SIMLUATION OF FATIGUE FACTOR AND FLUID FLOW INSIDE A FLANGE

ANSYS

Dinesh Poluru Baskar
dineshpb0812@gmail.com

Table of Contents

Figures

Introduction

The project task is to simulate a flange with given dimension using ANSYS and also to find the Goodman's Fatigue safety factor. This project is divided widely into two parts, the first part gives a detailed report of the flange creation, meshing and post processing while the second task is to calculate the static pressure and fluid flow inside a static valve.

Task 3:

Simulation of a flange and Non-linear analysis in statics

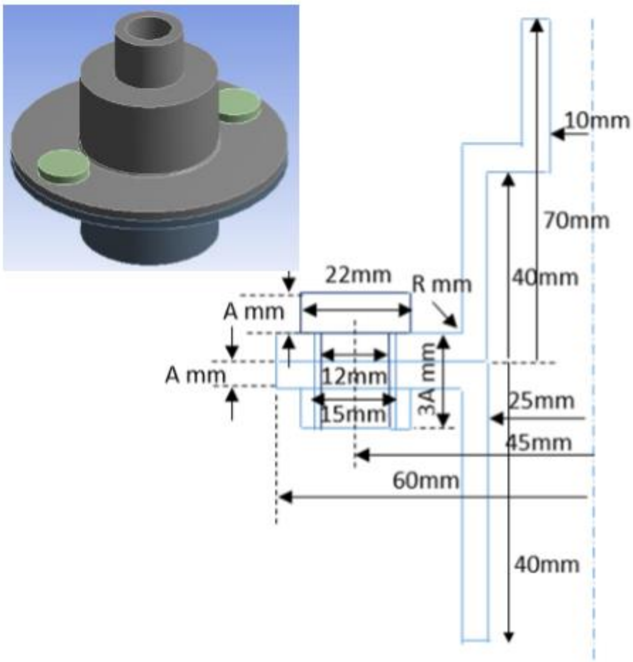
Prestress of bolt (16KN)

Fluid pressure 5 MPa

Calculate the maximum stress in the flange.

Special tasks:

- Provide a good structure mesh for the simulation to get more exact results more quickly.
- Compare the simulation time and results between direct and iterative solver of a system of linear equations.
- Find the Goodman's fatigue safety factor.



Method

Ansys provides a way to create any kind of geometry and run the simulation according to the dimensions and constraints provided. In this report I have created a flange with the given dimension and meshed it to run the simulation in order to find the results.

A good mesh is "A Beautiful Mesh" in which meshes are created to find the flow of fluid through the flange at that certain point. When the mesh is really fine then the results will be more accurate, however meshes can be made in many types and some of the meshes used are tetrahedral and square meshes and the results are compared according to that.

Two methods are used to run the simulation namely:

- Direct method
- Iterative method

These two methods are used to run the simulation and the values are compared between each other and also simulated the normal (or) automated mesh known as tetrahedral mesh. These are simulated using Ansys and these results will be discussed in this report.

Direct Method

In this method meshes are given to every part of the body and the simulation is run to find out the results.

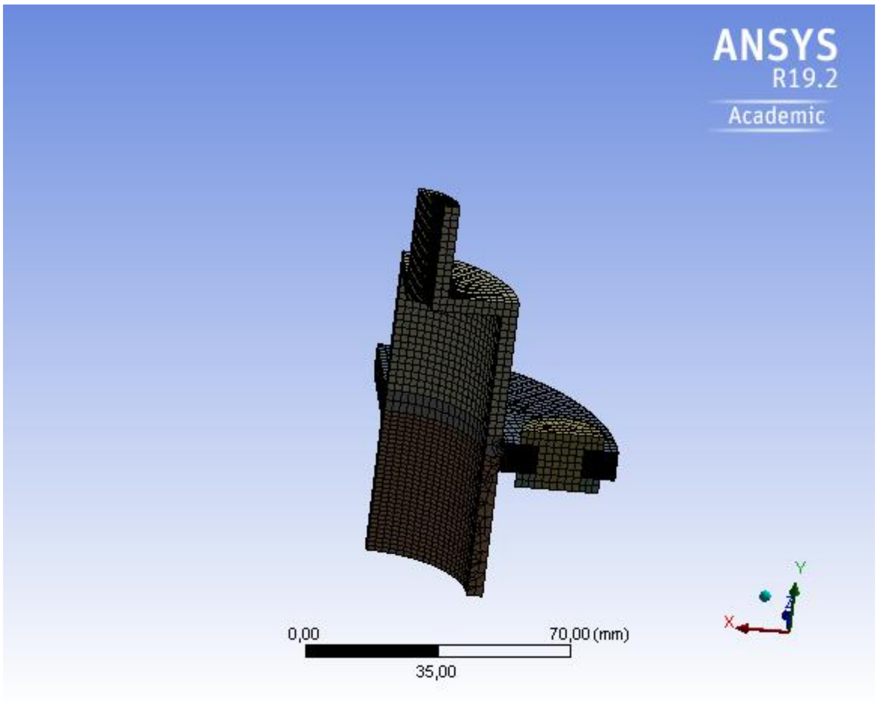
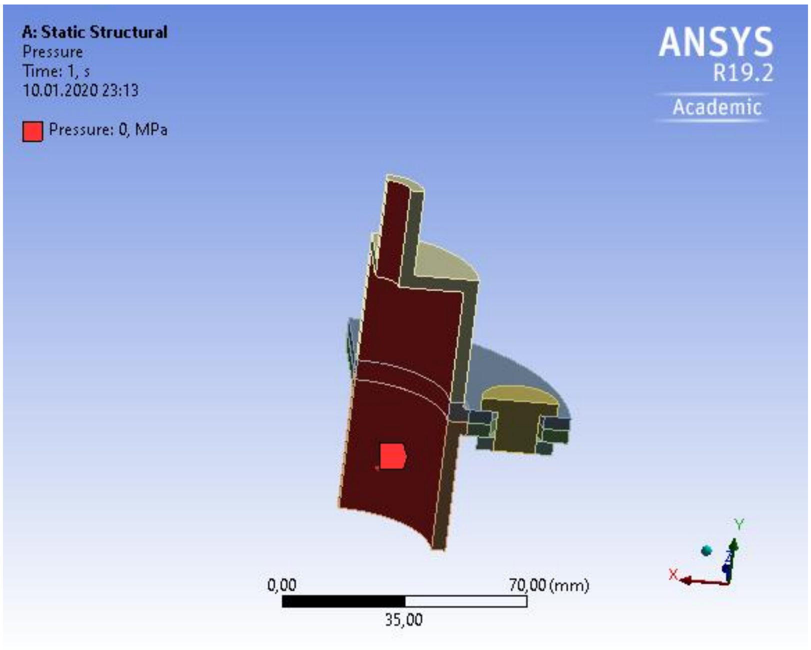


Figure 1: Shows the mesh for the flange.

