Danial Nayyar

Investigation and Simulation of Fluid Flow in Curved Pipes: From Dean Vortices to Structural Integrity Analysis

Introduction & Literature Review

Fluid analysis has been a topic of study for many centuries, and many notable figures have contributed to the field's advancements including Osborne Reynolds (Douglas, et al., 2011) who is credited with Reynolds number which depicts if a fluid's flow pattern is either laminar or turbulent. The Dean number, which is a topic in this paper, was named after the British scientist W.R. Dean, it details the behaviour of fluid flow patterns through a curved pipe by utilising a straight pipe connecting to a pipe with a small bend. The work of Dean stemmed off the work of Osborne Reynold's Reynold number. Dean analysed fluid motion at low Reynold numbers i.e., laminar flow, through which it was observed that there is secondary flow, appearing as 'counter rotating cells 'aptly being called Dean vortices. (Kalpakli, 2012)

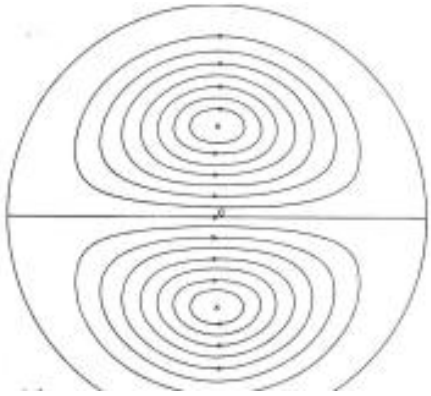

Figure 1: Depiction of Dean Vortices (Kalpakli, 2012)

The study of fluid flow in curved geometries is not only of interest in man-made such as pipes and car exhausts etc but is also vital in the study of rivers and their meanders as well the blood flow through veins and arteries of the human body (Devakar & Smarth, Mayuri, 2023). The reason fluid flow in curved pipes is analysed deeply is due to the implications of these vortices on the pipe structures. Due to the bend, there is a sudden change of motion for the fluid particles which as a result causes more force to be exerted on the outer wall of the pipe, which can lead to failure. This can lead to monetary losses for example in slurry transportation, due to damaged equipment, loss of the slurry being transported. Reasons such as these are why fluid flow in curved pipe geometries might be investigated. The formation of these secondary flows contributes to pressure losses, this is due to centrifugal causing a pressure gradient between the outer and inner walls of the pipe, this gradient will lead to losses hence the reason for the investigation. (Dutta & Nandi, 2016). Throughout this paper there will be key areas that are focused on, ranging from the discussion of flow patterns and how they are affected due to curvature ratio, how velocity profiles and pressure gradients are affected as well as the development of secondary flows and the effect of the dean number on secondary flows.

The Effect of Curvature Ratio on Flow Patterns and Axial Velocity Profiles

The curvature ratio is defined as the ratio between the radius a pipe can bend, R, to the diameter of the pipe, D. This curvature ratio is used to analyse how tightly a pipe can bend, until it is no longer safe and is prone to failure. This failure occurs because a bent pipe experiences both compression and tension on the inside and outside wall, respectively.

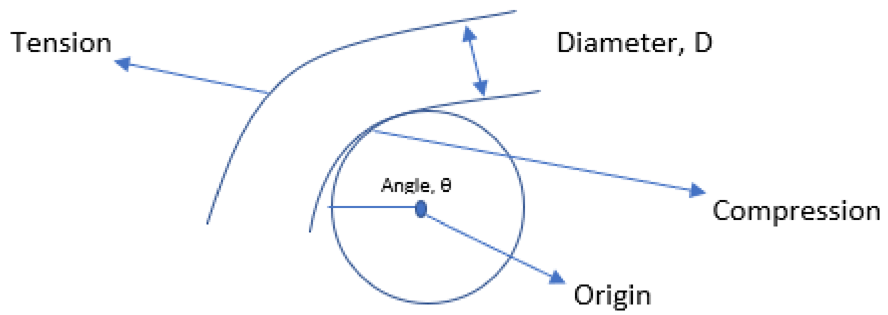

Figure 2: Curvature Ratio

An analytical study (Devakar & Smarth, 2023), investigated the flow of immiscible fluids through curved pipes and discovered that an increase in curvature ratio of the pipe i.e., increasing the amount the pipe is bent, increases the axial velocity towards the outer boundary of the pipe. Devakar & Smarth found that the development of secondary flows begins as curvature ratio continues to increase because of the difference between the centrifugal force at the inner and outer boundaries of the pipe. Devakar and Smarth attribute this increased axial velocity to the increased centrifugal force that the larger bend causes.

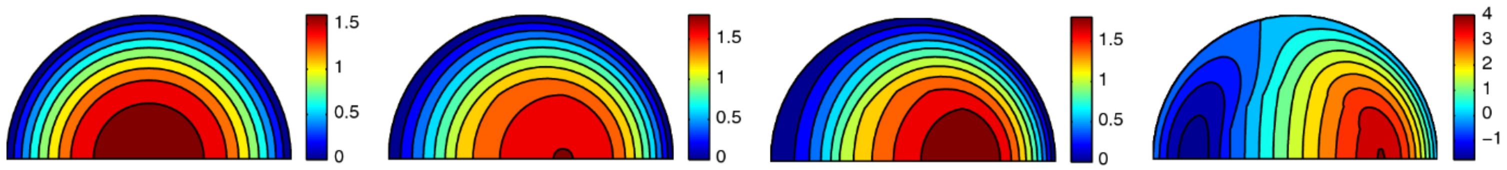

Figure 3: Effect of Curvature Ratio on Axial Velocity (Devakar & Smarth, 2023)

Fig 3 Displays the effect of increasing curvature ratio on the axial velocity and the more the curvature ratio increases the more distorted the fluid flow becomes, until eventually the secondary vortex flows begin to occur.

Devakar and Smarth' work show how the velocity profile changes due to an increase in curvature ratio. Figure 3a displays the velocity profile for a straight pipe by the lack of distortion in the velocity profile. The distortion of the velocity profile is evident and there is a clear trend between the distortion and the curvature ratio, as the curvature ratio increases the distortion does so as well. The level of distortion increases to the point where at maximum R/D of 0.05 Devakar and Smarth reported the development of secondary flow also known as dean vortices. This is also seen in the numerical study by Dutta and Nandi, they investigated the effect on velocity and pressure due to bend curvature in a 90° bended pipe. In this numerical study the curvature effect was analysed via a numerical simulation.

