Lecture 13 Notes:

Goals for today- Lecture 13

1. Cavitation in pumps
2. Pumps and their characteristics Net Positive Suction head
3. Pump Non-dimensional parameters, i.e. coefficients of capacity, head, and power
4. Series and parallel pumps

Net Positive Suction Head (NPSH):

In a pump curve, NPSH defines under what condition, the pump will cavitate i.e. $P_{inlet} < P_v$ where P_v→ saturated pressure for liquid at given temp.

Cavitation Video:
Cavitation -Easily explained! https://www.youtube.com/watch?v=U-uUYCFDTrc

If$P_v > P_{inlet}$→ fluid will boil→ flow will become bubbly (cavitation)→ pump vibration, extra noise, pitting of vane/blades since bubble will burst on the travel to the high press. Side (i.e. exit of pump)

*cavitation must be prevented always

Condirion to avoid cavitation (1)

$$NPSH \leq \frac{p_i}{\rho g} + \frac{v_i^2}{2g} - \frac{p_v}{\rho g}$$

NPSH-> read from pump performance curves

$\frac{p_i}{\rho g} + \frac{v_i^2}{2g}$→ total flow head

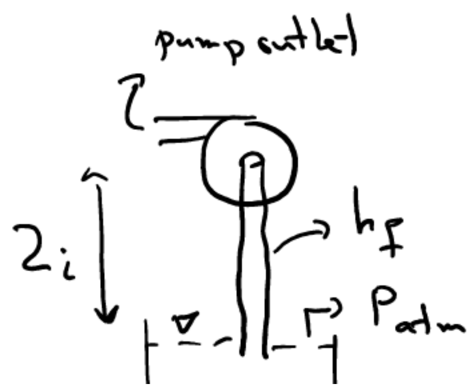
$\frac{p_i}{\rho g} + \frac{v_i^2}{2g} - \frac{p_v}{\rho g}$→ available NPSH

Using Bernoulli for a pump intake place above a reservoir

$NPSH \leq \frac{P_a}{\rho g} - Z_i - h_f - \frac{P_v}{\rho g}$ (2)

h_f→ both minor and major losses

Always design for hot condition!!



Sec 11.2: Basis Pump Formulas:

$$\left.\begin{matrix} p_\omega = \rho g Q H \\ p_p = \omega T \end{matrix}\right\} \eta = \frac{p_\omega}{p_p}$$

Power delivered to the fluid→ $p_\omega = \rho g Q H$

Power delivered to the pump (BHP)→ $p_p = \omega T$

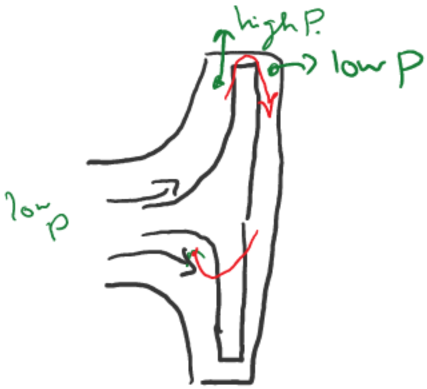
Pump designer wants to maximize this→ η

(3) $\eta = \eta_v \times \eta_h \times \eta_m$

η_v→ volumetric, Q that is last due to clearance between casing and vane/impeller/blades

η_h→ hydraulic, frictional loss over blades/vanes, outlet …

η_m→ mechanical efficiency (bearing, etc.)



Recall discussion about cut impeller/blade

See Fig. 11.7

$$\eta_v = \frac{Q}{Q + Q_L}$$

Q→ flow rate at outlet

$Q + Q_L$→ recirculated flow in the pump due to clearances

$$\eta_h = 1 - \frac{h_f}{h_s}$$

h_f→ frictional losses inside the pump

h_g→ total static head of the pump

$$\eta_m = 1 - \frac{P_f}{bhP}$$

P_f→ frictional losses of power due to mechanical components

bhP→ P_p

Dimensional Pump Parameters:

$gH = f_1(Q, \rho, \eta, \mu, D, \varepsilon)$

ρ→ RPM, ε → roughness

$P_p = f_2(Q, \rho, \eta, \mu, D, \varepsilon)$

Eliminate the dimensions

$\frac{gH}{\eta^2 D^2} = f\left(\frac{Q}{\eta D^3}, \frac{\rho\, \eta D^2}{\mu}, \frac{\varepsilon}{D}\right)$ (4)

$\frac{gH}{\eta^2 D^2}$→ C_H Coefficient of head

$\frac{bhP}{\rho \eta^3 D^5} = f\left(\frac{Q}{\eta D^3}, \frac{\rho\, \eta D^2}{\mu}, \frac{\varepsilon}{D}\right)$ (5)

$\frac{bhP}{\rho \eta^3 D^5}$→ C_p coefficient of power

$\frac{Q}{\eta D^3}$→ C_Q coefficient of capacity

In practice, due to high turbulence, $\frac{\varepsilon}{D}$ is show to have little effect, and experiment show that for many pumps, viscosity (for a given fluid) has the same percentage effect, i.e. $\frac{\rho\eta D^2}{\mu}$ term can be discounted for similarity analysis see fig.11.8
$C_H \approx C_H(C_Q) \quad \& \, C_p \approx C_\rho(C_Q)$ (6)
[For geometrically similar pumps using the <u>same</u> liquid]

Also, $\eta \equiv \frac{C_H C_Q}{Cp}$→ so , η is a finction of C_Q as well
Suctioned→ $CHS = g\left(\frac{NPSH}{\eta^{2\cdot} D^2}\right)$

Considering above similarity rules($\pi_{1m} = \pi_{1p}$) for pump will be: (7)

$$\frac{Q_2}{Q_1} = \frac{\eta_2}{\eta_1}\left(\frac{D_2}{D_1}\right)^3$$
$$\frac{H_2}{H_1} = \left(\frac{\eta_2}{\eta_1}\right)^2\left(\frac{D_2}{D_1}\right)^2$$
$$\frac{P_2}{P_1} = \frac{\rho_2}{\rho_1}\left(\frac{\eta_2}{\eta_1}\right)^3\left(\frac{D^2}{D_1}\right)^5$$

Eq(7) can be used to estimate the effect of changing parameters such as $\eta, D, \rho\ etc.$ on Q, P & H

Note: Although due to similarity principals, one expects $\eta_1 = \eta_2$, but in practice (effect of $\varepsilon, \mu, \ldots$) on has (8):

$$\frac{1-\eta_2}{1-\eta_1} \approx \left(\frac{D_1}{D_2}\right)^{\frac{1}{4}}$$
$$\frac{0.94-\eta_1}{0.94-\eta_2} \approx \left(\frac{Q_1}{Q_2}\right)^{0.32}$$

Larger purmps or higher Re (Q) will improve slightly the η

Note: Viscosity changes the pump p[performance drastically (see fig.11.10); There is no general eq. for, but needs testing.

Parallel & Series Pumps:

Parallel:
$Q_p = Q_1 + Q_2 + \cdots$
$H_p = H_1 = H_2 = \cdots$

- Each pump in a parallel arrangement should meet the required head individually
- Little limitation on number of pumps to be in parallel

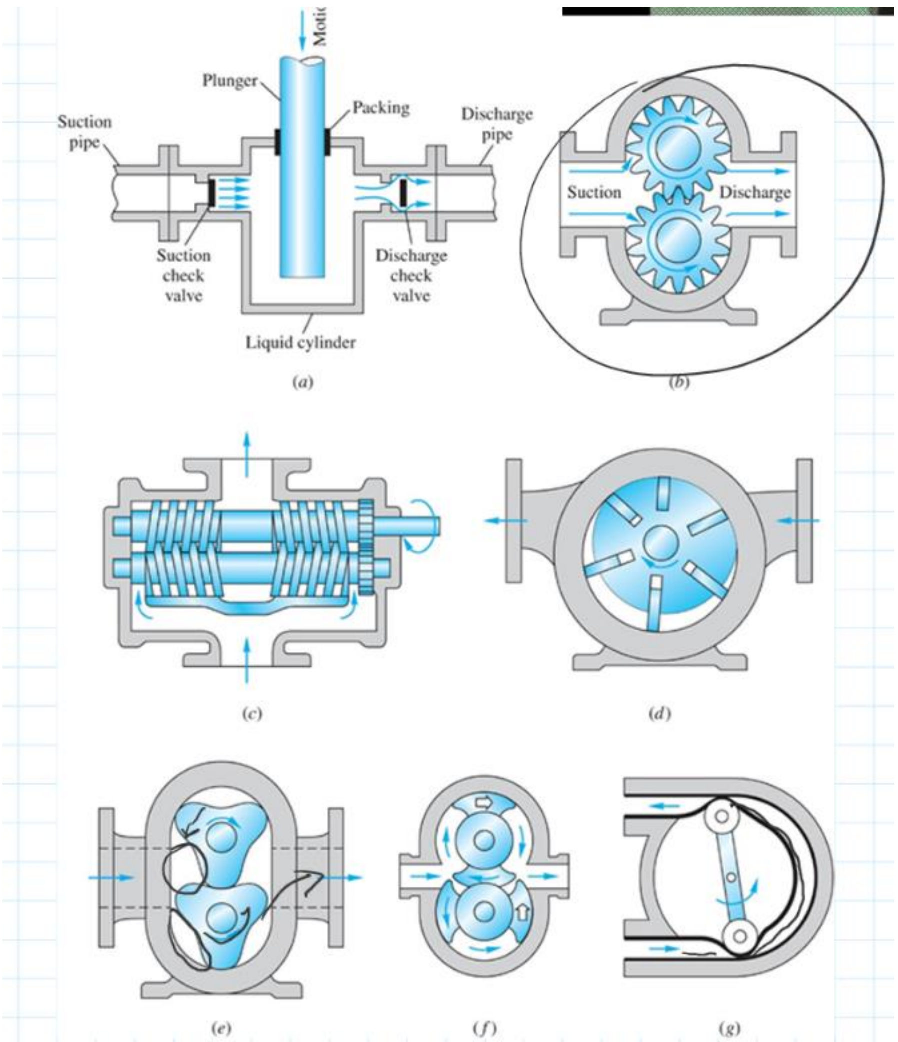
Series:
Parallel:
$Q_s = Q_1 = Q_2 = \cdots$
$H_s = H_1 + H_2 + \cdots$

Cannot have too many pumps in series due to pressure limitation for pump casing

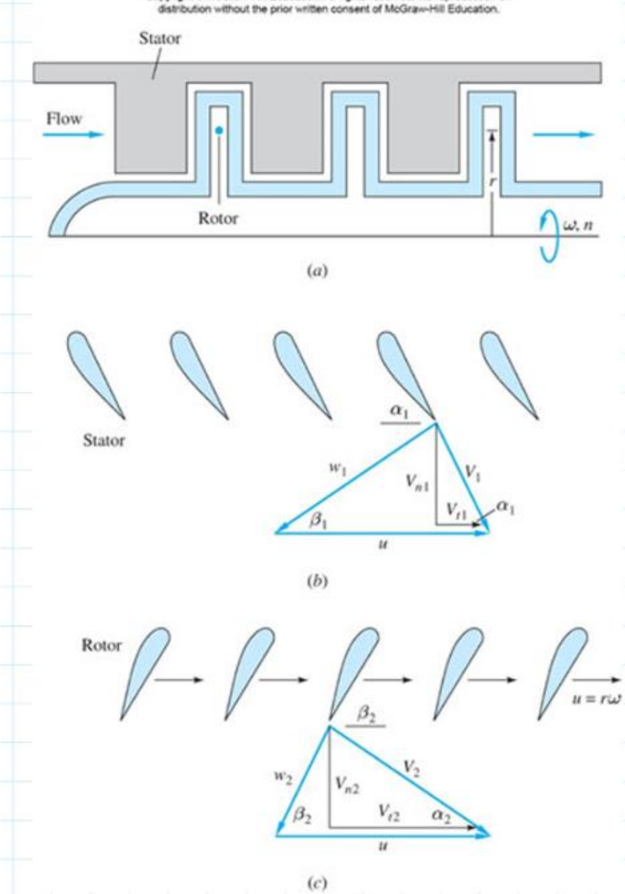
See Figs, 11.9

Fig. Chap 11
Basic principles and history of industrial pumps
https://www.youtube.com/watch?v=eWachJNuxSU&ab_channel=JAESCompany

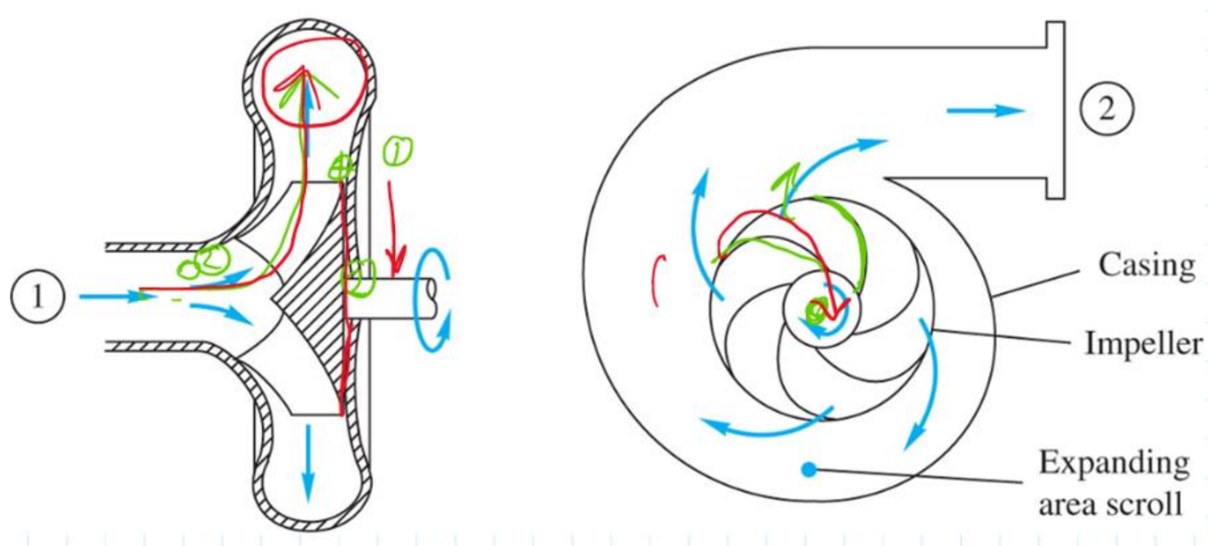
Breast Pump https://www.youtube.com/watch?v=2U0s5D8maNk&ab_channel=MyAmeda



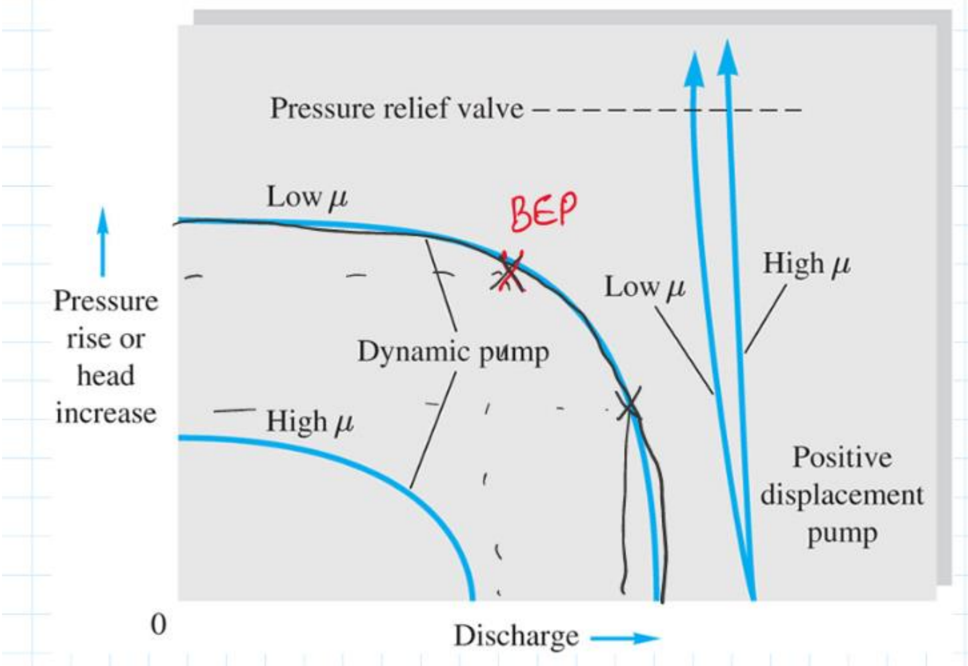
Copyright © McGraw-Hill Education. All rights reserved. No reproduction or distribution without the prior written consent of McGraw-Hill Education.