Pressure and Bolt:

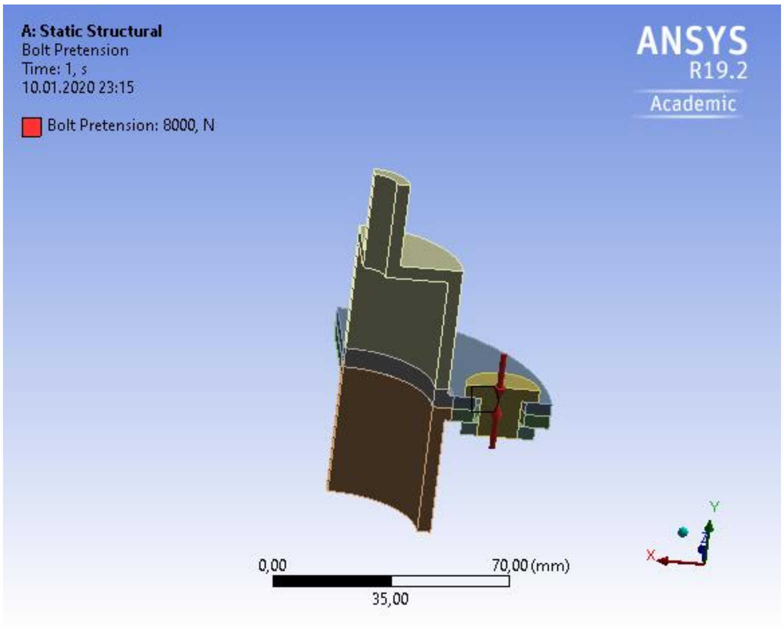
In this method I have added pressure for the fluid and the bolt pretension to show the results and compare it with the iterative method.



	Steps	Time [s]	Pressure [MPa]
1	1	0,	0,
2	1	1,	0,
3	2	2,	5,
*			

Figure 2: Shows the pressure in the flange.

Bolt:

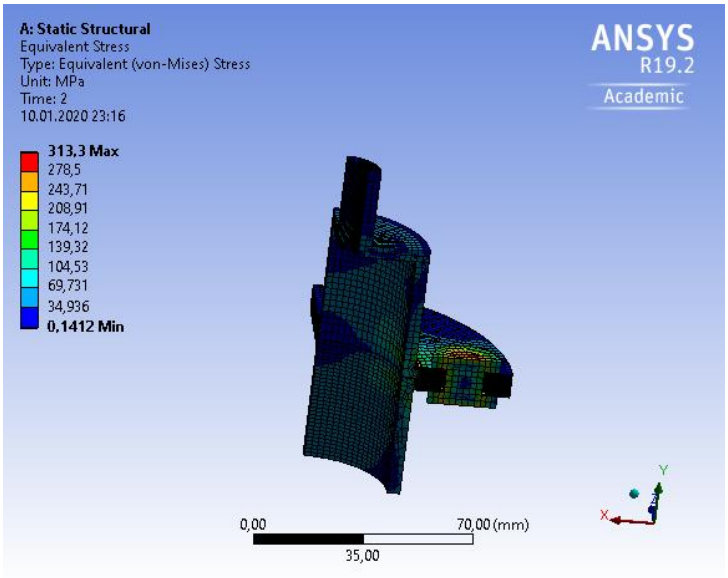


	Steps	Define By	Preload [N]	Preadjustment [mm]	Increment [mm]
1	1,	Load	8000,	N/A	N/A
2	2,	Lock	N/A	N/A	N/A
*					

Figure 3: Shows the bolt pretension.

Final Result:

In this final result I have run the simulation for 37 minutes and the results which I got here are provided, in which it shows that the maximum stress is around the edges of the flange because of the shape where the fluid flow will become laminar to turbulent at that point where the results will blow up to provide a non-linear characteristic.



Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done
MAPDL Elapsed Time	37 m 2 s
MAPDL Memory Used	8,6035 GB
MAPDL Result File Size	290,94 MB
Post Processing	
Beam Section Results	No
On Demand Stress/Strain	No

Figure 4: Shows the final result of the direct method simulation.

Now that we have seen the simulation results which run from time step 0 to 2 and the line graph which represents the time step are shown below.

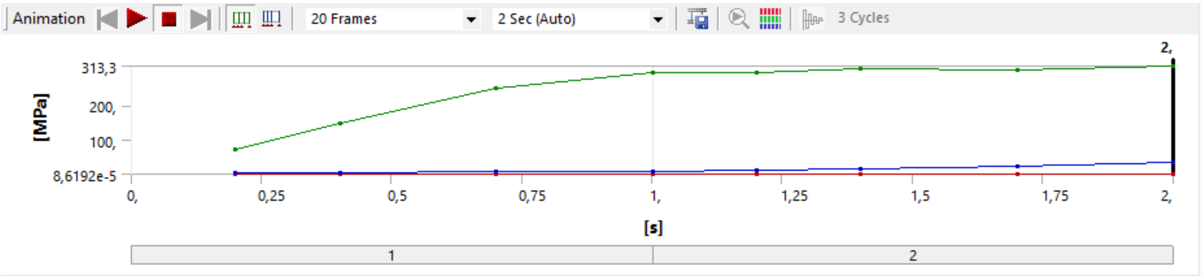


Figure 5: Shows the graph for the running of the simulation.