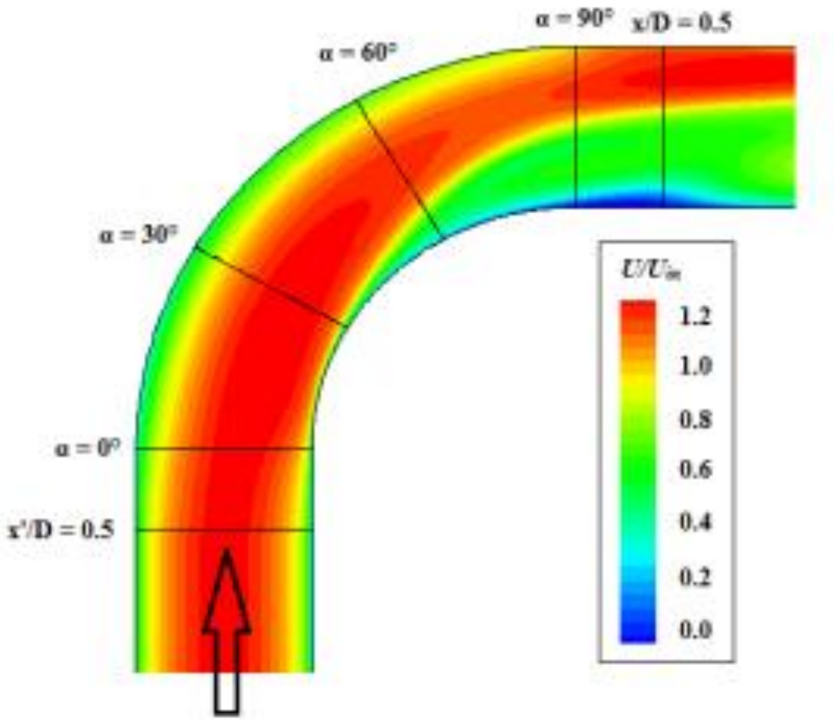

Figure 4: Plot of Axial Velocity in the pipe

Figure 4 above depicts how the velocity profile changes due to curvature of the pipe, it displays it shifts from the core and becomes less symmetrical as the pipe bend. In the straight section of the pipe the fluid appears to be observing the no-slip condition i.e., the maximum velocity is in the centre of pipe and velocity is minimal (very close to zero), however as the fluid approaches the bend the fast flowing core shifts towards the outer edge of the pipe due to the inertia forces/ centrifugal force and instead of the fastest flow being in the middle it ends up being in the outer flow, their data also shows that the max velocity fluid 'hugs' the outer wall even after the bend where the pipe is straight.

Pressure Gradient across Pipe Cross-Section

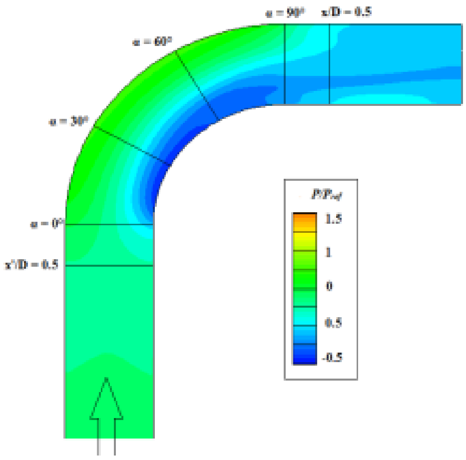

Figure 5: Pressure Gradient in curved pipe

Figure 5 displays how pressure changes in a bending pipe, the change begins to occur during the bend, from figure 5 during the straight section of the pipe there is no pressure gradient as seen by the green colour in the contour plot. However, as the fluid begins to go around the bend, there is a pressure drop at the inner wall, relative to the outer wall, this change is emphasised in figure 6, when α = 30°, 60° and 90° a gradient is observed. Dutta and Nandi attribute this to the bend and the centrifugal force caused from the circular motion, caused by the bend, of the fluid particles this causes the outer wall to experience a larger pressure and the inner wall experiencing a smaller one. Referring to figure 5, it can be observed that even after the bend the there is a significant pressure drop compared, which carried on from the inner wall to the inlet of the pipe, which could cause problem in transporting fluids. Dutta and Nandi concluded that after the outlet there is an attempt by the flow to stabilize itself and normalised pressure is less at the outer wall and more at the inner wall. (Dutta & Nandi, 2016)

Secondary Flows and Dean Number

The introduction of curved geometries gives rise to the development of secondary flows known as Dean vortices, named after W.R. Dean. (Kalpakli, 2012) These vortices develop due to the curvature of the pipe and as seen in figure 3 they develop when the curvature ratio is increased.

The Dean number according to Kalpakli:

$$D_e = \mathrm{Re}\sqrt{\frac{R}{Rc}} \quad Equation\ 1$$

Re = Reynolds Number
R/Rc = Curvature Ratio

Kalpakli stated that over the years the work of Dean had been extended by figures such as McConalogue & Srivastava who solved the equations numerically by Fourier-series expansion. McConalogue and Srivastava's work showed that secondary flows become more dominant as the Dean number is increased and this can be done by increasing the curvature ratio (assuming Reynolds' is kept constant). Kalpakli defines dean number as the parameter that describes the dean vortices, which appear as two symmetrical roll-cells, it is described as this because the mean axial velocity can be measured, which is vital as an experimental study like Kalpakli requires measurements. (Kalpakli, 2012)

Another study aimed to understand if the Dean number can characterize flow similarity in bent tubes (Cieslicki & Piechna, 2012). This study featured experimental and numerical results and compared the two. Cieslicki and Piechna used an experimental setup consisting of pressure transducers, different shapes of pipes i.e., coil type, pipes that appeared like sine waves.

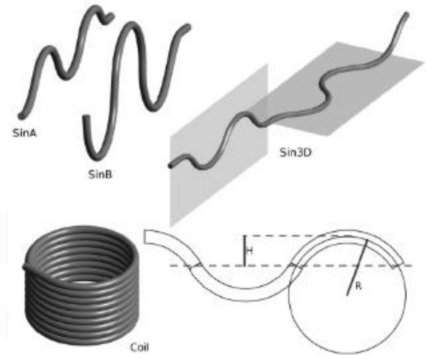

Figure 6: The Tube Structures used in the Cieslicki & Piechna study

Their results indicated that there was a significant overlap between the dean number and the shapes of their pipes, even for the sin3D tube. This significance is due to the fact that the sin3D pipe would have resulted in a more complex velocity pattern due to the alternating centrifugal forces. Cieslicki & Piechna concluded that the dean number can serve as a parameter of laminar flow in diffferent bent tubes for a limited range of dean number and curvature ratio.