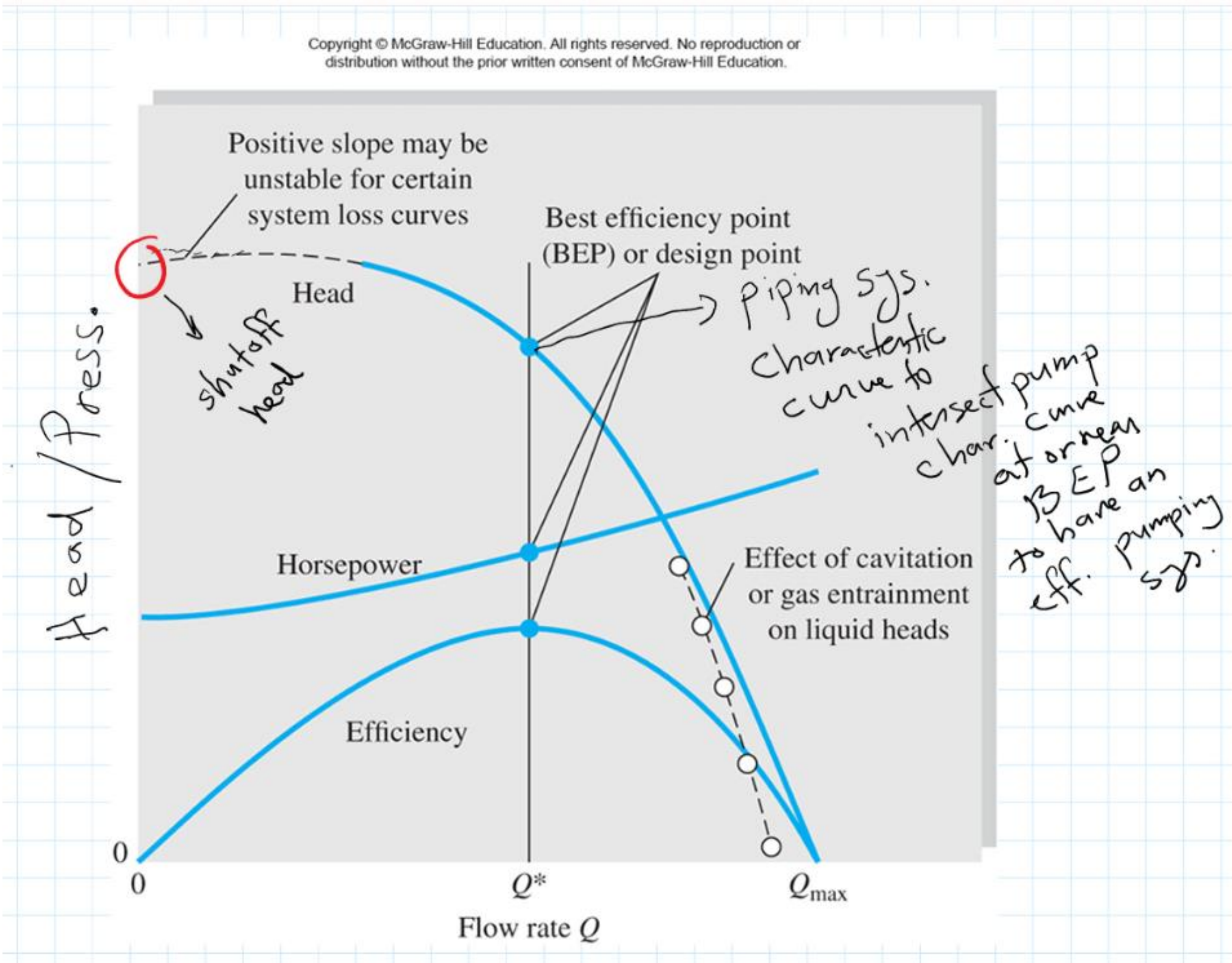


Copyright © McGraw-Hill Education. All rights reserved. No reproduction or distribution without the prior written consent of McGraw-Hill Education.



Copyright © McGraw-Hill Education. All rights reserved. No reproduction or distribution without the prior written consent of McGraw-Hill Education.





- Piping system characteristic curve to intersect pump char. curve at or near BEP to have an efficient pumping system

Copyright © McGraw-Hill Education. All rights reserved. No reproduction or distribution without the prior written consent of McGraw-Hill Education.

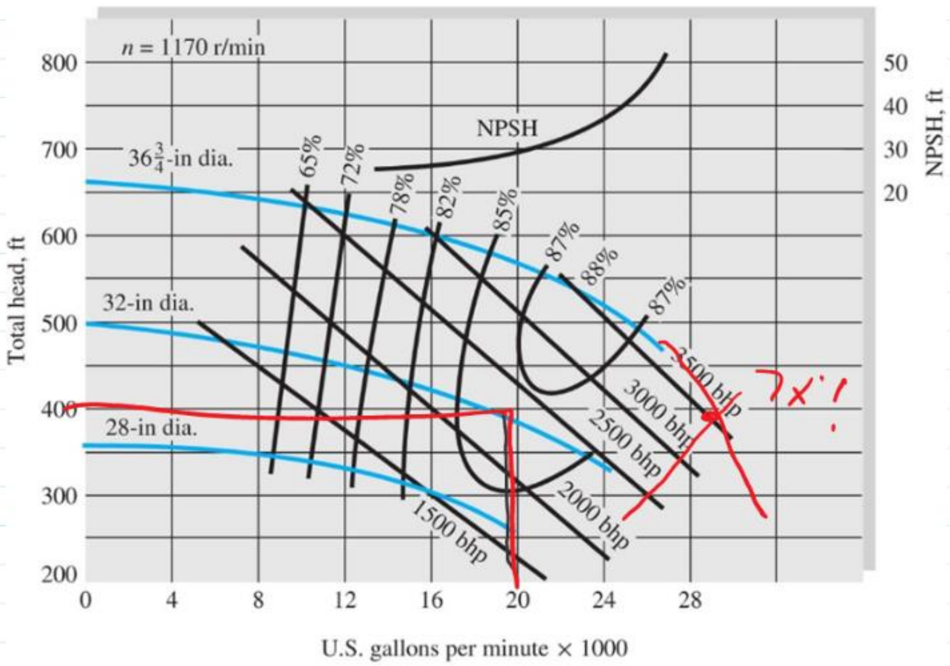
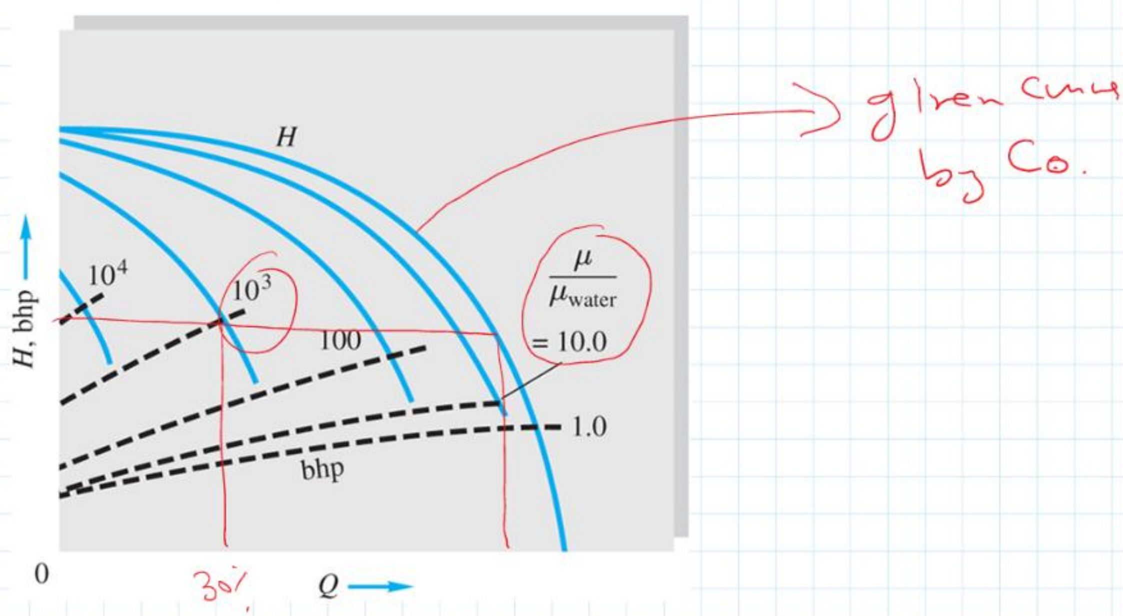
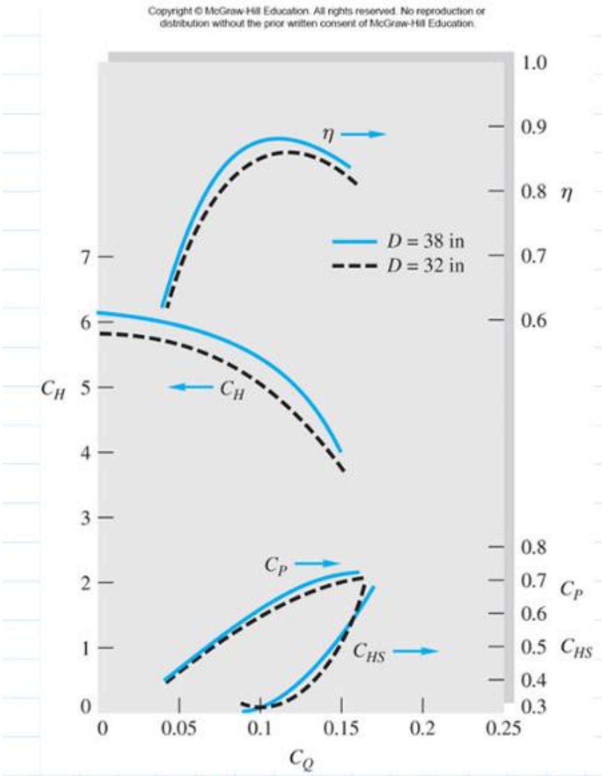


Fig 11.10

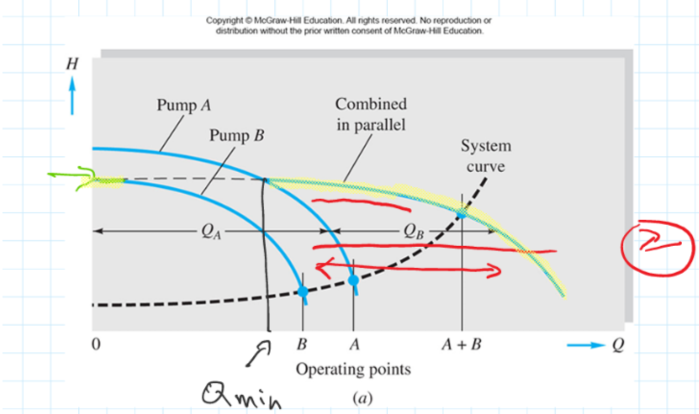
Copyright © McGraw-Hill Education. All rights reserved. No reproduction or distribution without the prior written consent of McGraw-Hill Education.



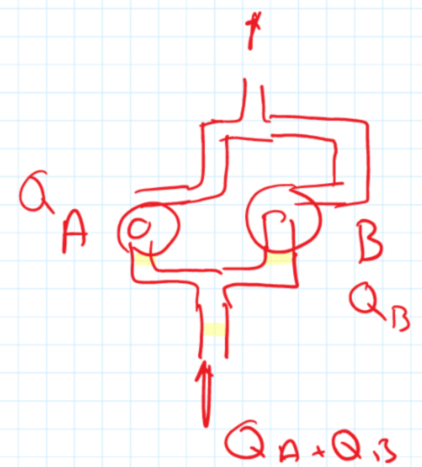
→ Given curve by C_Q
Fig 11.8



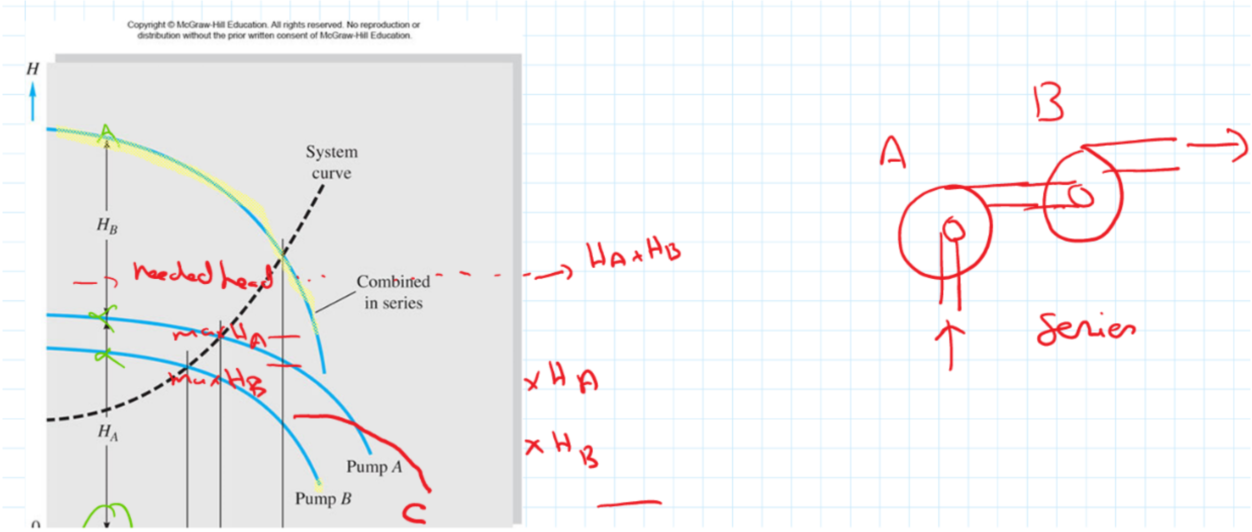
Copyright © McGraw-Hill Education. All rights reserved. No reproduction or distribution without the prior written consent of McGraw-Hill Education.
η
D = 38 in
D = 32 in
C_H
C_P
C_{HS}
C_Q
1.0
0.9
0.8 η
0.7
0.6
0.8
0.7 C_P
0.6
0.5 C_{HS}
0.4
0.3
7
6
C_H 5
4
3
2
1
0
0
0.05
0.1
0.15
0.2
0.25