	Time [s]	Minimum [MPa]	Maximum [MPa]	Average [MPa]
1	0,2	2,1447e-004	73,349	1,9178
2	0,4	1,3955e-004	146,66	3,8268
3	0,7	8,6192e-005	249,37	6,6431
4	1,	8,7653e-005	294,64	9,3987
5	1,2	0,11909	294,4	12,613
6	1,4	1,5045e-002	307,62	16,398
7	1,7	7,9426e-002	300,78	24,404
8	2,	0,1412	313,3	33,377

Figure 6: Shows the min, max and average data for the simulation.

Fatigue safety factor

Fatigue safety factor is applied by clicking the insert from the solution and fatigue safety factor and from that menu Goodman's factor is selected from the drop-down list of the solution solver and the simulation is run to find out the results.

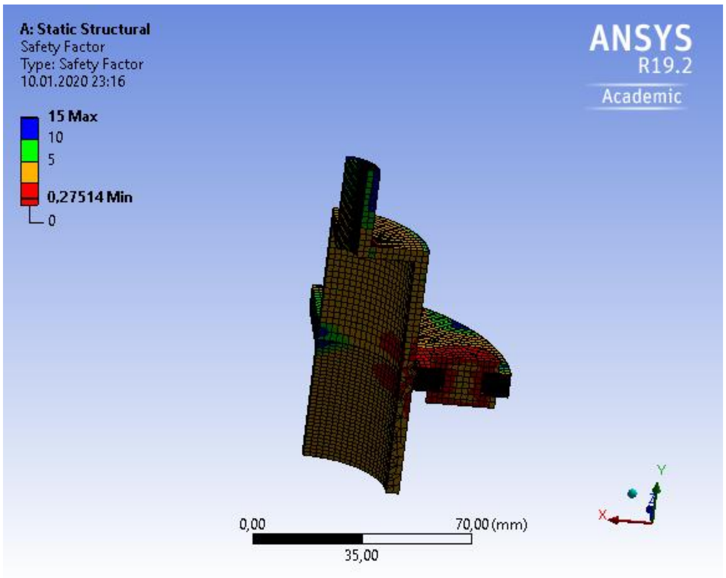


Figure 7: Shows the fatigue Goodman's safety factor.

Now that we have looked about the results which we got by running the simulation and now we will look into the force convergence and displacement convergence curve of the simulation which is run from 0 to 2-time step.

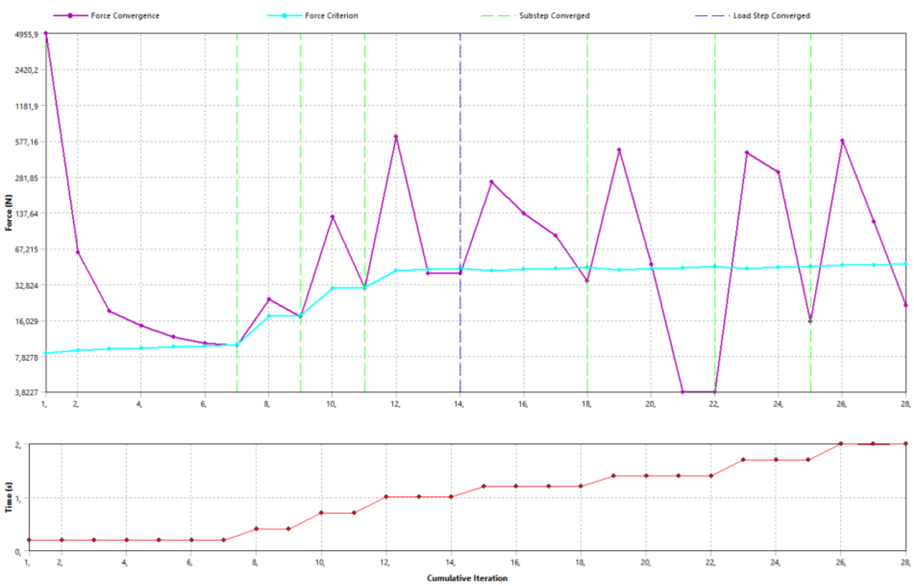


Figure 8: Shows the force convergence.

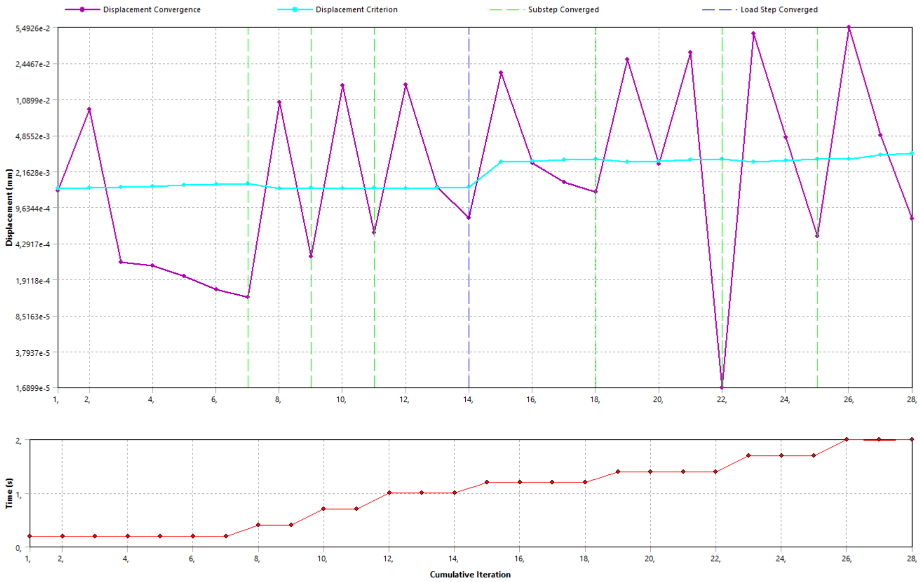
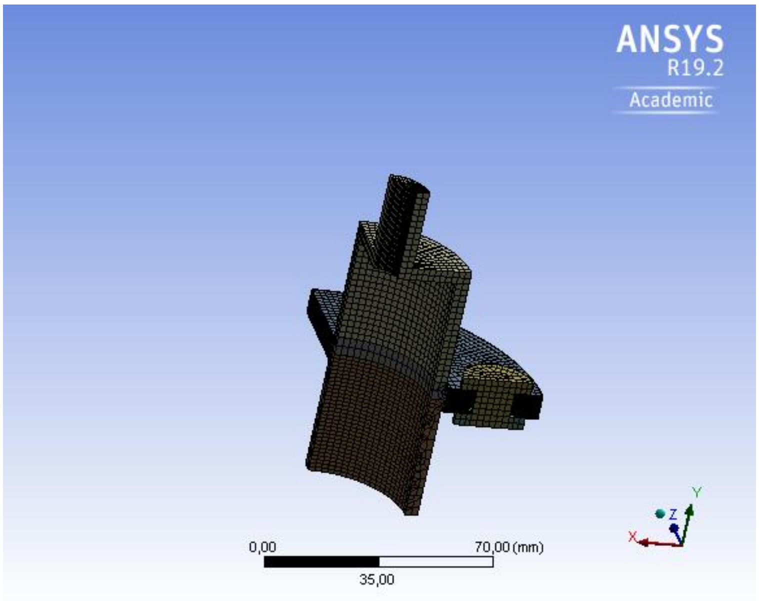