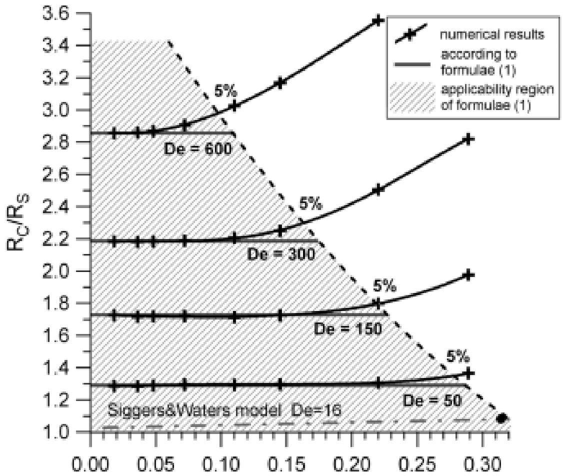

Figure 7: Flow Resistance ration vs curvature ratio for constant Dean Values (Cieslicki & Piechna, 2012)

Their graph above shows the relation of fluid flow resistance ratio (flow restance of bended, Rc, tube to flow resistance of striaght tube, Rs). The graph shows how the increase in curvature ratio impacts the flow resistance ratio and that values can be used to describe fluid flow for many curvature ratios.
Cieslicki & Piechna also describe how their analytical solution descibes the fluid flow for dean numbers below 20, however they admit that the for dean number above 20 their solution is no longer effective as it begins to diverge. Their numerical analysis describes a dependence of the flow resistance on the curvature ratio which can be better seen in figure

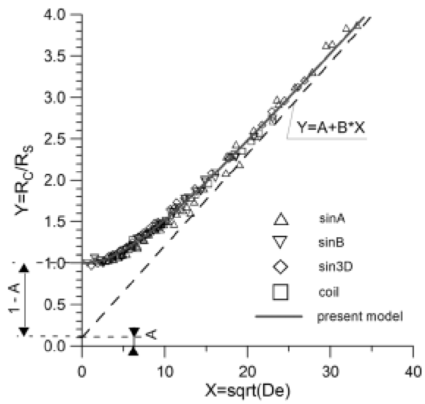

Figure 8: Flow Resistance vs Dean Values

Methodology (w/ Results & Discussion)

This section of the paper presents the findings of computational fluid dynamics (CFD) on a 3D flow in a curved pipe. The flow was modelled using the ANSYS software, it involved using both laminar and turbulent flow and then comparing them. The flow characteristics were examined in both a straight pipe and a curved pipe. The straight pipe features, and flow parameters used were taken from the research paper titled 'Measurement of fluid velocity development in laminar pipe flow using laser Doppler velocimetry' by Molki et al. This paper presented a non-intrusive method to obtain the velocity profiles in the smooth pipe experiencing laminar flow conditions with Reynolds number, R_e being 925 (Molki, et al., 2013) . The measurements of the pipe were used to design and simulate laminar flow through a straight pipe. This then led to the design and simulation of fluid flow in a curved testing various parameters including mesh density, flowrate and pressure.

Part A –

This part involves calculating the velocity profile of laminar flow in a straight pipe, the Molki paper depicted the diameter of the straight pipe as 24mm and the length of the pipe was set to 3000mm. As fluid enters the pipe it starts off with as an unstable flow and as the fluid progress through the length of the pipe it begin to stabilise. The velocity profile of the stable region in the pipe can be obtained to show how the fluid behaves in the straight laminar period of the flow in the pipe.
The equation for the velocity profile is also presented by Molki as:

$$u(r) = Umax\left[1 - \left(\frac{2r}{D}\right)^2\right] \quad Equation\ 2$$

(Molki, et al., 2013)

$$u(r) = velocity\ in\ radial\ axis$$
$$Umax = Maximum\ Velocity\ in\ pipe$$
$$r = radius\ of\ pipe$$
$$D = Diamter\ of\ the\ pipe$$

The equation above shows that the velocity of the fluid at any point in the pipe is a function of the pipe's radius. When *r=D* the velocity at the wall is equal to zero as well. This can be seen in the figure below:

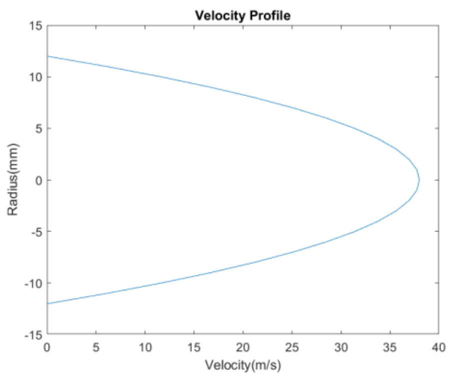

Figure 9: Velocity Profile (for the pipe in the Molki Paper)

Figure 9 shows how the velocity within the straight pipe fluctuates as a function of the radius of the pipe. It illustrates how maximum velocity is in the centre of the pipe, when *r=0,* i.e, the centre of the pipe, the fluid is flowing at maximum velocity. The reason for the velocity being zero at the wall is attributed to the no-slip condition. The no-slip conditions means that at the wall, the fluid is stationary and does not actually move. Using SolidWorks, the dimensions described in the Molki paper were used to create the straight pipe to then analyse using the Ansys software.