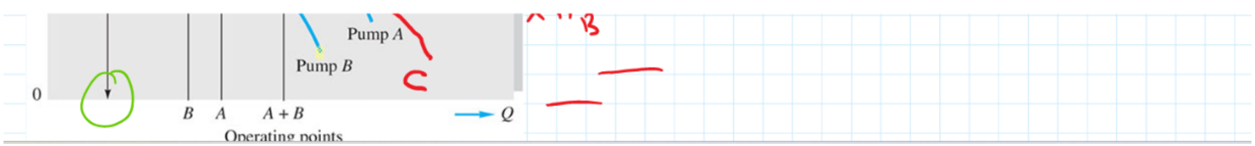
Copyright © McGraw-Hill Education. All rights reserved. No reproduction or distribution without the prior written consent of McGraw-Hill Education.
H
Pump A
Pump B
Combined in parallel
System curve
Q_A
Q_B
0
B
A
A + B
Q
Operating points
(a)
Qmin



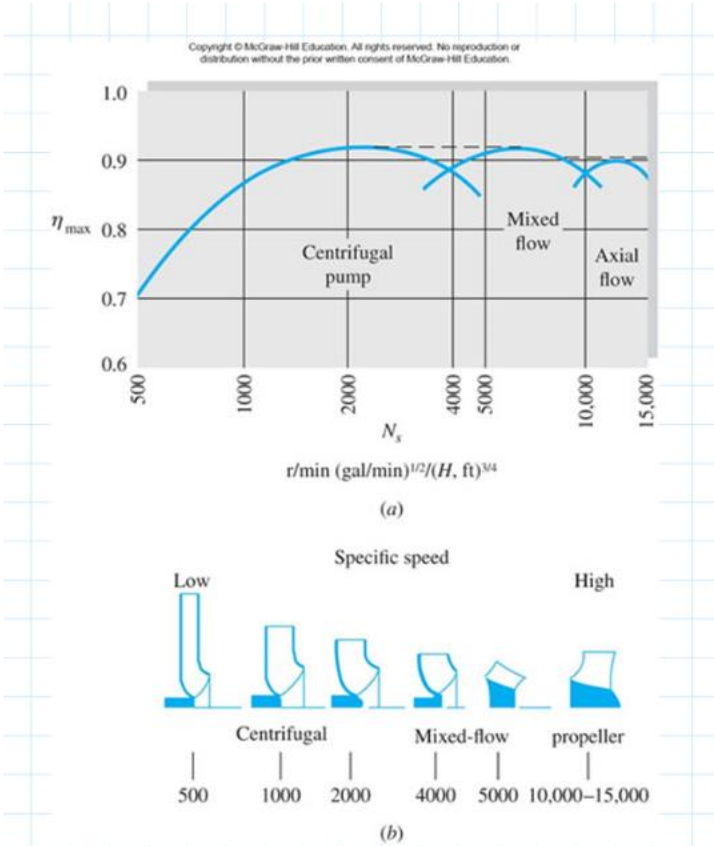
Q_A
A
B
Q_B
$Q_A + Q_B$

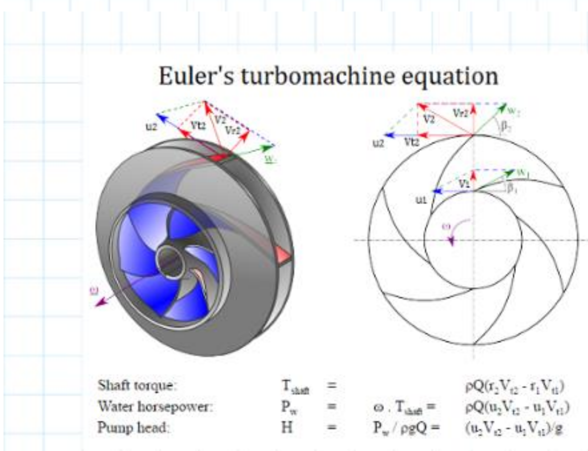


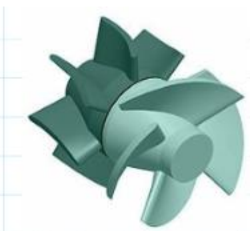
Copyright © McGraw-Hill Education. All rights reserved. No reproduction or distribution without the prior written consent of McGraw-Hill Education.
H
System curve
H_B
needed head
$H_A + H_B$
Combined in series
max H_A
max H_B
x H A
x H B
H_A
Pump A
Pump B
C
0
A
B
Series



Pump A
Pump B
C
0
B
A
A + B
Q
Operating points


Copyright © McGraw-Hill Education. All rights reserved. No reproduction or distribution without the prior written consent of McGraw-Hill Education.
1.0
0.9
η_{max} 0.8
0.7
0.6
Centrifugal pump
Mixed flow
Axial flow
500
1000
2000
4000
5000
10,000
15,000
N_s
r/min (gal/min)$^{1/2}$/(H, ft)$^{3/4}$
(a)
Specific speed
Low
High
Centrifugal
Mixed-flow
propeller
500
1000
2000
4000
5000
10,000–15,000
(b)

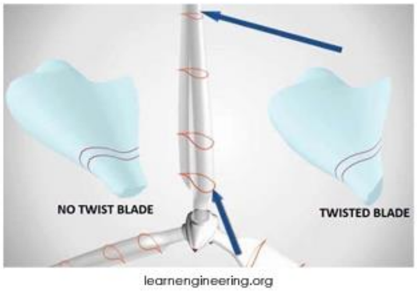

Euler's turbomachine equation
u2
Vt2
V2
Vr2
u1
V1
Shaft torque: T_{shaft} = $\rho Q(r_2 V_{t2} - r_1 V_{t1})$
Water horsepower: P_w = $\omega \cdot T_{shaft}$ = $\rho Q(u_2 V_{t2} - u_1 V_{t1})$
Pump head: H = $P_w / \rho g Q$ = $(u_2 V_{t2} - u_1 V_{t1})/g$

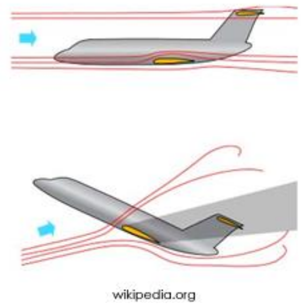


Introduction to Computational Fluid Dynamics (CFD Analysis):

Airfoil:

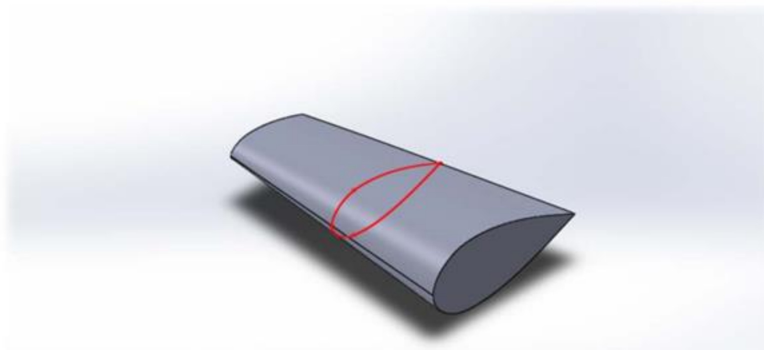
Airfoils are used various for various applications, from turbines blades to airplane wings





Example- simplified airfoil cross section

For example, this section of a wing can be simplified into a 2 dimensional airfoil for easier analysis.

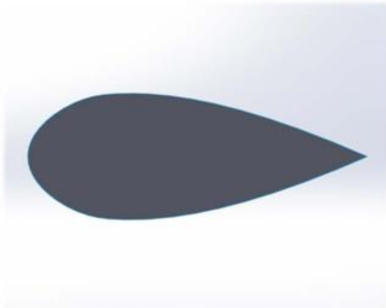


For this example we will use the following values & assumptions:

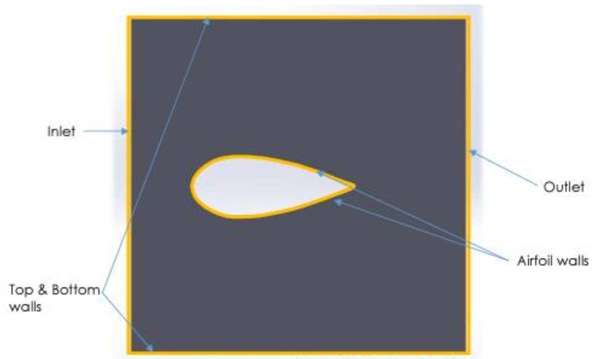
- Fluid is air
- Velocity at inlet is 100 m/s
- No slip condition on airfoil surface Assumptions:
- 2-dimensional
- Steady State
- Incompressible
- Atmospheric pressure at end of controlled volume

Physical vs. CFD:

- Notice in CFD we analyze the surrounding volume of fluid, not the solid the airfoil
- This is a controlled volume analysis,
    - Walls of the CV needs to be defined (yellow lines/curves)
        - the walls must be at a reasonable distance from the airfoil to avoid <u>the flow to be altered by the walls.</u>



Physical model (solid airfoil)

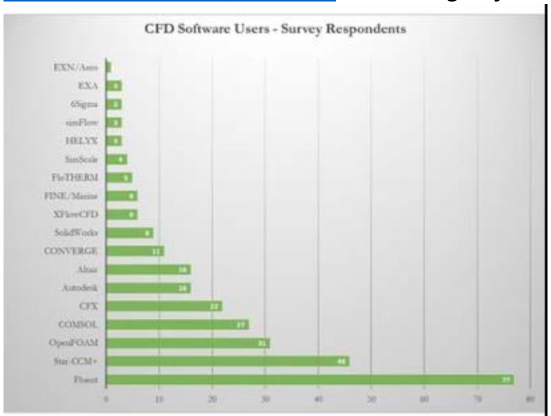


Numerical model (airfoil is hollow)

CFD Analysis:

- There are many commercial and open source CFD codes
    - Fluent (by ANSYS)
    - Open Foam
    - Star-CCM+
- Some are specialized (FINE/Marine), some are more general (Fluent)

www.resolvedanalytics.com *Percentage of Respondent 2016*



www.resolvedanalytics.com Percentage of respondents 2016

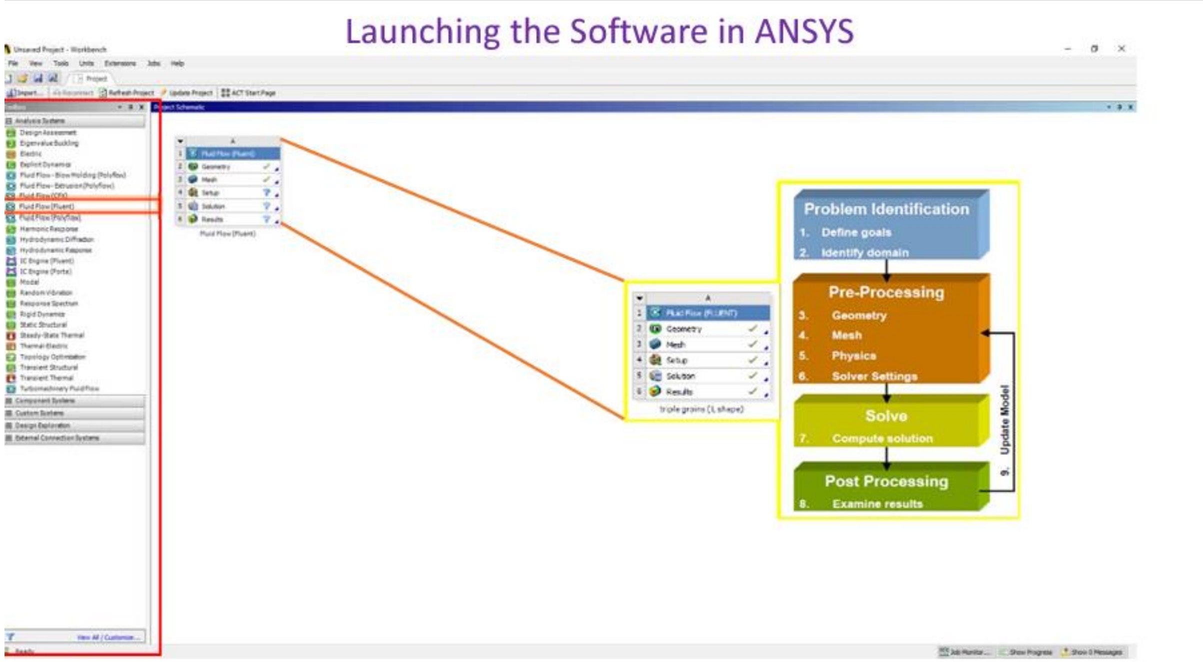
Overall structure of CFD analysis

The general process for performing a CFD analysis according to NASA:

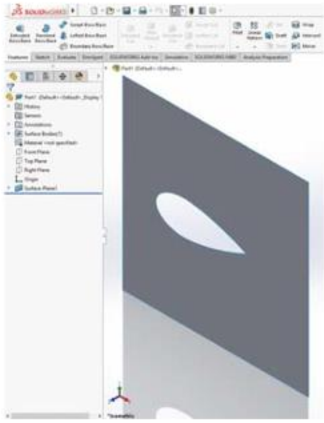
- Formulate the Flow Problem
- Model the Geometry and Flow Domain
- Establish the Boundary and Initial Conditions
- Generate the Grid
- Establish the Simulation Strategy
- Establish the Input Parameters and Files
- Perform the Simulation
- Monitor the Simulation for Completion
- Post-process the Simulation to get the Results
- Make Comparisons of the Results
- Repeat the Process to Examine Sensitivities
- Document

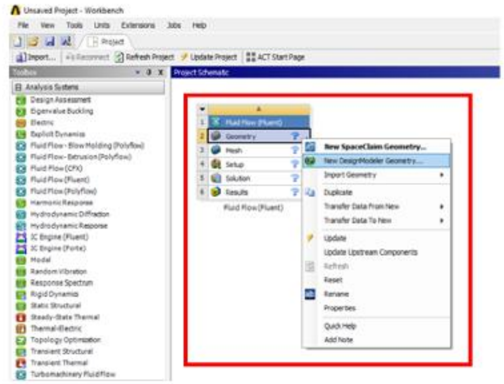
L. Jawad Aziz, M. T. Nasret, Int. J. Sci& EnggRes, 5, 143, 2014