Figure 9: Shows the displacement convergence graph.

Now that we have seen the results of the direct method by using normal meshes which we have given for every part of the flange. Now we take a look at the iterative method which is also run by the same method as direct method expect by changing the solver type from direct method to the iterative method.

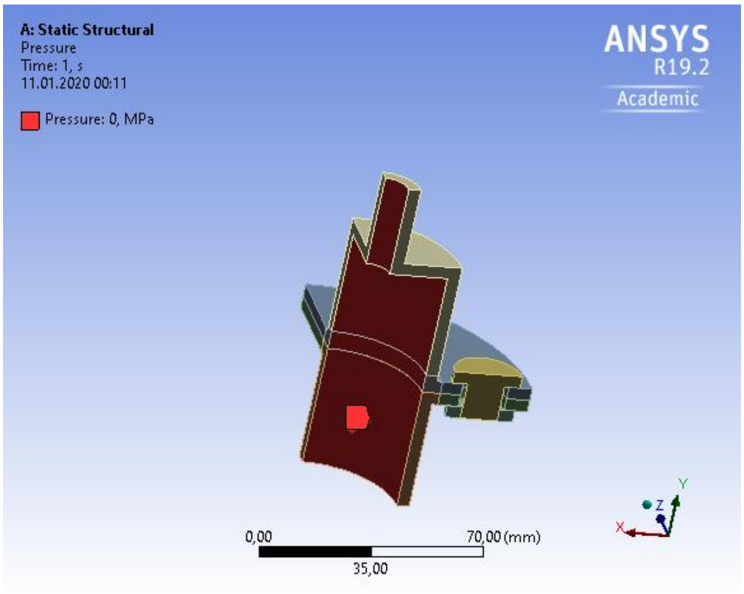
Iterative Method

This method is done by using iterative method solver type in the solution tab and the simulation is run to find out the results which are discussed below.



Pressure and Bolt:

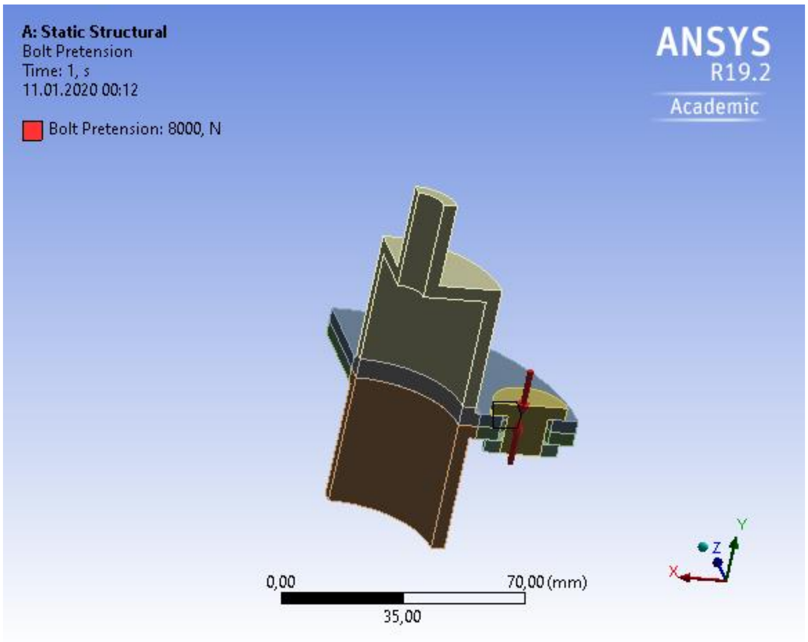
The pressure is added to the flange and for the bolt pretension too so as to run the simulation and find out the results for the simulation.



	Steps	Time [s]	Pressure [MPa]
1	1	0,	0,
2	1	1,	0,
3	2	2,	5,
*			

Figure 10: Shows the pressure data in the flange.

Bolt:

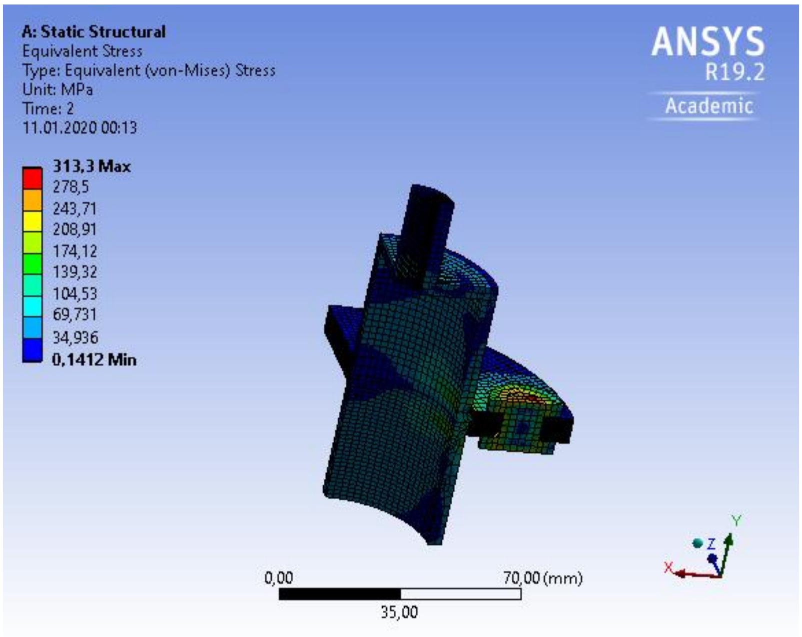


	Steps	Define By	Preload [N]	Preadjustment [mm]	Increment [mm]
1	1,	Load	8000,	N/A	N/A
2	2,	Lock	N/A	N/A	N/A
*					

Figure 11: Shows the bolt pretension.