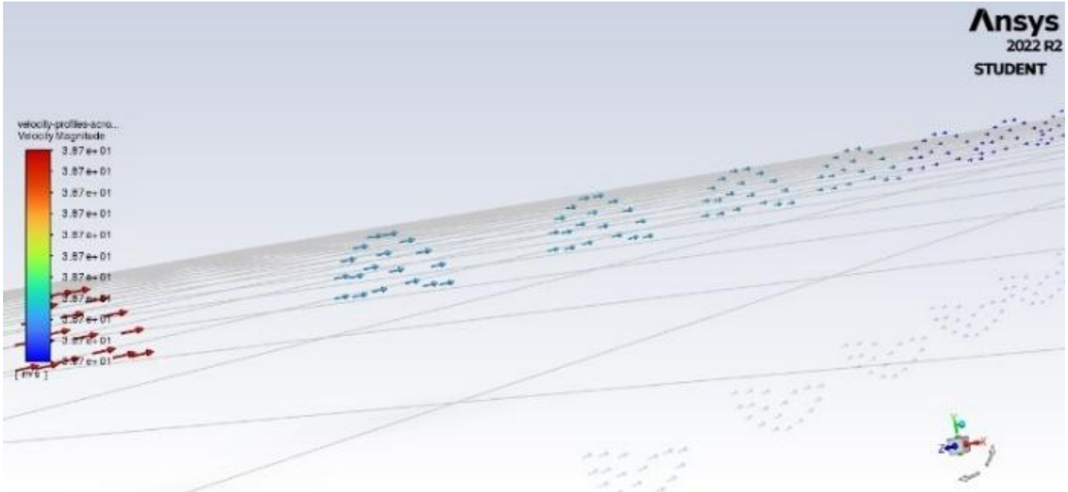

Figure 10: Change in velocity displayed as velocity vectors

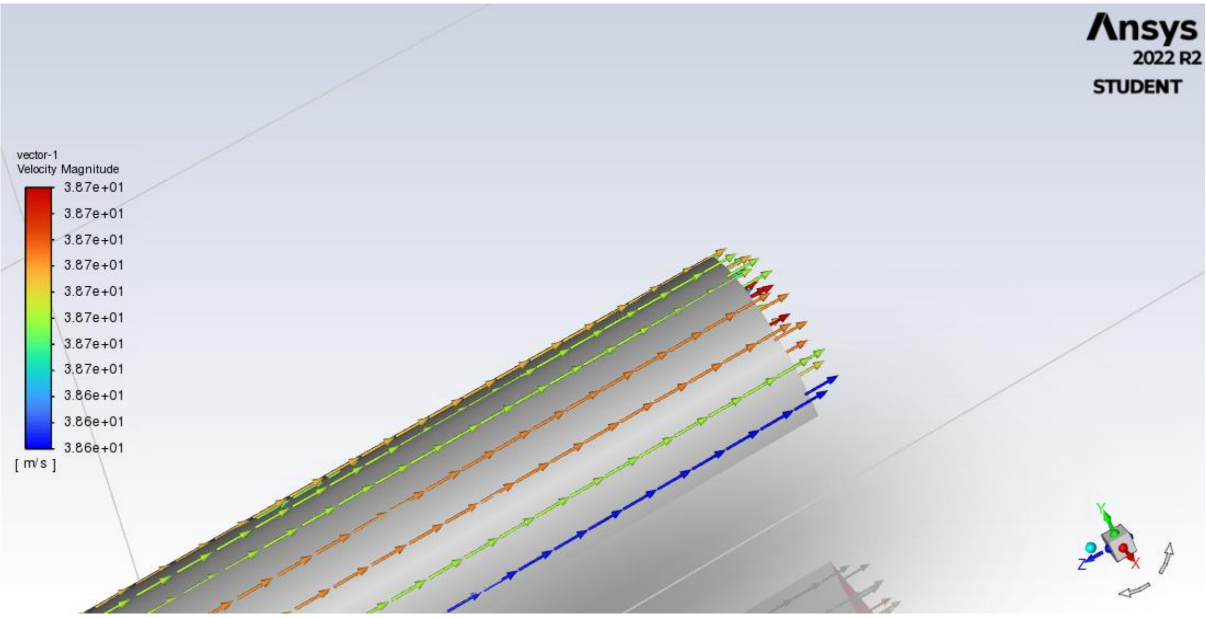

Figure 11: Vector plot of the velocity at outlet

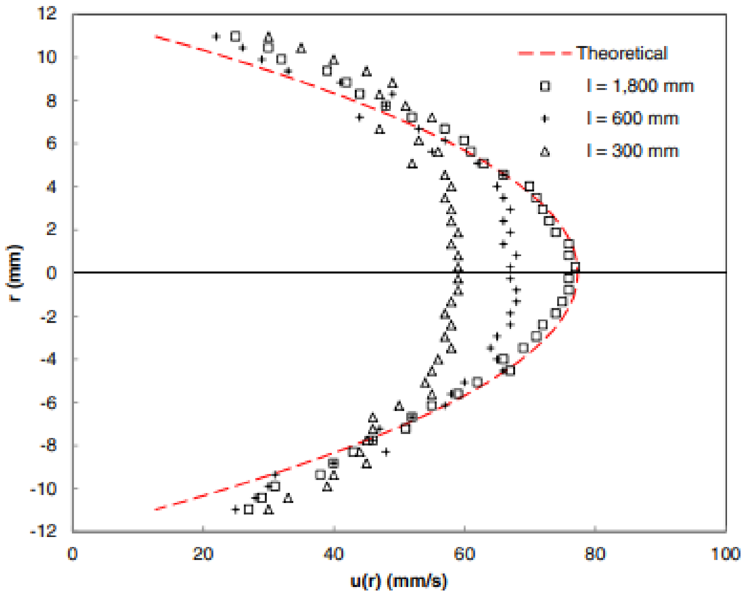

Figure 12: Results from the Molki Paper

Figure 12 above shows how the velocity varies across the pipe's length. This is re-enforced by the results obtained by Molki et al. The parabolic nature in figure 12 occurs due to the relationship of velocity and the pipe's radius. The equation above illustrates this parabolic relationship. Figure 11 displays the change in velocity from minimum at the boundary and maximum at the centre as shown by the colours blue and red respectively. To complete this analysis the pipe was modelled, and continuous domain was obtained by meshing the straight pipe. This domain was then subjected to boundary conditions (i.e. 0 pressure, inlet velocity of 38 m/s etc). A residuals test was carried out to show the change in velocity whilst setting the convergence rate to the order of 10^{-6}. The convergence rate represents the rate at which the sequence approaches end point i.e. the solution.