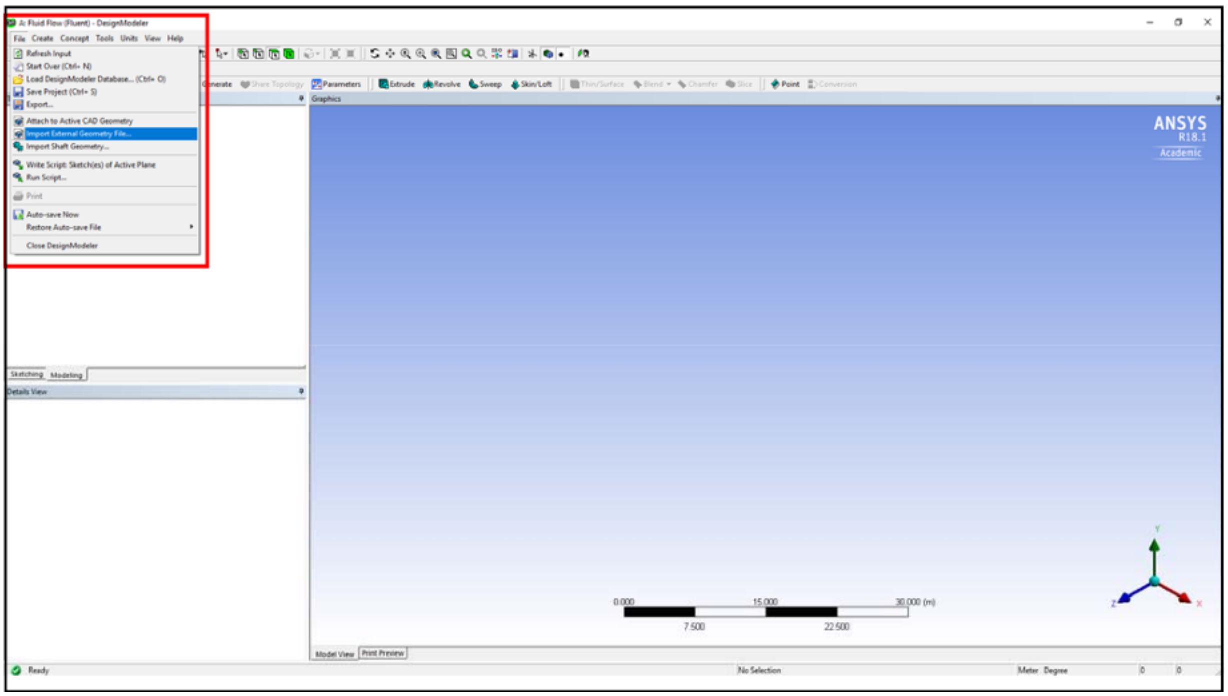
- We will use Fluent in this course
    - To make the geometry you can use Solidworksand then import to Fluent
- It is available in the counsels under ANSYS software package in Petire Building room 020 (the computer labs)



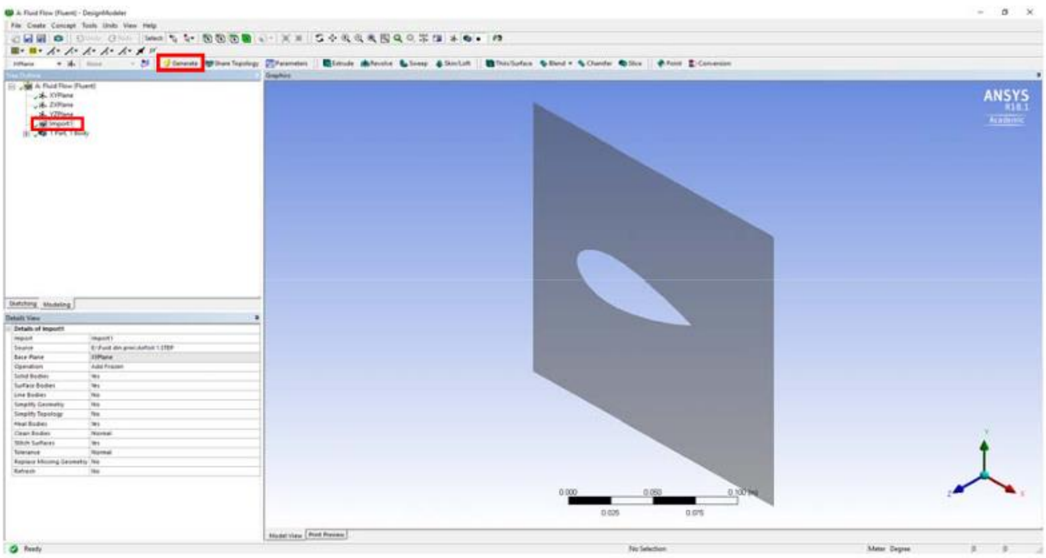
Creating the geometry for analysis The geometry can be created in SolidWorks, but then has to be imported in ANYSYS





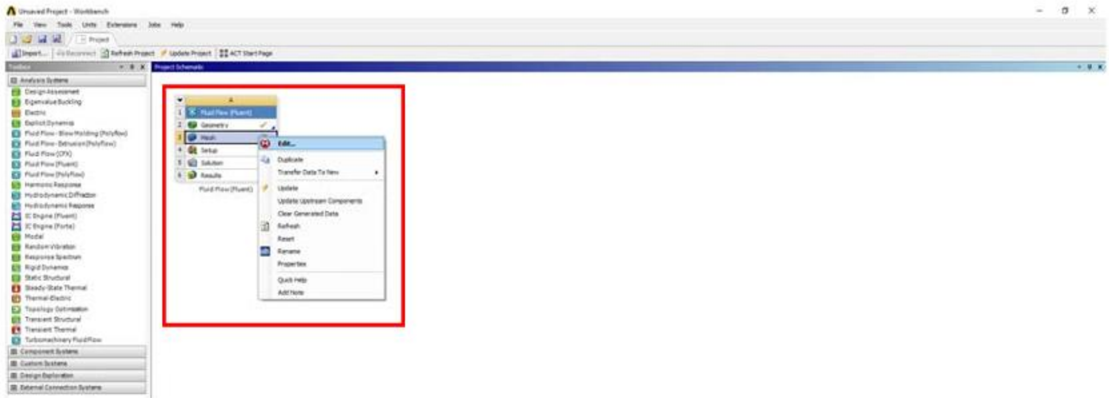


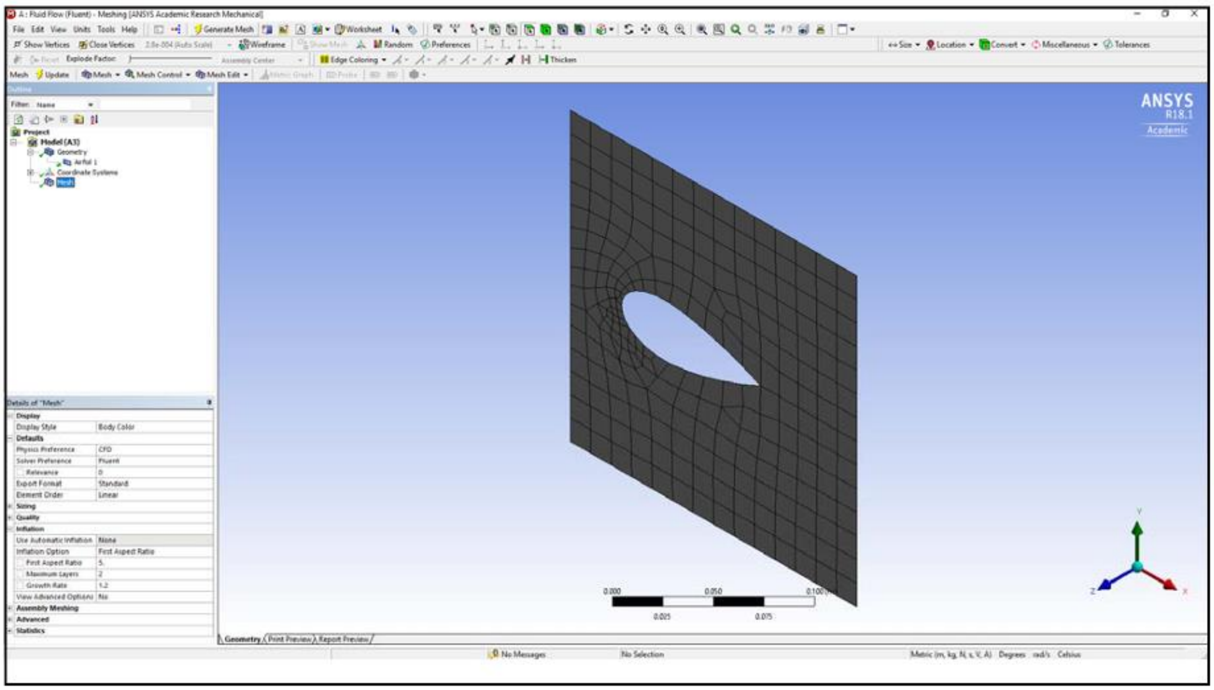
After importing, click “Generate” for the object to appear and finish the importing process



Mesh

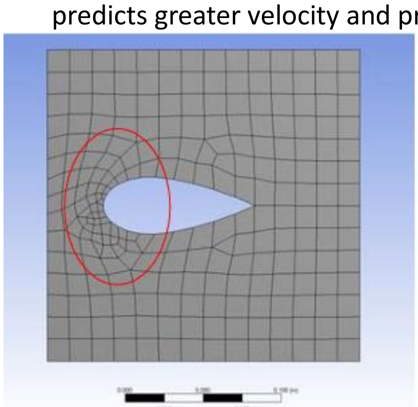
Now that there is a geometry, we can divide it into elements.





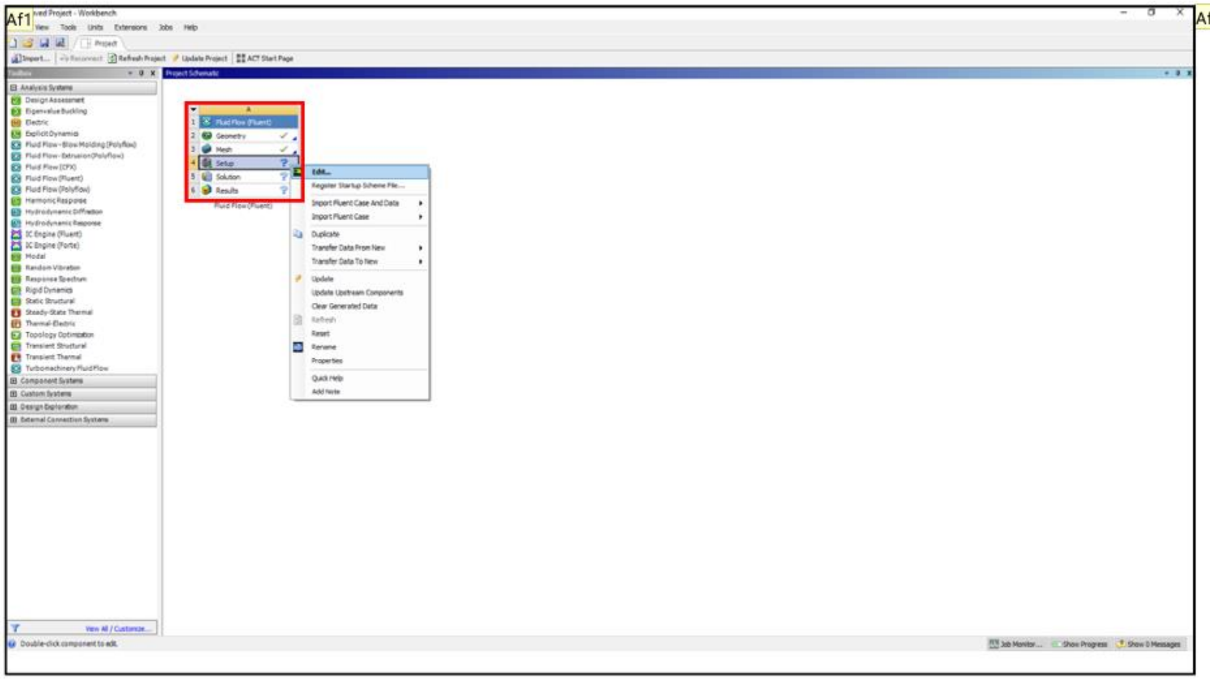
Mesh shape

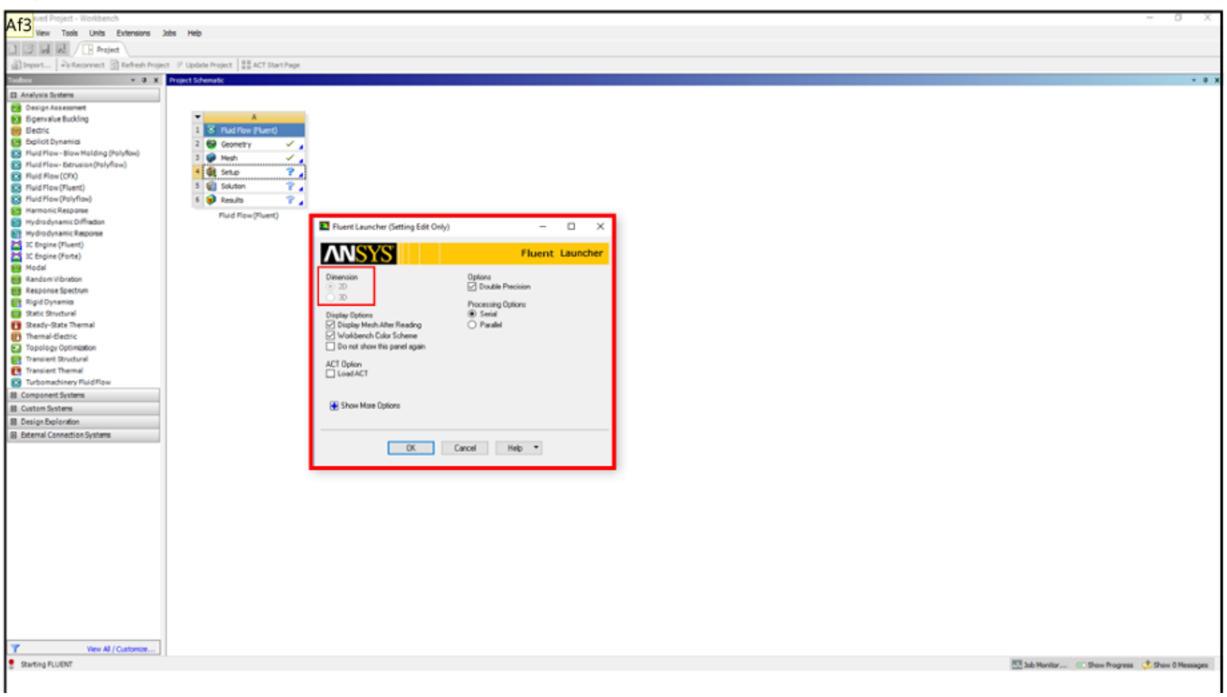
- The mesh was generated using an automatic method, which resulted in 234 elements.
- Note that the elements are smaller in the highlighted section, this is because the program predicts greater velocity and pressure gradients in front of the airfoil.