Final Result:

The final result of this iterative method shows that the simulation time is around 31 minutes which is less when compared with the direct method. And the results for the direct method and iterative method is almost same.



Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done
MAPDL Elapsed Time	31 m 46 s
MAPDL Memory Used	1008, MB
MAPDL Result File Size	290,94 MB
Post Processing	
Beam Section Results	No
On Demand Stress/Strain	No

Figure 12: Shows the final result for the iterative simulation.

Now we take a look at the result data and graphs.

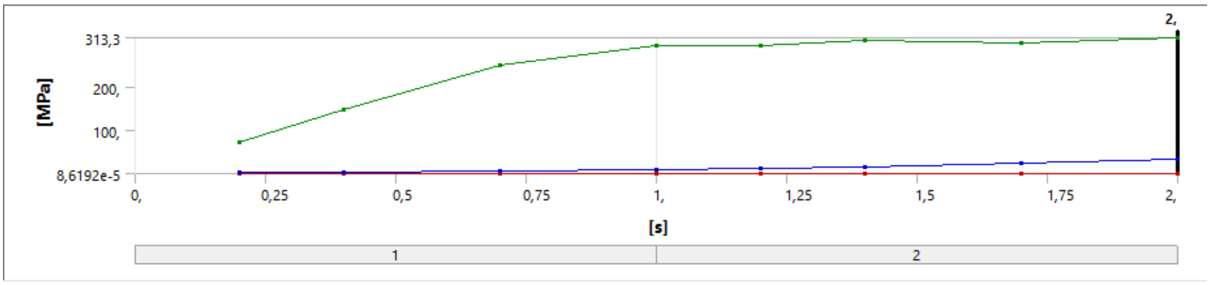


Figure 13: Shows the graphs for the iterative method solver.

	Time [s]	Minimum [MPa]	Maximum [MPa]	Average [MPa]
1	0,2	2,1447e-004	73,349	1,9178
2	0,4	1,3955e-004	146,66	3,8268
3	0,7	8,6192e-005	249,37	6,6431
4	1,	8,7653e-005	294,64	9,3987
5	1,2	0,11909	294,4	12,613
6	1,4	1,5045e-002	307,62	16,398
7	1,7	7,9426e-002	300,78	24,404
8	2,	0,1412	313,3	33,377

Figure 14: Shows the min, max and average data.

Fatigue safety factor

The fatigue safety factor is provided by using the fatigue solver in the solution and providing the Goodman's factor of safety to run the simulation.

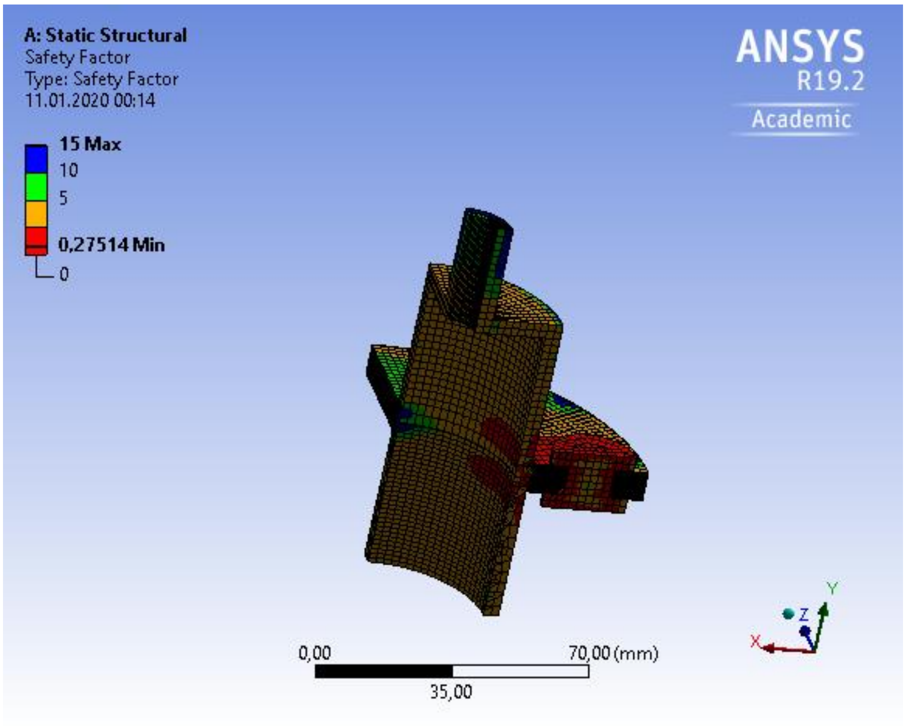


Figure 15: Shows the fatigue safety factor for iterative method.

Now that we have looked about the results and fatigue safety factor for iterative method and now, we look at the force and displacement convergence graphs for the same.