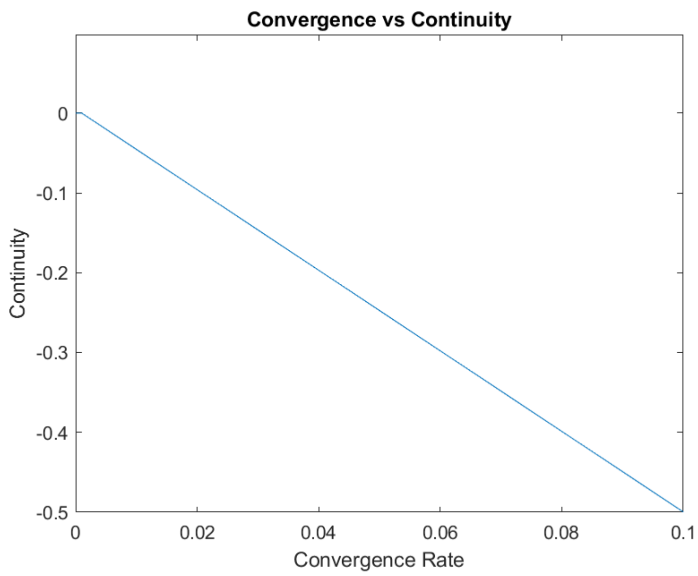

Figure 13: Convergence vs Continuity

Figure 13 above shows, graphically, how the increase in convergence rate results in the decrease in continuity. The effect of continuity is not only limited to the length of the tests, as by comparison the number of iterations also decreases as the convergence rate decreases. Although the reduction in test time can be seen as a positive the increase in error i.e. the continuity, would mean that the final values are inaccurate. The difference in final values obtained in the simulation and the experiments by Molki et al could be attributed to the convergence rate. The lower the order of the convergence rate the smaller the error, however Molki et al do not mention the use of their convergence rate. The error margin between the experimental and theoretical values obtained by Molki et al. is 0.5% ((Molki, et al., 2013). The data collection method by the LDV was to take the average of 20 signal bursts however, due to the limitations of the system it couldn't take readings from 1mm away from the wall due to light reflections (The pipe in the experiment was clear and could be the reason for the light reflections within the pipe and therefore the reason for the difference between the simulation results obtained in ANSYS Fluent and the experimental result as well as the theoretical results by Molki et al.) and low velocity near the wall.

Part B –

B.1) Mesh Density

The second part involves investigating the effect of the number of elements in the fluid's mesh. This test involved testing for different meshes, on the bend pipe, including the default provided by the Ansys software, as shown in table 2 below. The table below shows the varying velocity, shear stress and pressure at the same point in the pipe (in the bend) with respect to the number of elements. This test involved the fluid be in the laminar state with a velocity of 0.02 m/s. This was velocity was calculated by first assuming a Reynolds number, in the laminar region, of 1000 and then using the equation (2) below to calculate the velocity, u:

$$u = \frac{Re\,v}{\rho d} \qquad v = viscosity\ (0.001003,\ d = diametetr(50x10^{-3}) \quad Equation\ 3$$

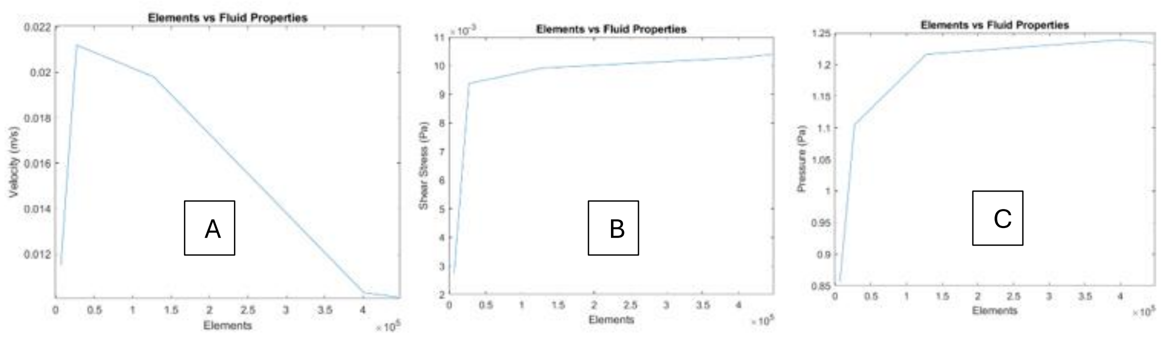

Figure 14: Number of Elements vs Fluid Properties

Figure 14 above clearly shows how, as the number of elements increase there is greater accuracy and figures 14 B and C show how there is a plateau forming suggesting that as the number of elements in the mesh increase the accuracy will increase as well however it is until a certain point only. After that point the number of elements will only increase the analysis time without benefitting accuracy. From table 1 below a comparison between test numbers 4 and 5 show that there is a percentage difference of 0.35%.

Test Number	Mesh Type (Element Size)	Number of Elements	Nodes	Velocity Magnitude at point (middle of bend) (m/s)	Shear Stress at point (middle of bend) (Pa)	Pressure (middle of bend)
1	Auto (0.1501)	6520	9483	0.011482772	0.0027497451	0.85618079
2	0.005	27132	34112	0.021211432	0.0093779772	1.1046921
3	0.003	127584	147242	0.019838112	0.009918997	1.2162042
4	0.002	400932	443889	0.010256268	0.010282873	1.2386325
5	0.00195	446028	491400	0.010115267	0.010403606	1.2343009

Table 0-1

This notion is further shown in the residuals plot for tests 1,4 & 5, as shown in figure 15 above.

