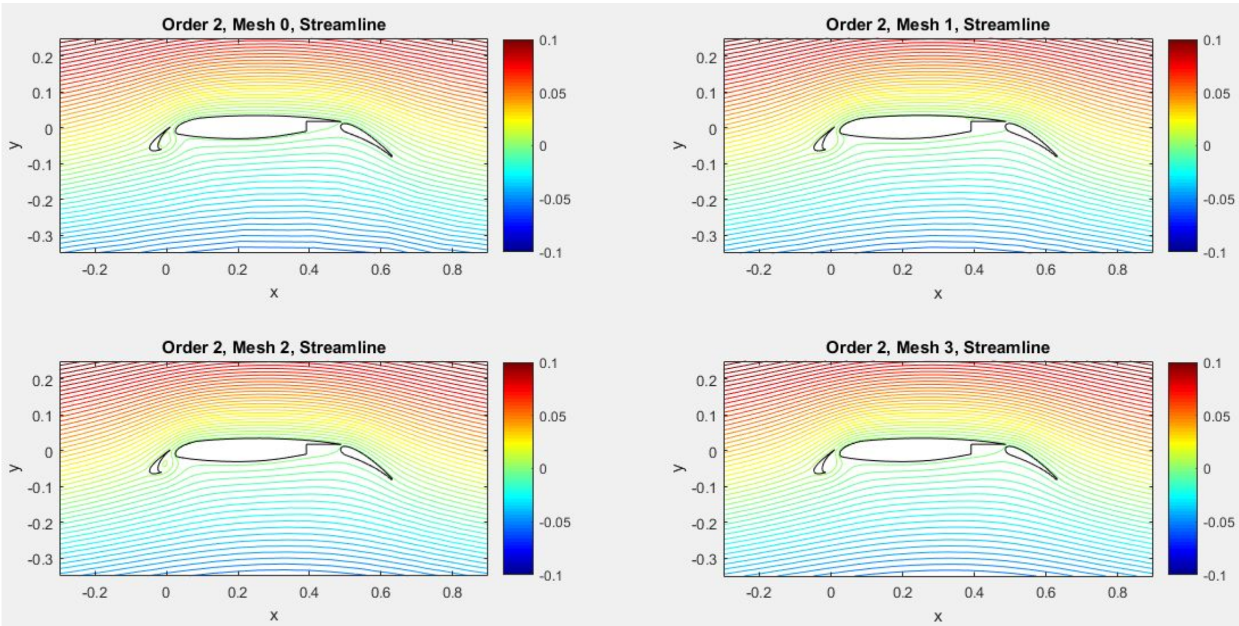


Figure 11. Second-order streamline contour for Mesh 0, 1, 2, 3.

CONCLUSION

First and second order finite volume methods were implemented on a three element high lift airfoil. Element-wise coefficients of lift was obtained. The other post-processings include: cp plot, pressure contour, Mach contour, streamline contour, and mash flow rate between gaps of the airfoil elements. It is found that the coefficient of lift values increased from first order solution to second order solution. It is with the second order solution that the coefficient of lift seems to convergence with element amount increase. It is found that the pressure is lower at the upper surface and higher on the bottom. Corresponding, the Mach is higher at the top and lower on the bottom. There is also mass flow between the gaps of the airfoil elements, which also increased between first and second order FVM. The streamline, however, looks the similar for all meshes and orders.