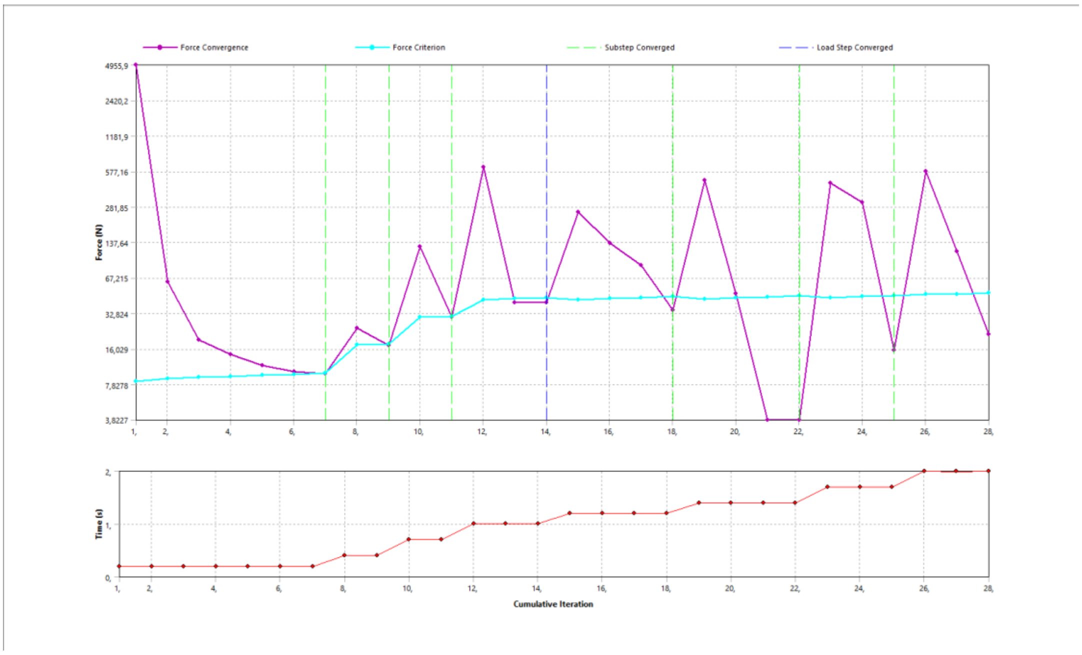


Figure 16: Shows the force convergence.

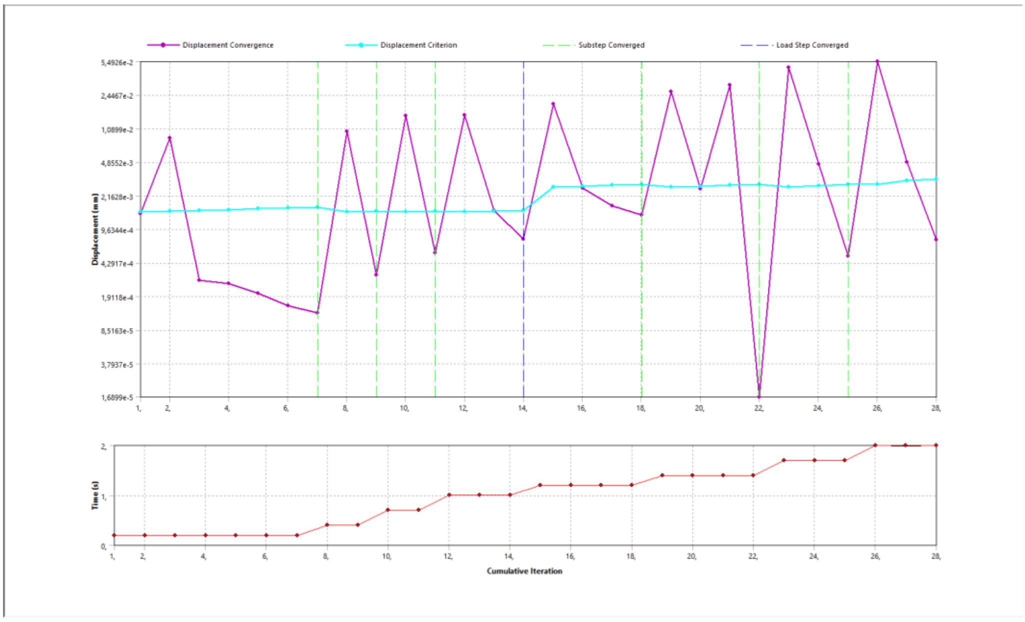
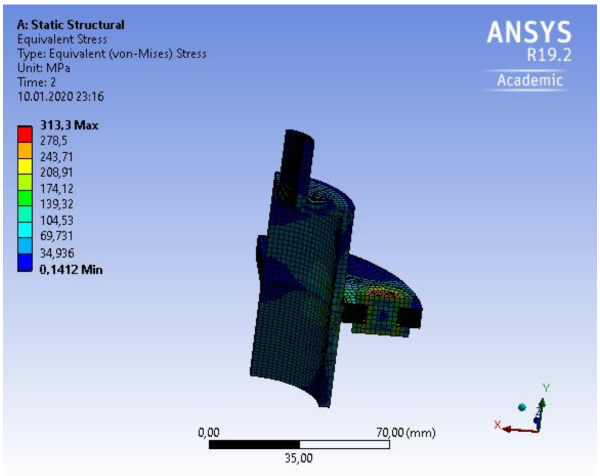
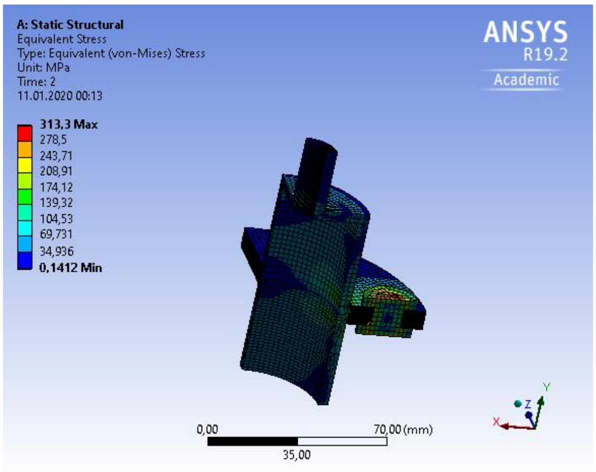


Figure 17: Shows the displacement convergence.

Now that we have looked about the direct method and iterative method of simulation. Now we compare the results of the direct and iterative method.

Comparison between the results:





The above pictures show the comparison between the direct method and iterative method of simulation. As we can see clearly the results which I simulated are almost equal to each other, this shows that these methods do not have any influence in the simulation. But there is a slight change in the simulation time which is shown below.

Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done
MAPDL Elapsed Time	37 m 2 s
MAPDL Memory Used	8,6035 GB
MAPDL Result File Size	290,94 MB
Post Processing	
Beam Section Results	No
On Demand Stress/Strain	No

Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done
MAPDL Elapsed Time	31 m 46 s
MAPDL Memory Used	1008, MB
MAPDL Result File Size	290,94 MB
Post Processing	
Beam Section Results	No
On Demand Stress/Strain	No

The above data shows the simulation elapsed time for the direct and iterative method respectively. This shows that iterative method takes less time to run the simulation because it works with Jacobian iterative method.