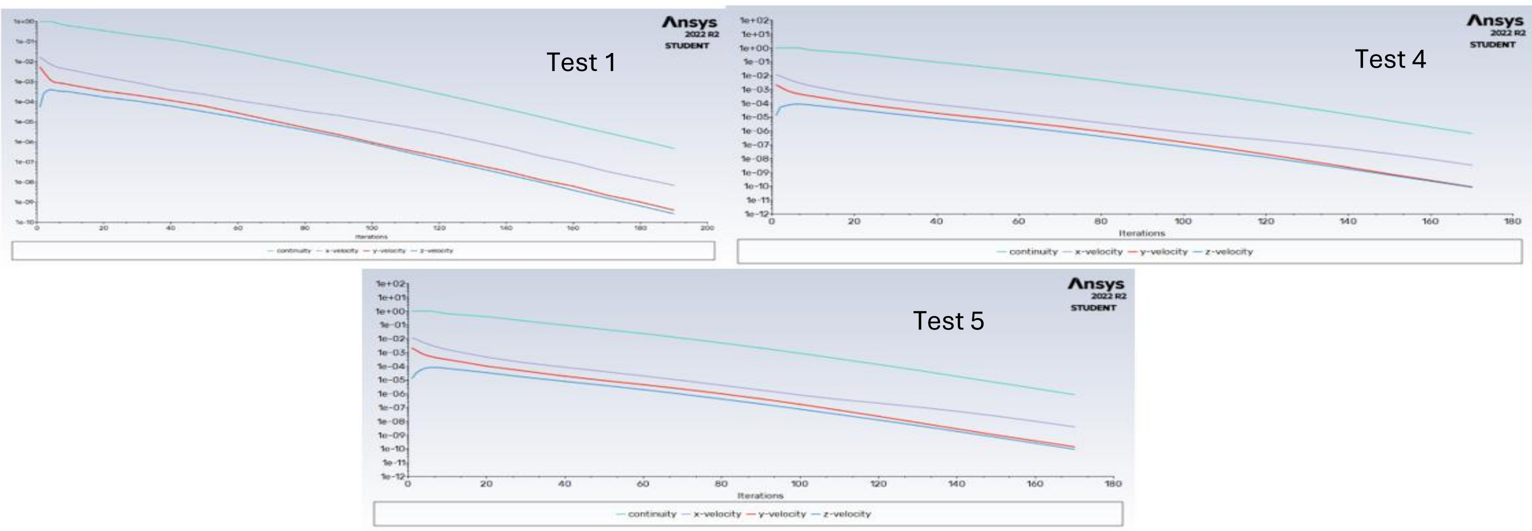

Figure 15: Residual Plots

B.2) Flow Rate

To understand the effect of flowrate within the pipe, different flowrates were tested. Ranging from an R_e of 1000, to give a velocity of 0.02 m/s, to $R_e = 2000$ giving a velocity of 0.04 m/s and finally an R_e value of 3000 to give a value of 0.06 m/s. These values were chosen as they represent the distinct flow regimes, with R_e equalling 1000 being in the laminar region, $R_e = 2000$ to the region it starts to from laminar to turbulent and finally when $R_e = 3000$ the flow is well off in the turbulent region.
This section will detail the how the change in flowrate will affect the velocities around the bend (and throughout the pipe), as well the change in pressure and shear stress.

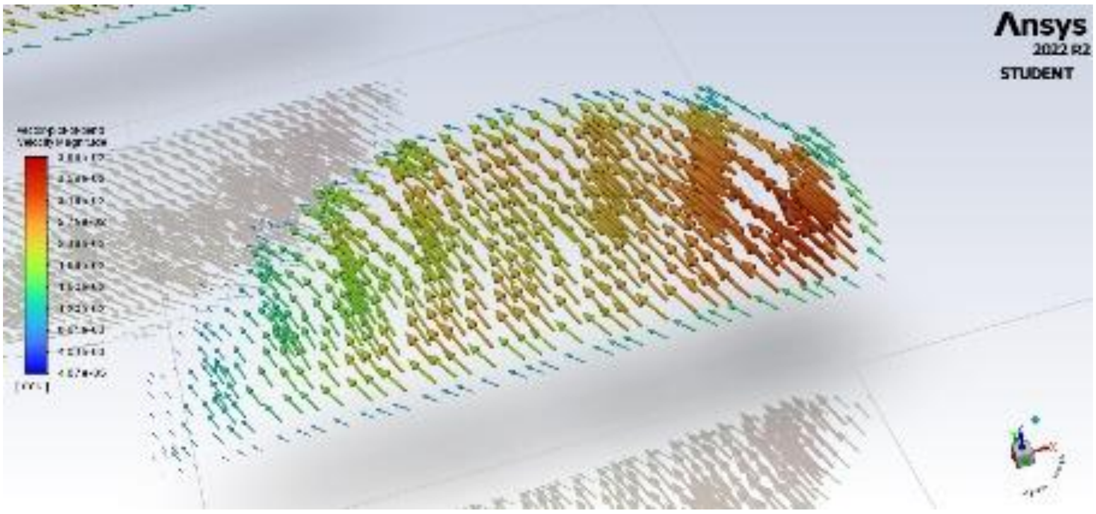

Figure 16: Velocity at the bend (Re = 1000)

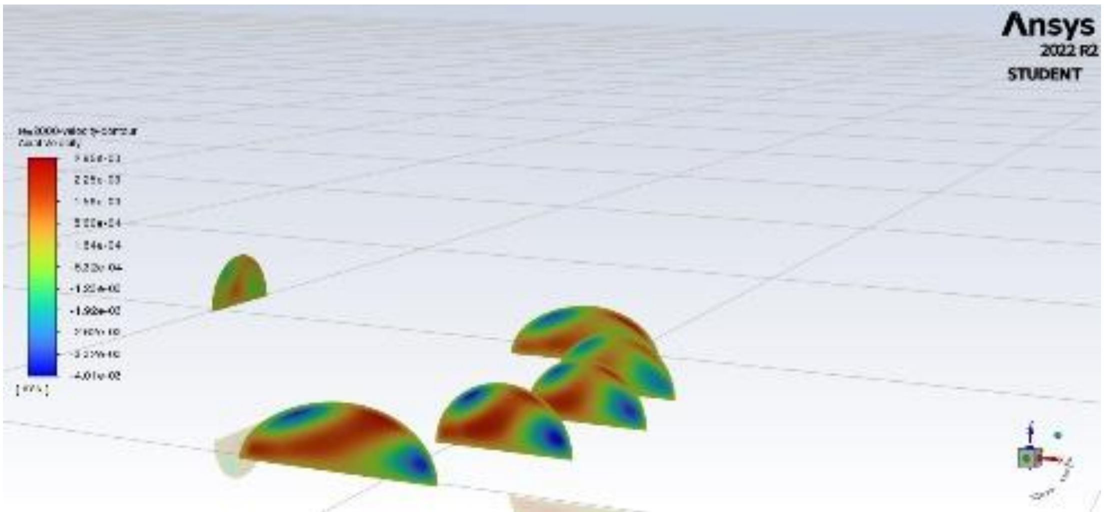

Figure 17: Contour plot of Velocity at the bend (Re = 2000)