Fluent

Now that we have defined the geometry and the mesh size, we can continue to edit our case in “Fluent”, where we’ll set all the parameters to solve.





Two Dimensional Flow:

Continuity $\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho n) + \frac{\partial}{\partial y}(\rho v) + \frac{\partial}{\partial z}(\rho w) = 0$

Navier-Stokes

$$\rho g_x - \frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) = \rho \frac{Du}{Dt}$$

$$\rho g_y - \frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) = \rho\frac{Dv}{Dt}$$

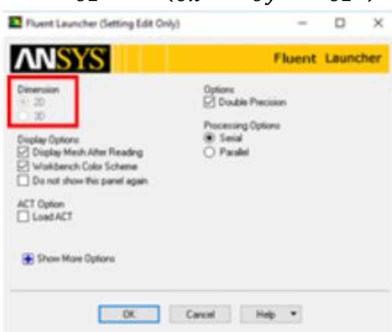
$$\rho g_z - \frac{\partial p}{\partial z} + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) = \rho\frac{Dw}{Dt}$$

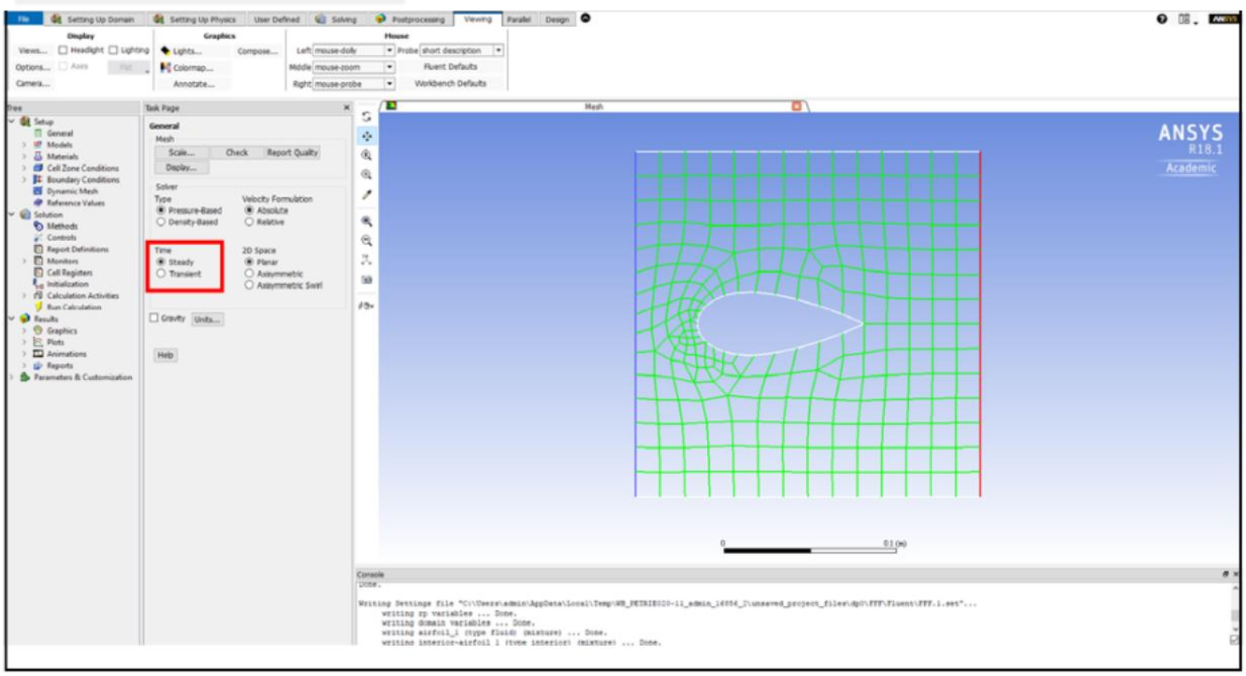
$$\frac{\partial}{\partial z} \rightarrow 0$$

$$\frac{\partial^2 u}{\partial z^2} \rightarrow 0$$

$$\frac{\partial^2 v}{\partial z^2} \rightarrow 0$$

$$\rho g_z - \frac{\partial p}{\partial z} + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) = \rho\frac{Dw}{Dt} \rightarrow 0$$





Define Steady State

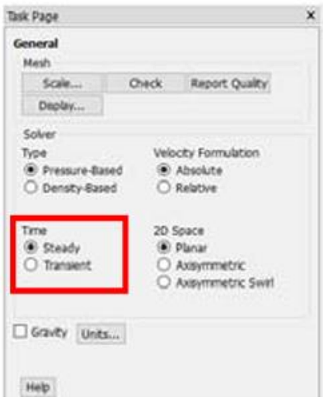
$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) + \frac{\partial}{\partial z}(\rho w) = 0$ Continuity(steady flow)

$\frac{\partial \rho}{\partial t}$→ 0

Navier-Stokes (steady flow):

$$\rho g_x - \frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} \vdash \frac{\partial^2 u}{\partial y^2}\right) = \rho \frac{Du}{Dt}$$

$$\rho g_y - \frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} \vdash \frac{\partial^2 v}{\partial y^2}\right) = \rho \frac{Dv}{Dt}$$



Define Incompressible Flow:

$\rho\left(\frac{\partial}{\partial x}u + \frac{\partial}{\partial y}v\right) = 0 \rightarrow \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$ Continuity (Incompressible Flow)

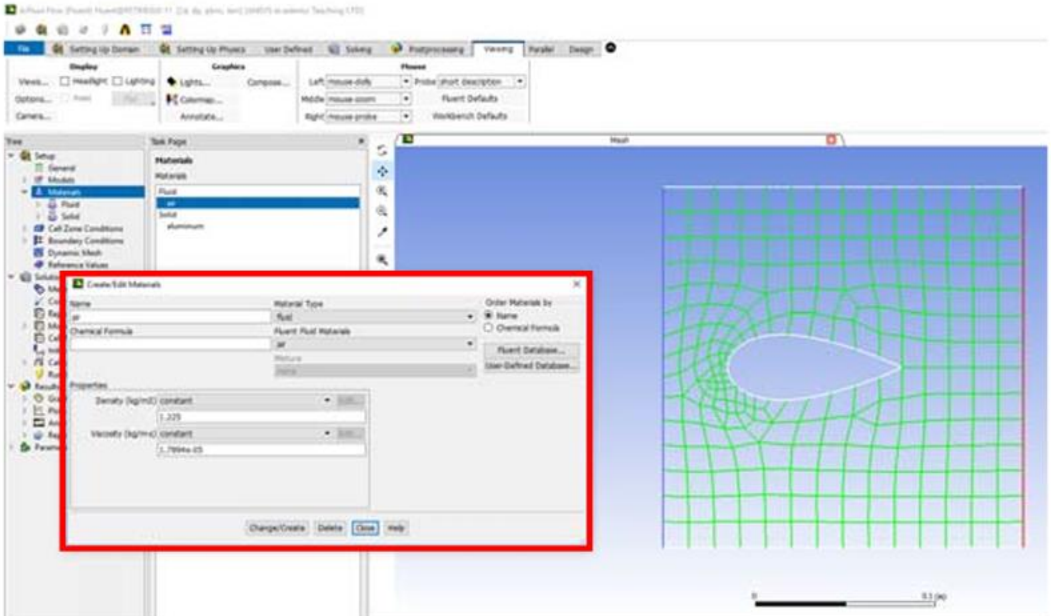
Navier-Stokes(Incompressible Flow)

$$\rho g_x - \frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} \vdash \frac{\partial^2 u}{\partial y^2}\right) = \rho \frac{Du}{Dt}$$

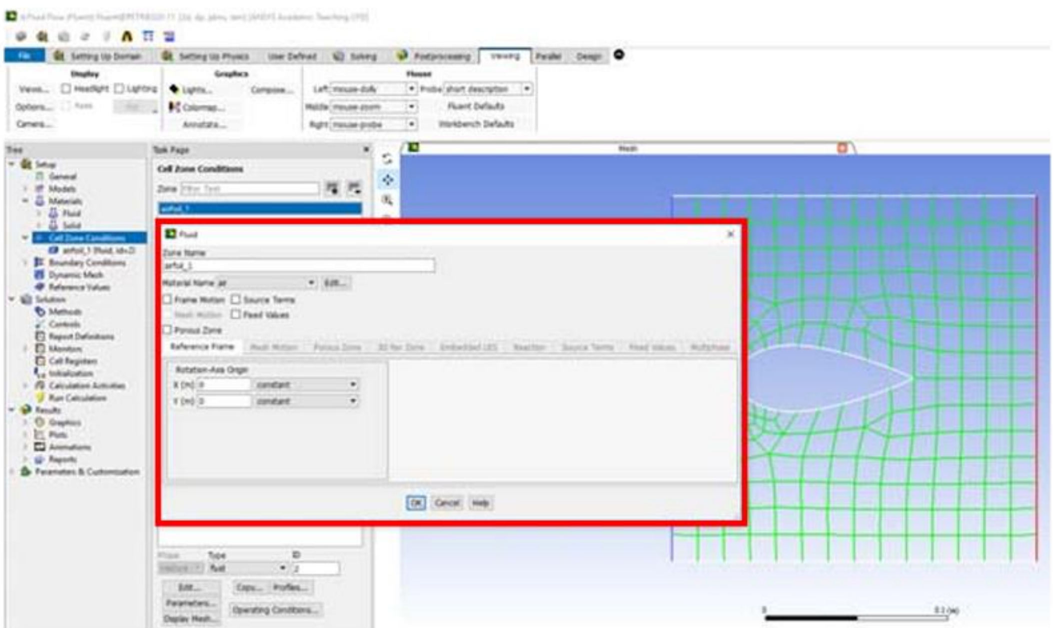
$$\rho g_y - \frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} \vdash \frac{\partial^2 v}{\partial y^2}\right) = \rho \frac{Dv}{Dt}$$

Define material

A material can be selected (in this case air), or any other material with user defined properties (UDF), this allows the user to later apply the material for the system



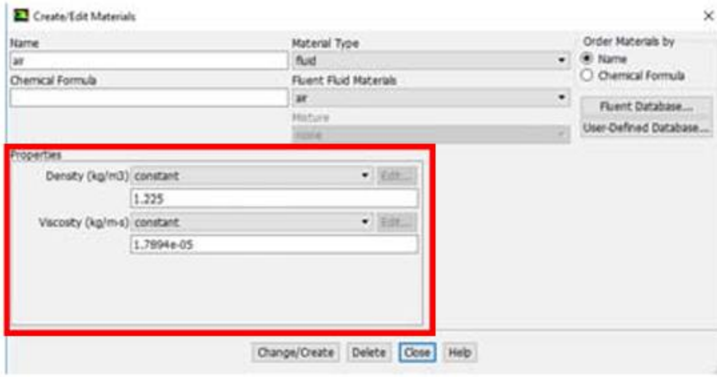
This option assigns the material to the system, in the fluid region



Define material Density (ρ) and viscosity (μ) are known:

$$\rho g_x - \frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} \vdash \frac{\partial^2 u}{\partial y^2}\right) = \rho\frac{Du}{Dt}$$

$$\rho g_y - \frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} \vdash \frac{\partial^2 v}{\partial y^2}\right) = \rho\frac{Dv}{Dt}$$



Boundary Conditions:

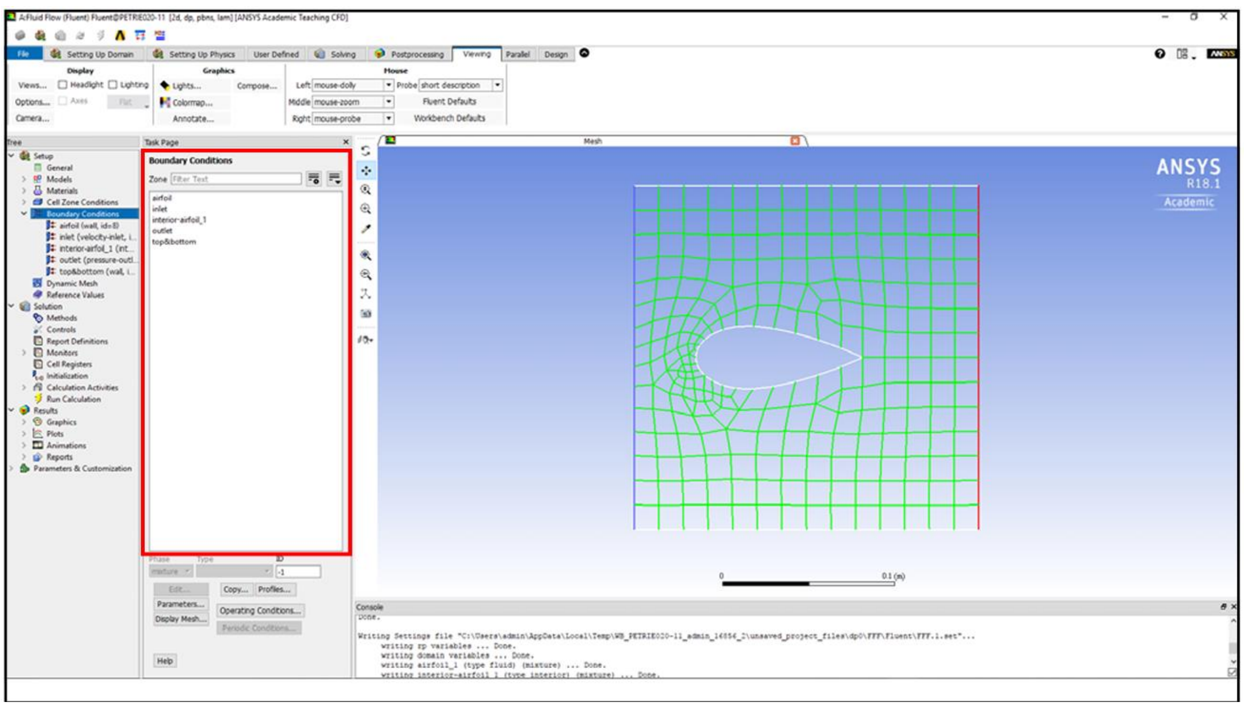
Now that the material is known and the equations to be used are simplified, we can specify the boundary conditions:

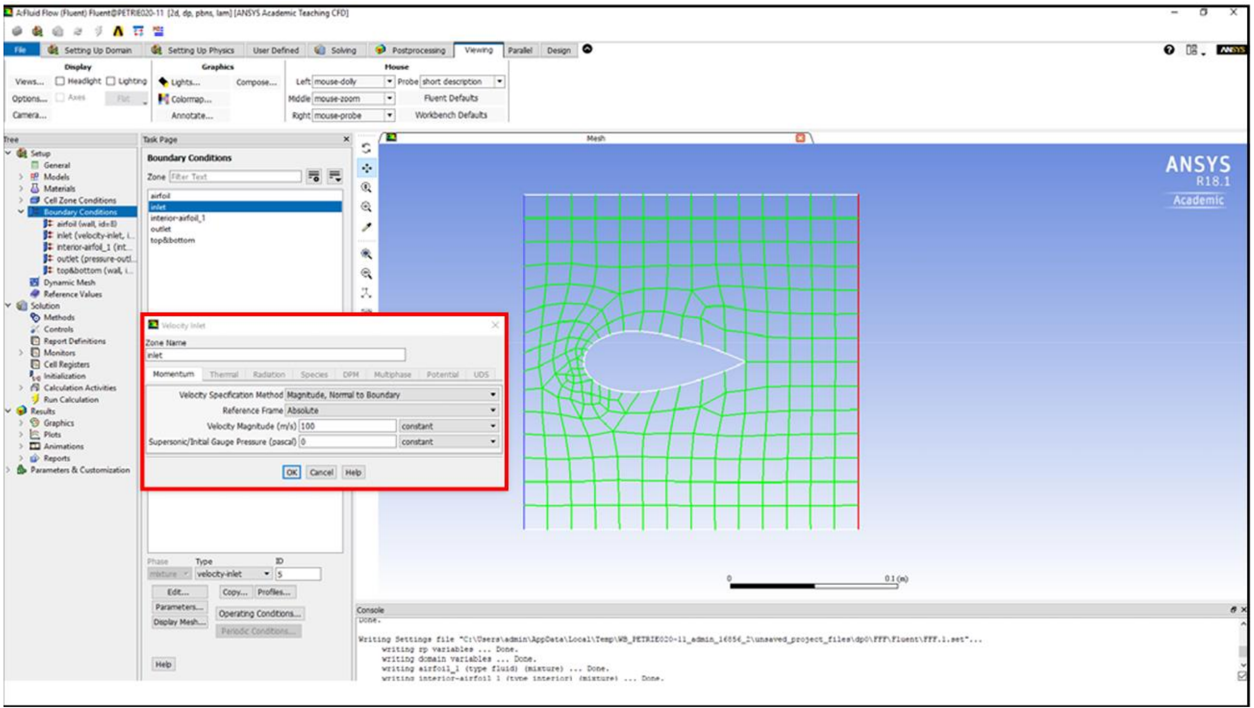
Continuity(Steady and Incompressible Gas) $\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$

Navier-Stokes equations (constant density and viscosity)

$$\rho g_x - \frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) = \rho\frac{Du}{Dt}$$

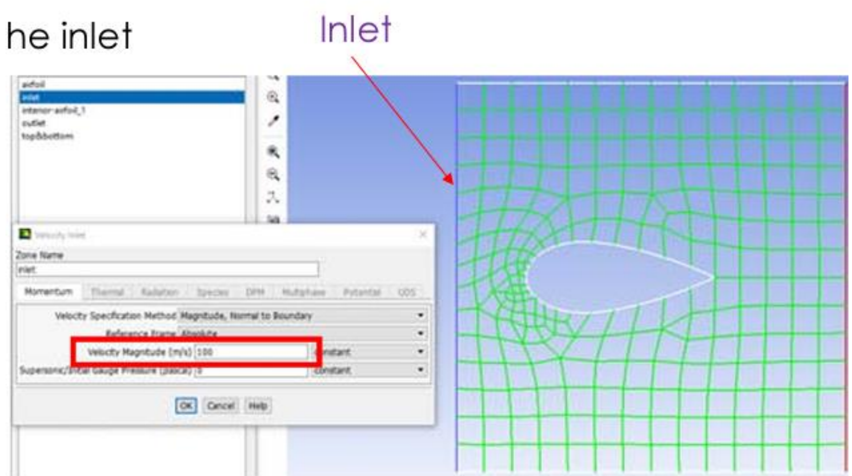
$$\rho g_y - \frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) = \rho\frac{Dv}{Dt}$$

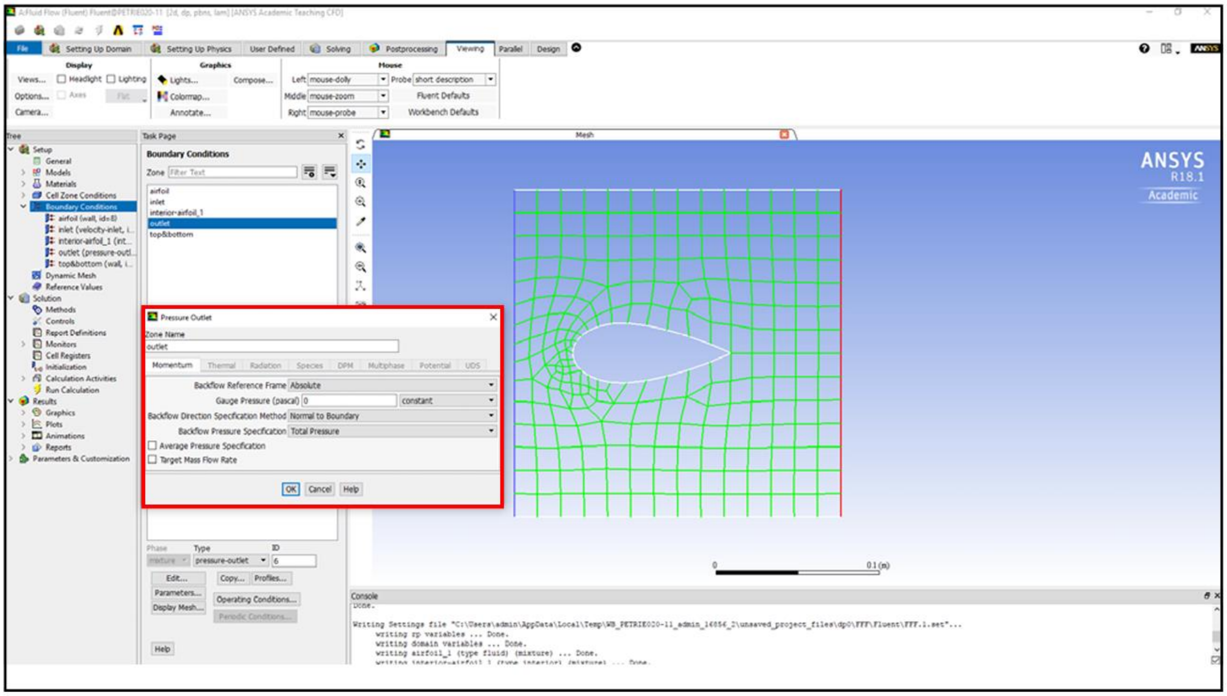




Boundary conditions cont'd:

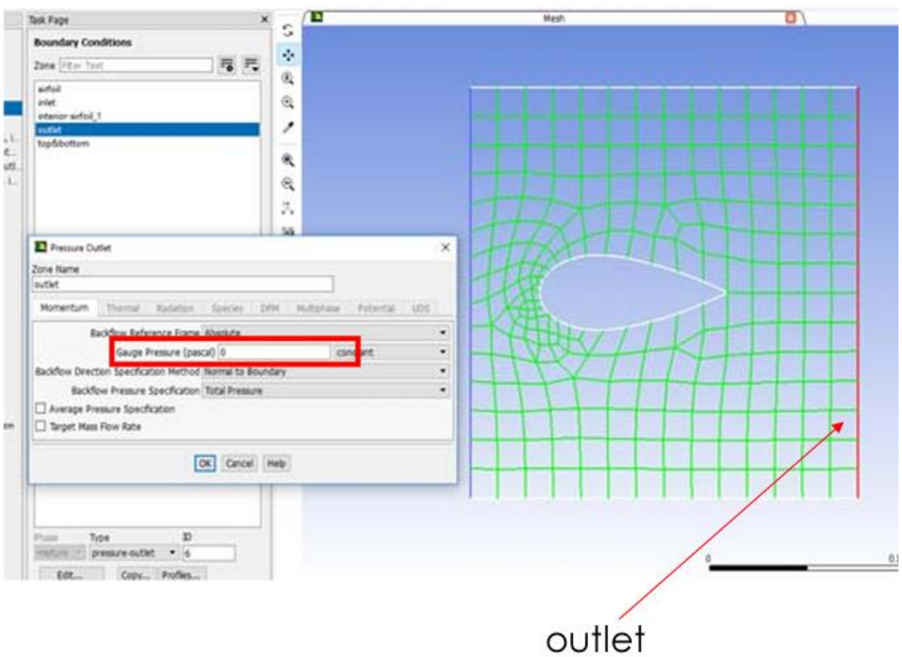
- Inlet velocity:
- u and v are known, u = 100, v = 0, at the inlet

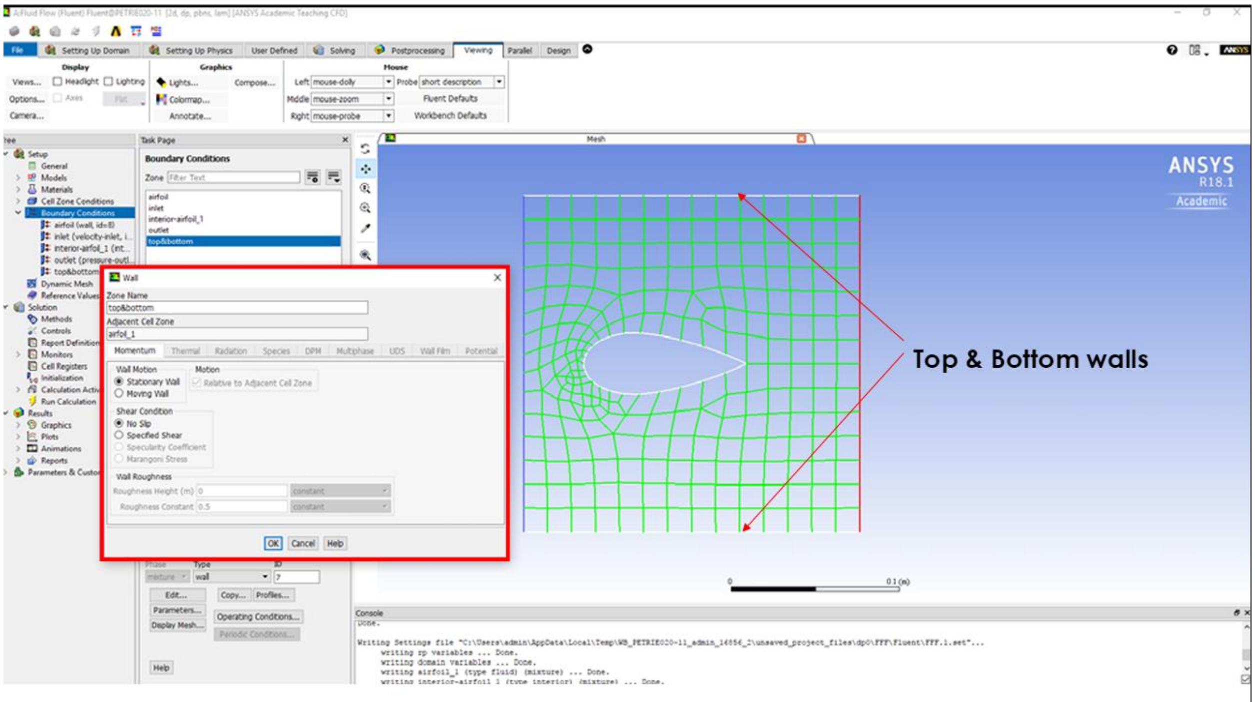


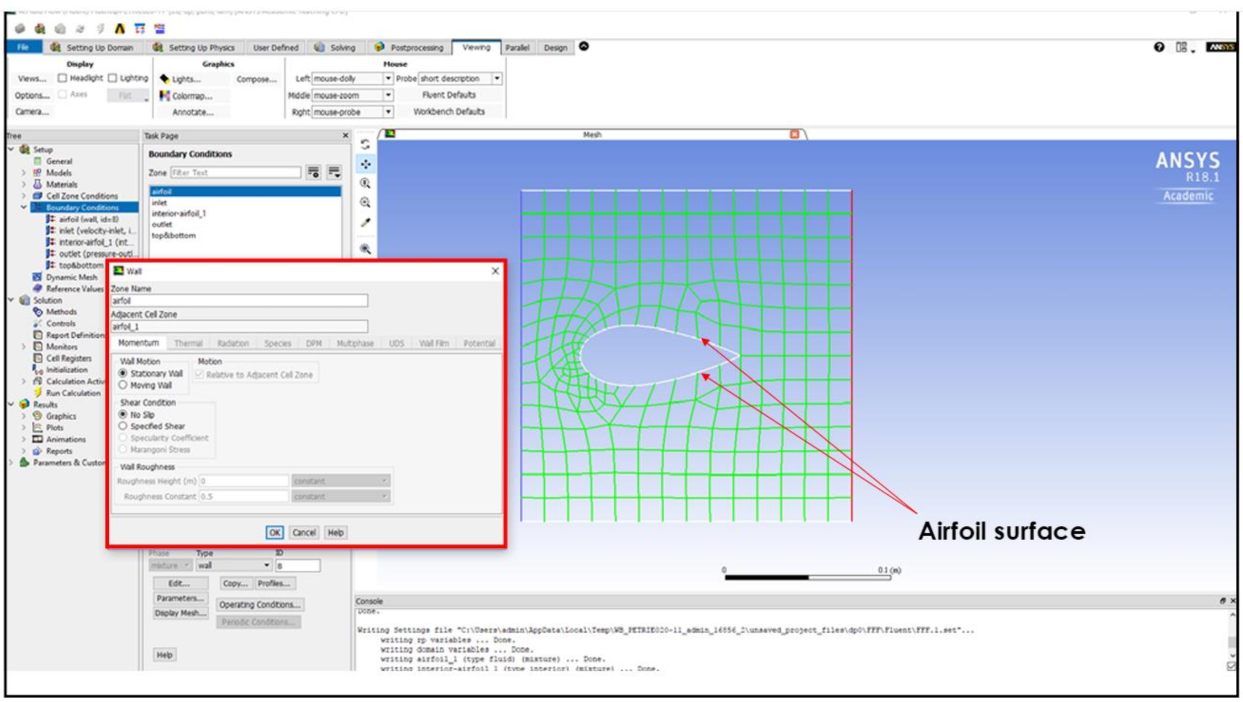


Outlet pressure

- Pressure is know

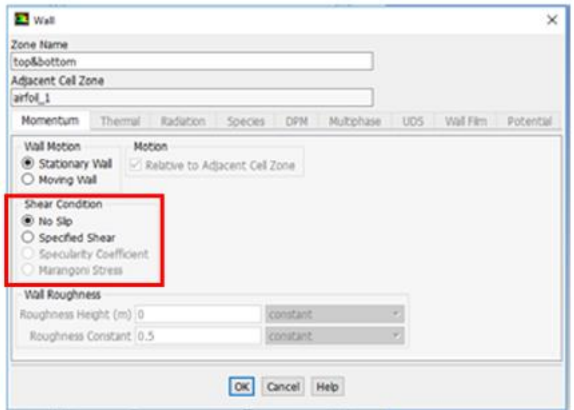


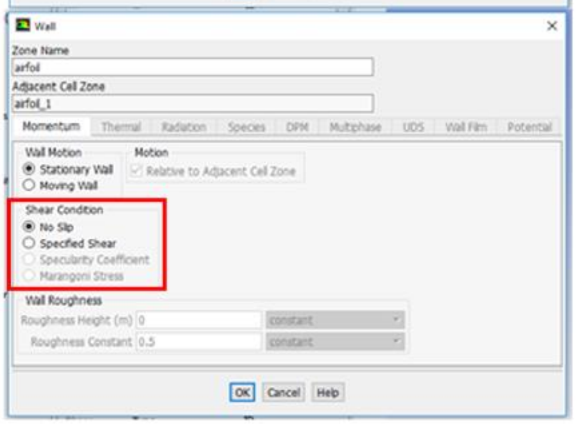




No slip condition on walls:

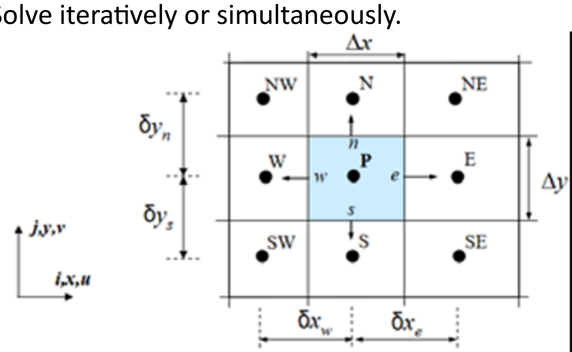
- Zero velocity on first fluid layer



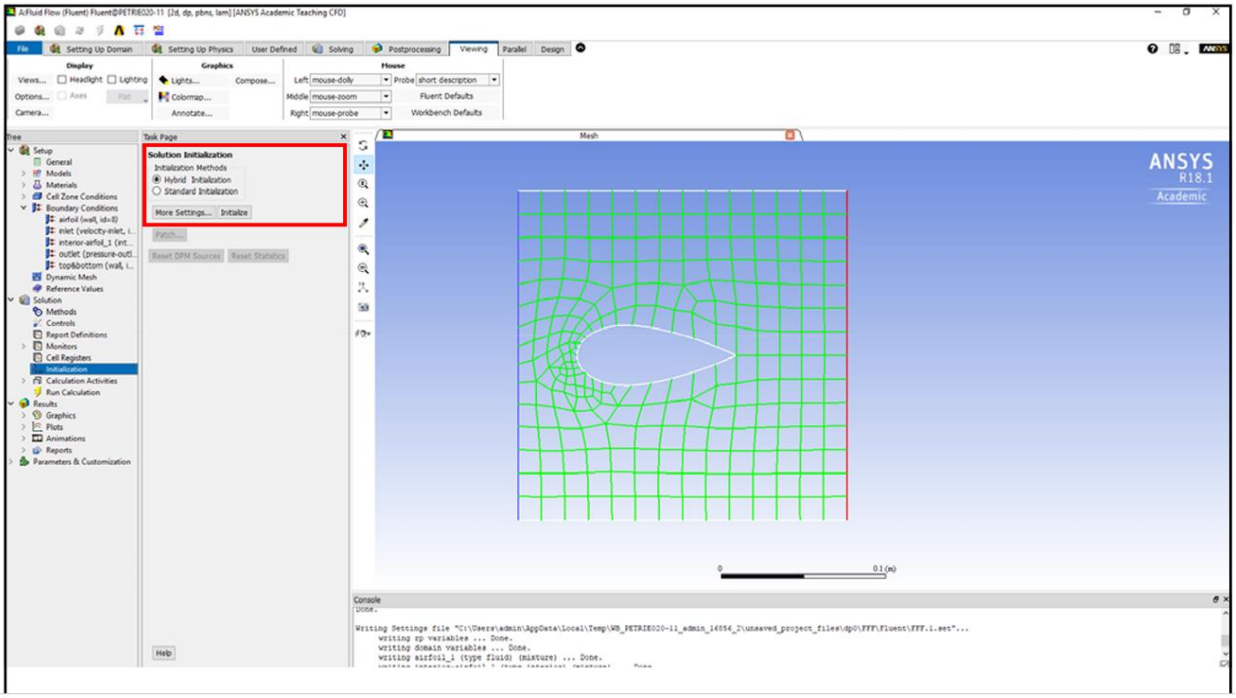


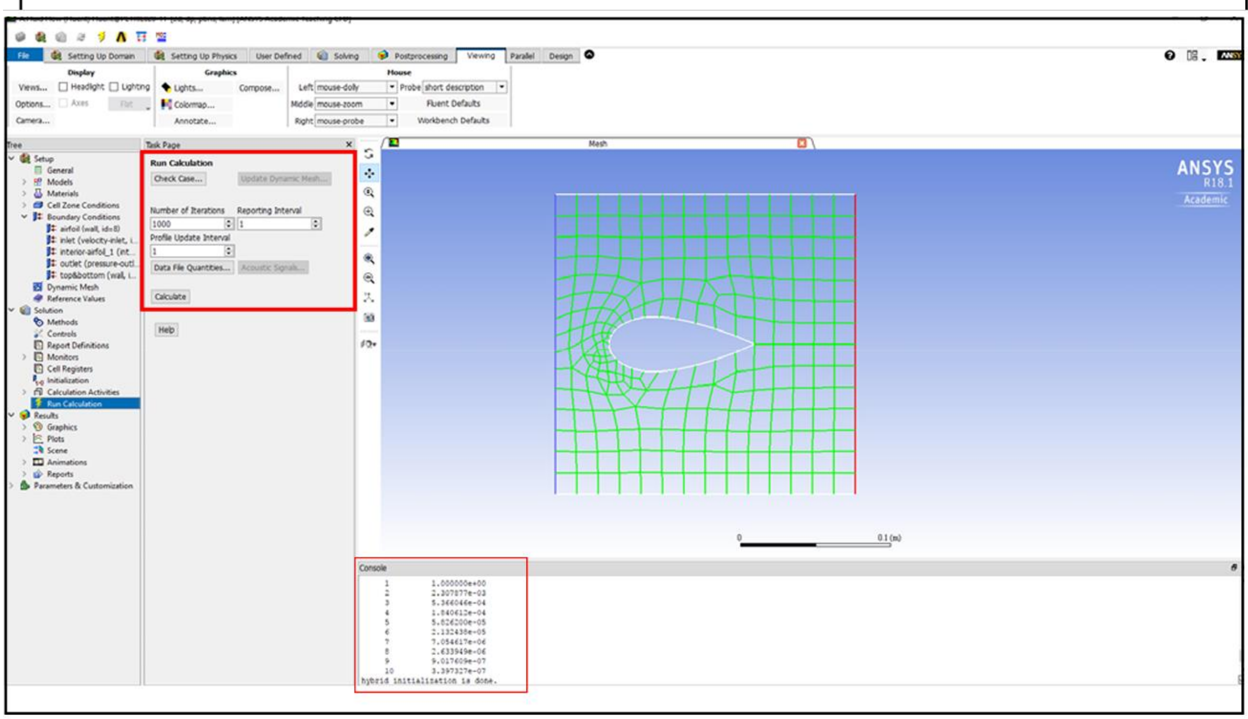
Finite Volume Technique (Fluent)

- Divide the domain into control volumes (elements)
  - This is the mesh we created
- Integrate the differential equation over each control volume and apply the divergence theorem
  - To evaluate derivative terms, values at the control volume faces are needed:
    - have to make an assumption about how the value varies (this is called initialization) –different from Boundary Conditions.
- All above results in: a set of linear algebraic equations
  - Note each element is providing 3 PDE in our example, we have 100s of elements (see the mesh), so the matrix will be a large one to solve
- Solve iteratively or simultaneously.

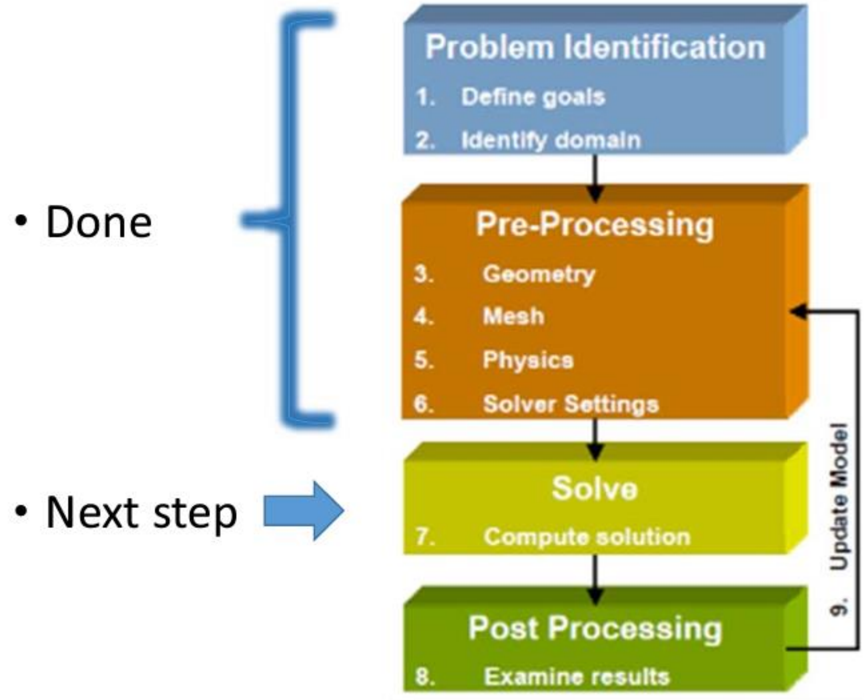


-



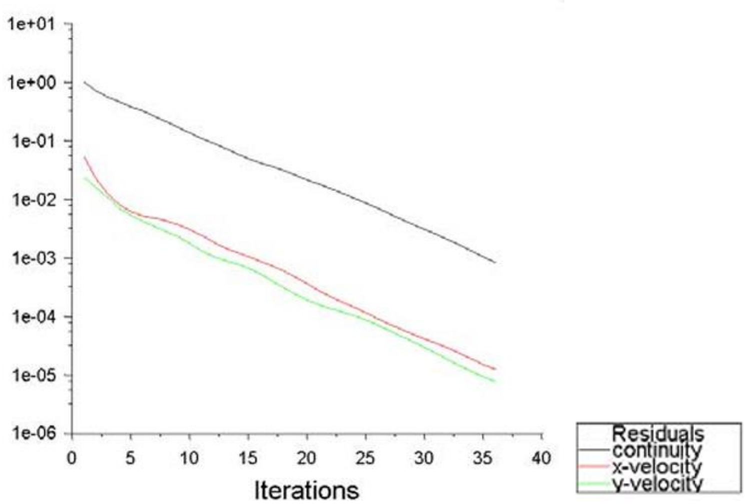


The problem is now all setup:

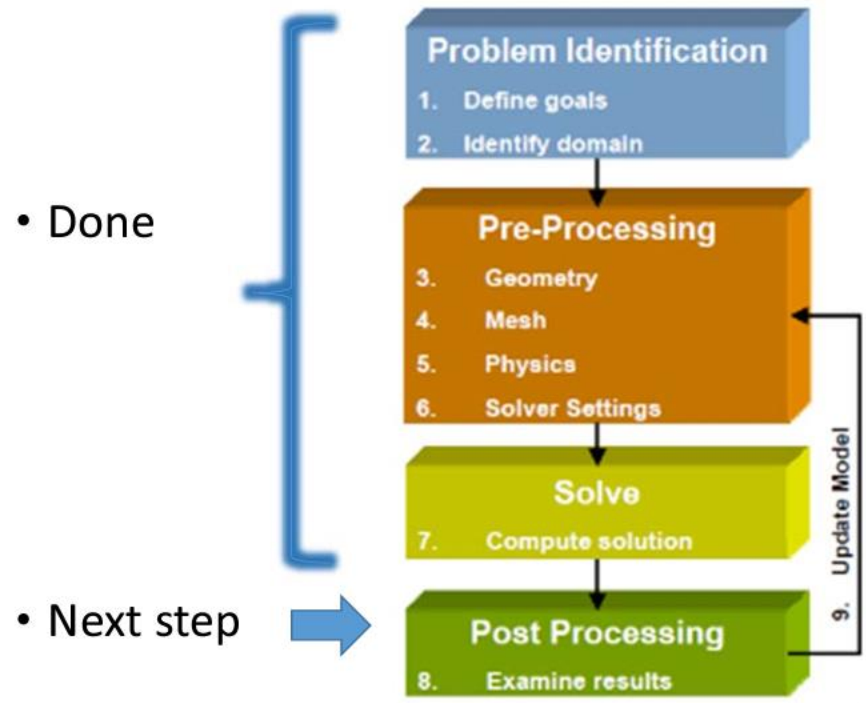


Residuals

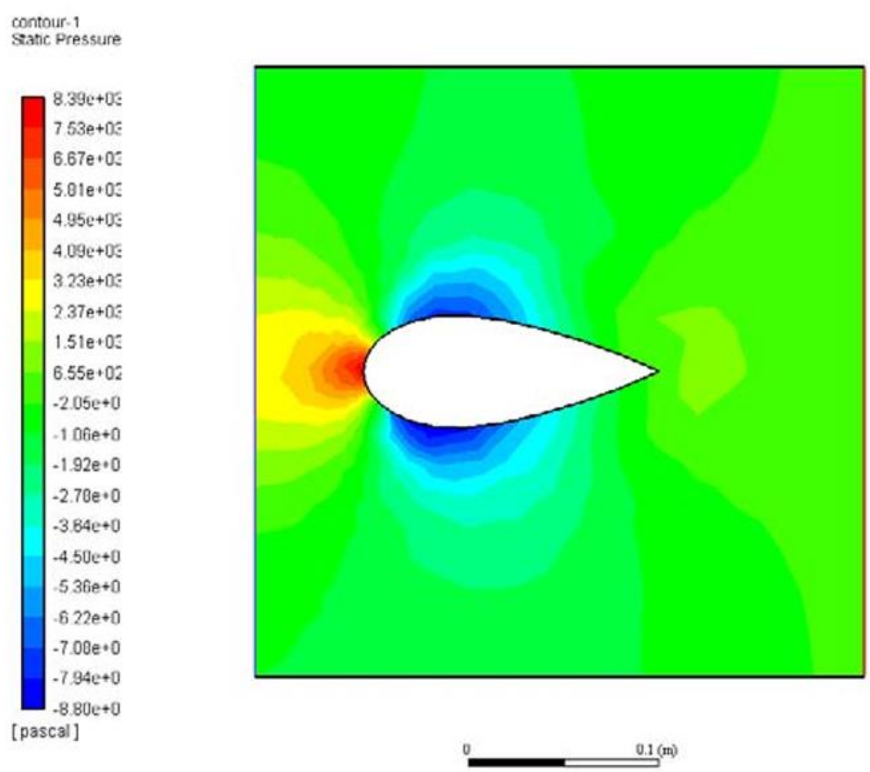
- The residuals are errors due to discretization of a continuous domain into mesh and solving over each element
- Usually an acceptable tolerance for this errors should be set --this called sometimes **convergence criterion**
  - The equations were allowed to have an error of 110
  - To start, It is a good idea to start with the default values in the software for the convergence criterion.
- In the FLUENT residuals are reported for each conservation equation.