Tetrahedral Method

In this method the same procedure of simulation is done but with only exception is that the mesh grid is changed from normal method to tetrahedral mesh type in which the mesh grid is changed from square to triangle shape. This will have an influence in the final result which we will be looking below.

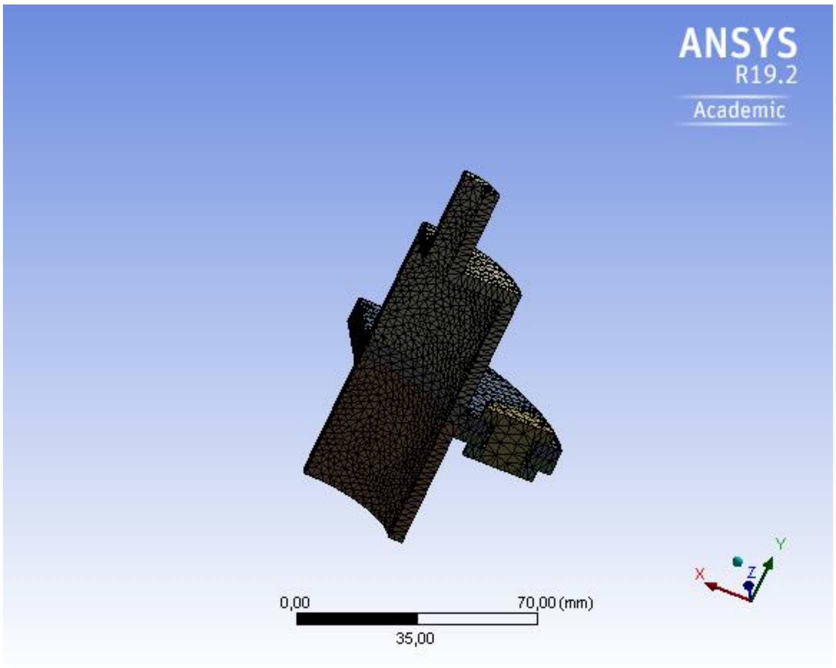
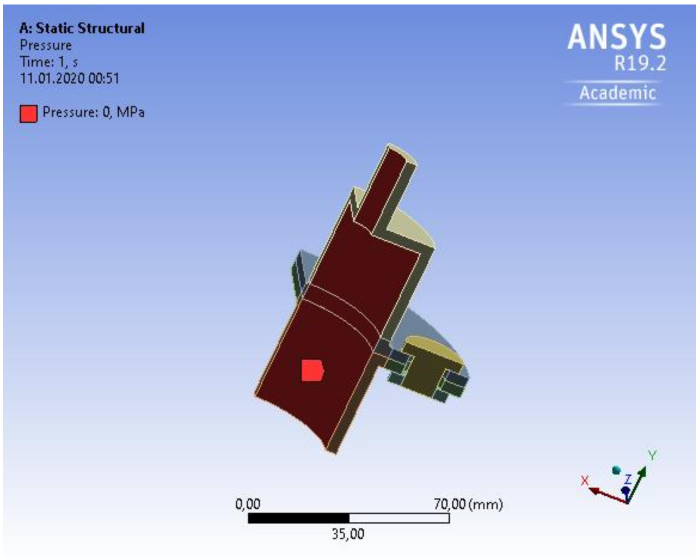


Figure 18: Shows the tetrahedral mesh.

Pressure and Bolt pretension:

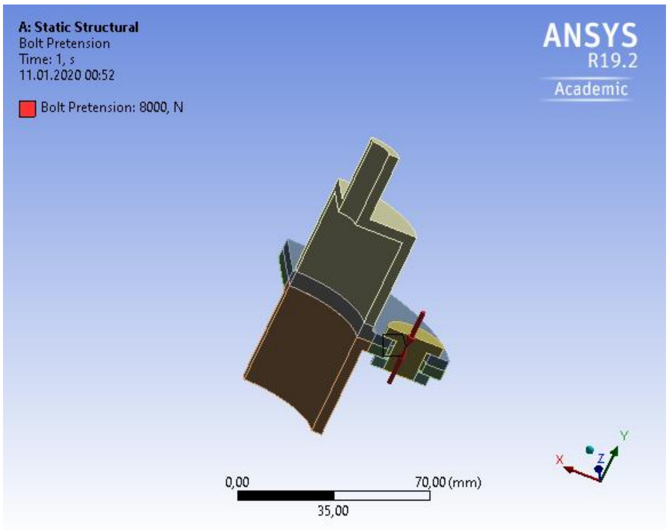
The pressure is added to the flange as we saw in the above method and the simulation is run to find out the result.



	Steps	Time [s]	Pressure [MPa]
1	1	0,	0,
2	1	1,	0,
3	2	2,	5,
*			

Figure 19: Shows the pressure of tetrahedral mesh flange.

Bolt:

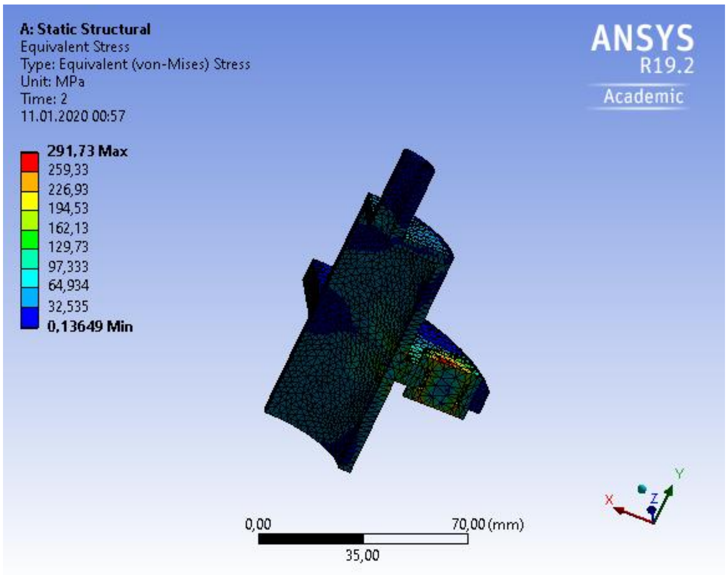


	Steps	Define By	Preload [N]	Preadjustment [mm]	Increment [mm]
1	1,	Load	8000,	N/A	N/A
2	2,	Lock	N/A	N/A	N/A
*					

Figure 20: Shows the bolt pretension.