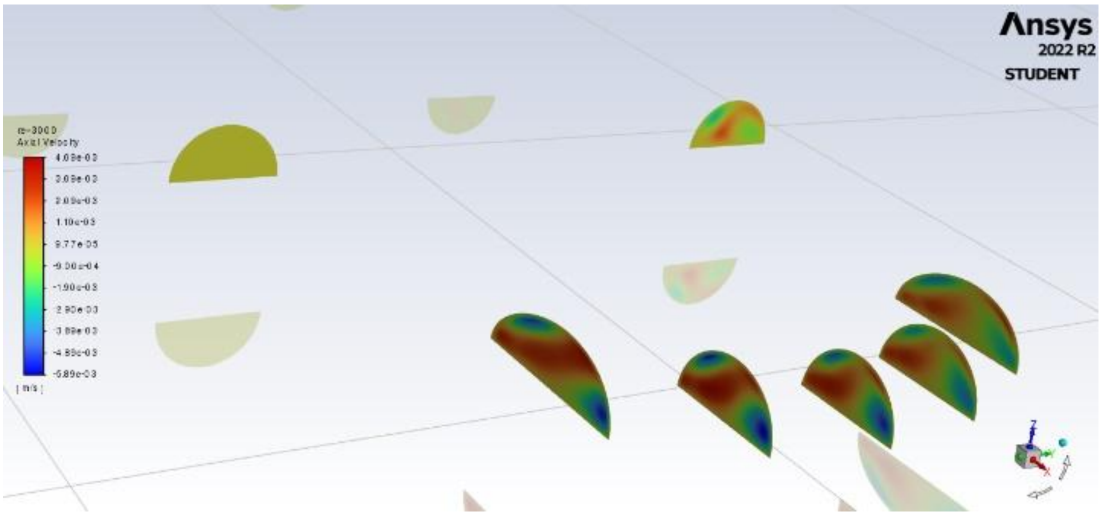

Figure 18: Contour plot of velocity at bend (Re = 3000)

From observation alone it is clear that the as the Reynolds number is increased the measured parameters, velocity and pressure, also increase. This rise in Reynolds number is attributed to the increase in velocity from 0.02 m/s to 0.06 m/s with respect to Re = 1000 to Re = 3000. The rise in pressure is attributed to the centrifugal forces that occur due to the bend in the pipe coupled with the increase velocity. As the fluid particles travel around the bend the inertial forces increase due to the increased velocity. The table below summarises this.

	Re = 1000		Re=2000		Re=3000	
Location	Velocity	Pressure	Velocity	Pressure	Velocity	Pressure
Above midpoint before bend exit	0.013	1.158	0.036	4.207	0.056	6.841
After bend	0.012	0.897	0.034	3.436	0.055	6.931
After bend 5D before exit	0.007	0.175	0.029	0.773	0.046	5.567
Below midpoint 45°angle	0.016	1.318	0.033	4.627	0.052	1.246
Inlet 5D	0.007	2.118	0.028	7.844	0.042	12.745
Inlet before bend diameter	0.007	1.484	0.029	5.219	0.046	8.498
Midpoint	0.007	1.234	0.034	4.344	0.055	7.072

Table 0-2

This rise in turbulence results in secondary flows being developed as seen in figures 14 and 15. The secondary flows ae seen due to the difference in colours in the semi circles, the red equates to higher speed and conversely the blue is opposite. The two blue sections and the swirling pattern in the semi circles depict a vortex type flow known as Dean vortices (*Equation 1 above*). The dean number is a dimensionless quantity that depicts dean vortices that arise due to curved geometries. According to Kalpakli (Kalpakli, 2012), secondary flows became more dominant as the dean number increases. Figures 14 and 15 show these secondary flows in action, these came about due to the bend in the pipe. The secondary flows are visible due to the pressure drop on either side in the semi circles. This result is supported by many literatures that have investigated dean vortices such as Devakar and Smarth.

The relationship between the Dean number and the velocity is investigated below

Figure 19: Model of the Curved Pipe (U-Pipe) used

Figure 20 (and table 0-3) below show a clear linear relationship between the Dean number and the max velocities recorded. This relationship can be attributed to the fact that the Reynolds number is shown in equation 1 and rearranging it, would show that the Dean number has a linear relationship to velocity.

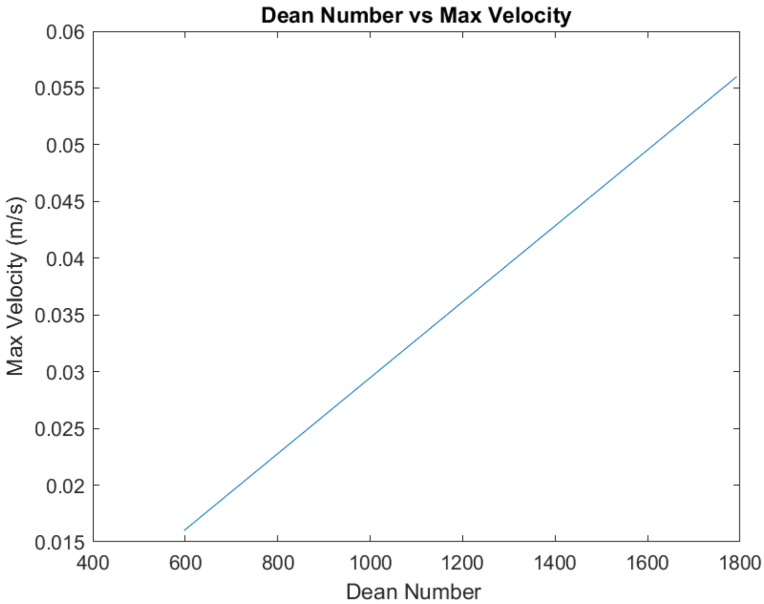

Figure 20: Dean Number vs Max Velocity

Test	Max Velocity	Reynolds Number	Dean Number
1	0.016	1000	597.61
2	0.036	2000	1195.23
3	0.056	3000	1792.84

Table 0-3

B.3) Pressure Surge

When a valve is suddenly closed pressure changes occur through out the pipe. These changes are not instant but are propagated through by pressure waves. In some instances, the water hammer phenomenon may also occur due to the rapid decrease in velocity (consequently, a rapid rise in pressure). The vibrations caused by this pressure wave can result in significant damage to the pipe, in some instances. This water hammer pressure, P_H, is equal:

$$\Delta P = \rho c u \quad c = speed\ wave\ propogates\ back$$

$$For\ a\ rigid\ pipe, c = \sqrt{\frac{B}{\rho}}$$

B = Bulk Modulus of Elasticity (Stress/Volumetric Strain)

However, for this pipe, the bending nature of the pipe means it is not rigid and therefore tensile force per length would be considered. In this case the tensile force is equal to the water hammer pressure's force.

$$(Hoop\ Stress\ in\ pipe\ wall)\sigma = \frac{p_H D}{2a} \quad a = thickness\ of\ pipe$$

Reusing the equation for *c,* Bulk Modulus B would be replaced (for non-rigid pipes) by.

$$B' = \frac{1}{\frac{1}{B} + \frac{D}{aE}}$$

If the pressure surge was equal to 50 Pa, then the hoop stress in the pipe wall would not be compromised because the Hoop stress, σ would be 250 Nm^{-2}. Assuming the wave takes 0.5 seconds to propagate then $c = \frac{l}{t} = \frac{1.1}{0.5} = 2.2$ m/s.