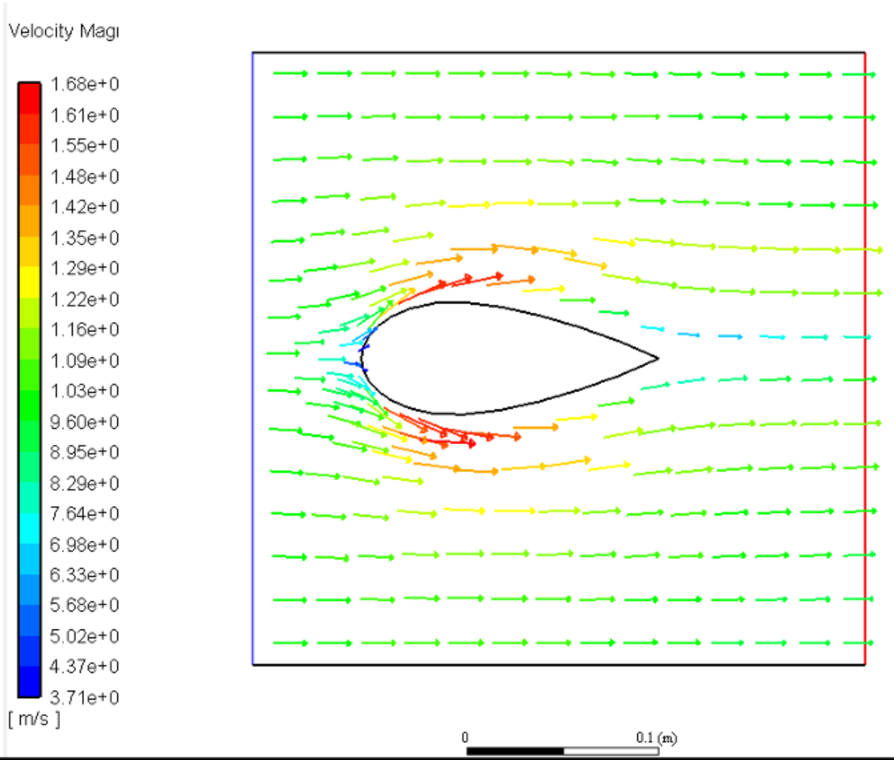
The problem is now solved



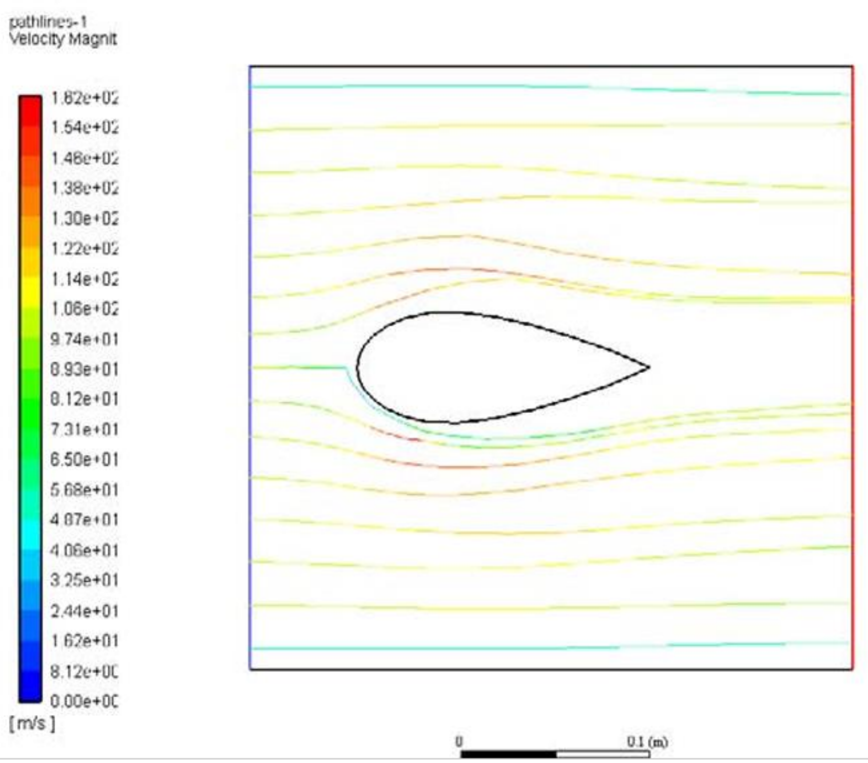
Results – Pressure contours;

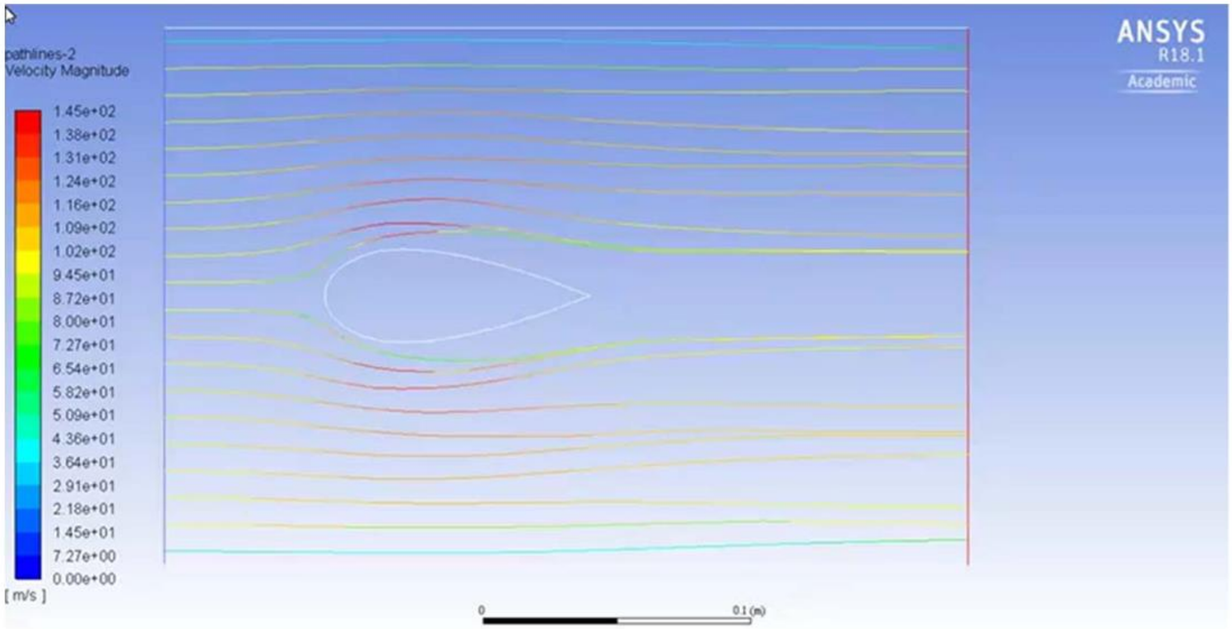


Results –Velocity Vector Field

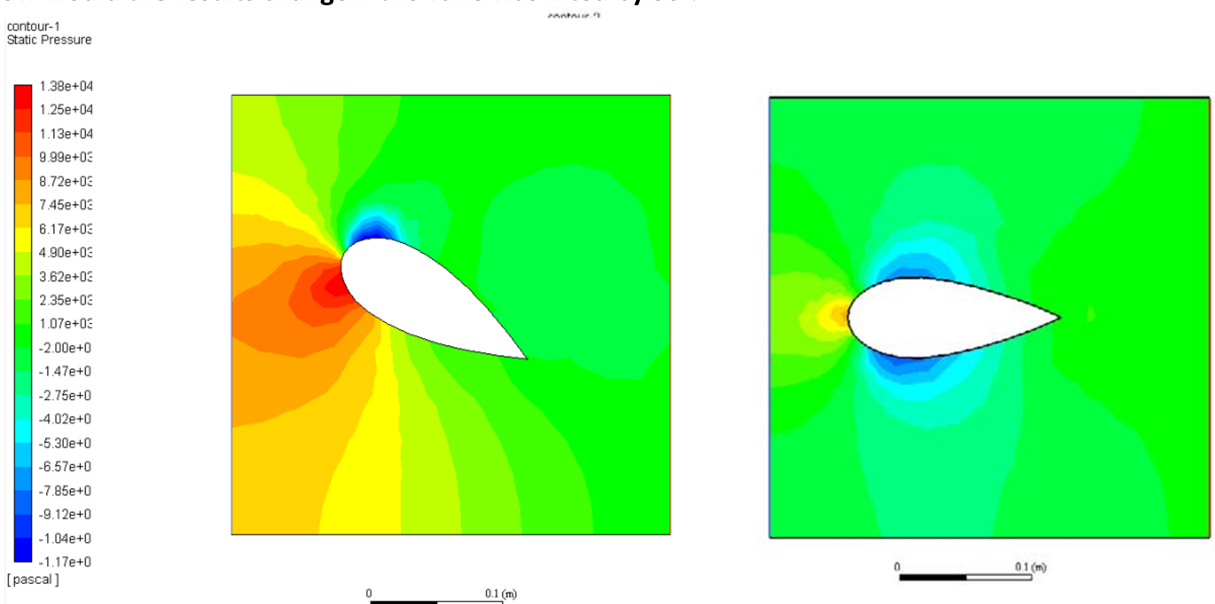


Results –Pathlines by Velocity Magnitude



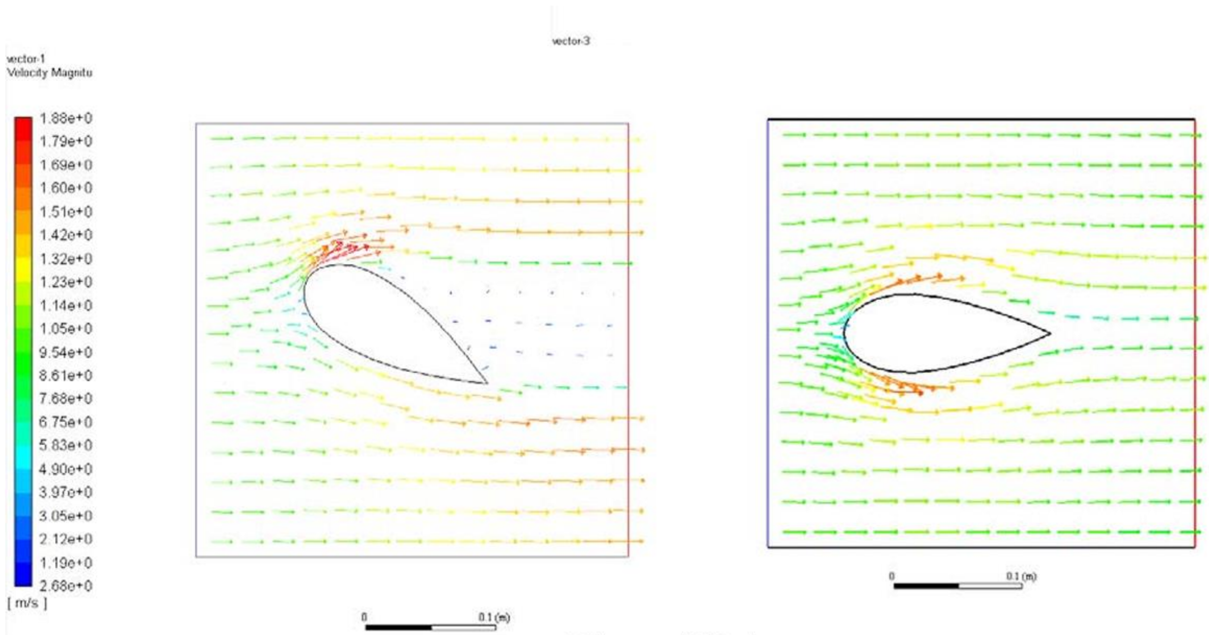


How would the results change if the vane was lifted by 30˚?



Contour plot representation of Pressure values

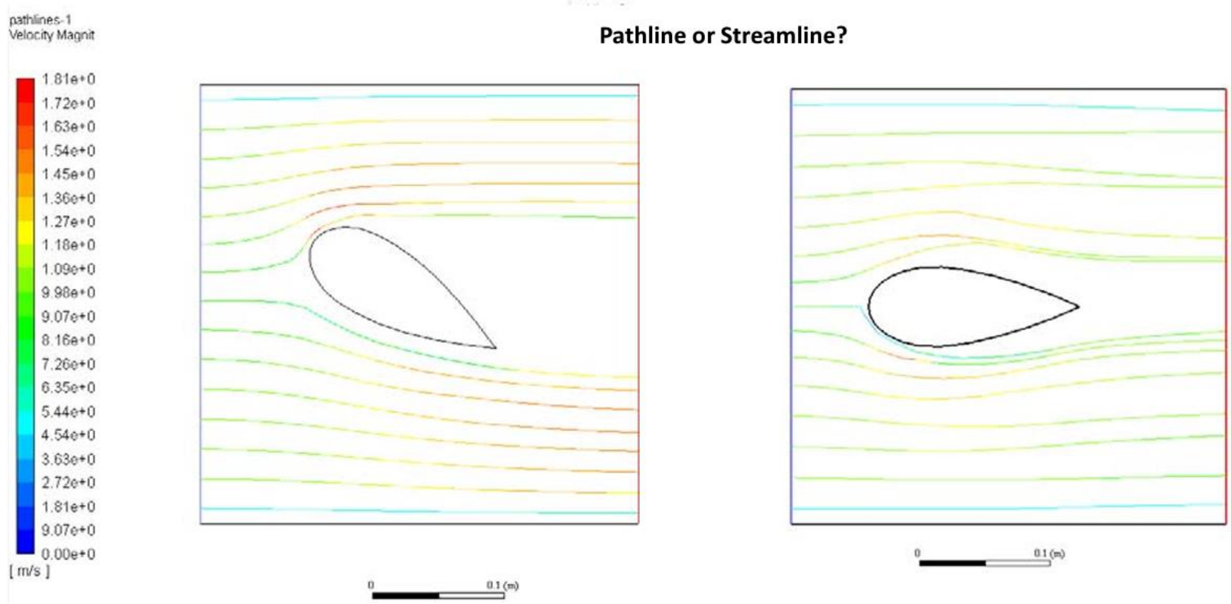
Velocity field -sensitivity/design analysis



Vel. vector (V) is shown

Streamlines

Pathline or Streamline?



The end!