Assuming the pipe is a stainless-steel pipe, and has a thickness of 5mm, the Bulk modulus would be 163 $x10^9$Pa (The Engineering ToolBox, n.d.), the Young Modulus is approximately 200GPa. This would result in the bulk modulus (for non-rigid pipes).

$$B' = 1.781\ x10^{10}\ Pa$$

B.4) Post processing

Throughout this task, default settings provided by Anys were used due to their simplistic nature and the fact that they would be adhere to most engineers as they would apply to most engineering problems. The default vector plots provided by Ansys give great insight as to how the particles within the fluid move with respect to their surroundings i.e no-slip conditions at the pipe walls. However, they do not provide full description for other parameters. The contour plots were tailored to provide a clear insight into the pressure changes around the curved geometry, the detail they provided was significant and hence used not only for pressure but for velocity as well. Infact, without the contour plots, (commonly used for pressure) the secondary Dean vortices caused by turbulent flows would be much more difficult to spot by using the vector plots. Their useful ness was seen in figures 14 and 15. Furthermore, researchers such as Devakar & Smarth alos used velocity contour plots to convey their data. Other type of analysis provided by Ansys include particle tracking, in which particles are tracked through the gemoetry and changes made to the particle due to the geometry i.e. increases in temperature mass , viscosities etc.

Ansys default setting provide a relatively in-expensive method to analyse fluid problems, the alternative to softares such as Ansys would be experimental procedures which would have to be repeated multiple times to ensure reliability of results, however experimentation is a costly endeavour and in some cases i.e. dealing with non-newtonion fluid, or difficult geometries would make the task more difficut as experimentation would involve either involve intrusive systems or non- intrusive methods like LDV (Molki, et al., 2013), which would require funding.

Conclusion

Different Types of Investigations

Throughout this paper many research papers have been presented each with their own form of analysis ranging from analytical to experimental to numerical. Each type of study provides its own advantages and consequently has their disadvantages as well.
For example the investigation by Cieslicki & Piechna covers numeical anslysis, through which they use to generate CFD simiulation, resulting in 'ideal' results. However upon experimentation theses results do not exactly match the simulation.

Simplifications during Simulation

Computation of such situations can be incredibily difficult and hence adequate simplications need be used to reduce computation time. These include limiting the changing i.e. investigating for a small range of Reynolds number addittionally during the construction f the geometries, care should be taken to not analyse the whole shape when a part of it can be used. In cicular cross sections, symmetry would allow to analyse fluid through only a quarter of a pipe instead of the the whole circular pipe. This would not compromise any results

Further Reading

Ansys powerful system can be used to understand and simulate many fluid dynamic problems. It can be used in many situation for example the analysis of 'dirty wind' in Formula 1 to fluid over a swimmer's head to the analysis of the wake from jets. The analysis of laminar and turbulent jet wakes was investigated by ANSYS. It detailed how the wake differ and how turbulent jet wake spread faster compared to their counter part due to the mixing in the shear layer. Furthermore, an investigation into the velocity of the wakes was examined and showed how the turbulent wake decays at a significant rate relative to the laminar.

References

Cieslicki, K. & Piechna, A., 2012. Can the Dean number Alone Characterize Flow Similarity in Differently Bent Tubes?. *Journal of Fluids Engineering*.

Devakar, M. & Smarth, M., 2023. Analytical study on the flow of immiscible fluids through a curved pipe. *Journal of the Brazilian Society of Mechanical Sciences and Engineering,* 45(2).

Douglas, J. F., Gasiorek, J. M. & Swaffield, J. A., 2011. *Fluid Mechanics*. 6th ed. s.l.:Prentice Hall.

Dutta, P. & Nandi, N., 2016. Effect of bend curvature on velocity & pressure distribution from straight to a 90° pipe bend - A Numerical Study. *REST Journal on Emerging trends in Modelling and Manufacturing,* Volume 2, pp. 103-108.

Kalpakli, A., 2012. Experimental study of turbulent flows through.

Kalpakli, A., 2012. Experimental study of turbulent flows through. p. 4.

Molki, A., Khezzar, L. & Goharzadeh, A., 2013. Measurement of fluid velocity development in laminar pipe flow using laser Doppler velocimetry. *Europena Journal of Physics,* Issue 34, pp. 1127-1134.

The Engineering ToolBox, n.d. *Metals and Alloys- Bulk Modulus Elasticity,* s.l.: s.n.

Appendix

1) To calculate the values for figure 9: Velocity Profile (For Pipe In The Molki Paper)
   a. Python code to iterate through equation 1:

C: > Users > dani_ > OneDrive > Desktop > Programming > Python > Python(Uni) > MATHS CW PT2.py > ...

```python
r = [-12, -11, -10, -9, -8, -7, -6, -5, -4, -3, -2, -1, 0, 1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12]
D = 24
umax = 38
u = []
for i in r:
    u2 = umax *(1- ((2*i)/D)**2)
    u.append(u2)
print(u)

```

   b. Matlab code to plot figure 9:

Editor - C:\Users\dani_\OneDrive\Desktop\Engineering\Engineering Year 3\Control & Mech\MATHS_CW_PT_2_Graph.m

Avionics_Tutorial_Version_of_CW.m | Spring_Mass_Input.m | Plot_Results.m | Dynamics_CW.m | MATHS_CW_PT_2_Graph.m

```matlab
r = [-12 -11 -10 -9 -8 -7 -6 -5 -4 -3 -2 -1 0 1 2 3 4 5 6 7 8 9 10 11 12];
D = 24;
umax = 38;
u = [0.0, 6.069444444444448, 11.611111111111107, 16.625, 21.11111111111111, 25.06944444444444, 28.5, 31.4027777777777

plot (u,r)

```

2) MATLAB code to plot the graph for convergence vs continuity (figure 6)

```matlab
cont = [5.53e-7 , 5.354e-5, 4.500-5];
conv = [10e-6, 10e-4, 10e-2];

plot (conv,cont)
```