Final Result:

The simulation for this is also run by the same method as in the case for normal and iterative method. The final results shown here shows some change in the result because of the mesh we have applied for the simulation.



Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,
Information	
Status	Done
MAPDL Elapsed Time	4 m 39 s
MAPDL Memory Used	219, MB
MAPDL Result File Size	81,875 MB
Post Processing	
Beam Section Results	No
On Demand Stress/Strain	No

Figure 21: Shows the result of tetrahedral mesh.

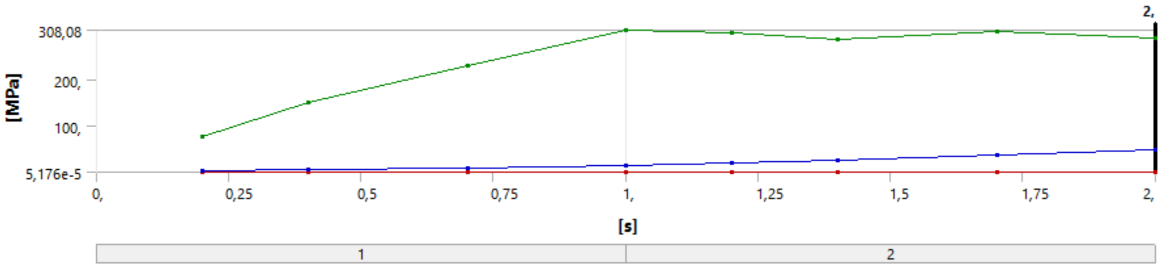


Figure 22: Shows the graph for the tetrahedral mesh.

	Time [s]	Minimum [MPa]	Maximum [MPa]	Average [MPa]
1	0,2	5,176e-005	76,275	2,6916
2	0,4	1,3192e-004	152,56	5,3823
3	0,7	2,6171e-004	231,5	9,3696
4	1,	5,6233e-004	308,08	13,184
5	1,2	0,16105	301,05	19,118
6	1,4	2,4234e-002	287,3	25,939
7	1,7	0,17664	306,21	37,441
8	2,	0,13649	291,73	49,365

Figure 23: Shows min, max and average results.

Fatigue Safety factor

In this method fatigue safety factor is added to the solution and from the solver type Goodman's solver is selected and simulation is run to find out the results.

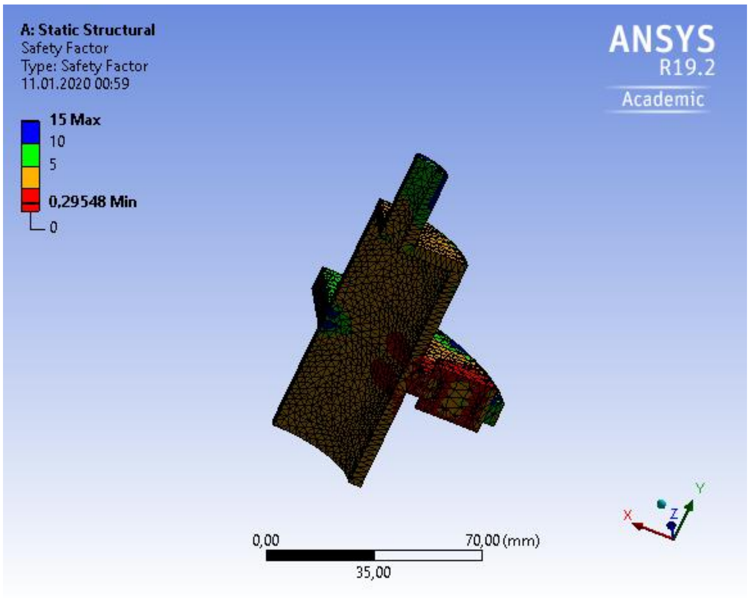


Figure 24: Shows the fatigue safety factor for tetrahedral method.

Conclusion:

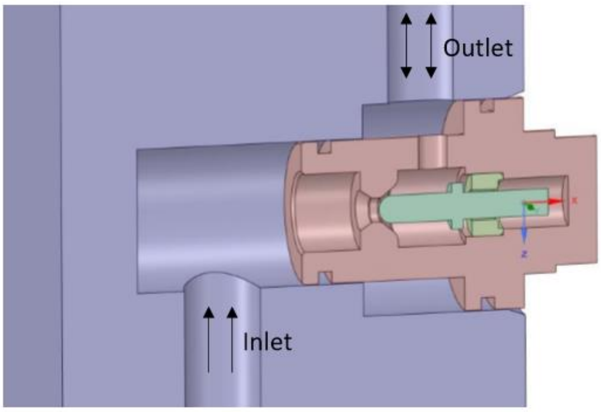
As we can say from the results the direct and iterative method provide almost same results, but with the tetrahedral mesh we are provided with different result which is precisely there is a reduction in the maximum stress. But the simulation elapsed time is reduced drastically in the tetrahedral mesh when compared with direct and iterative method.

Task 2:

Fluid Simulation for a static valve

Boundary conditions: Inlet – Total pressure, Outlet – open, 0 bar

Calculate the flow, static pressure and fluid force for the stroke = 1mm



Method

This task is given in order to find out the fluid flow simulation inside a flange. This simulation shows where the laminar and turbulent flow occurs in a flange so that we can make a precise calculation according to that. The body part is moved to a certain distance in order to allow the fluid to run from inlet to outlet of the flange.

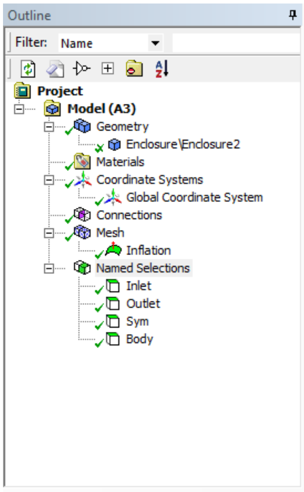
I was provided with my own pressure, temperature, density and kinematic viscosity for the simulation.

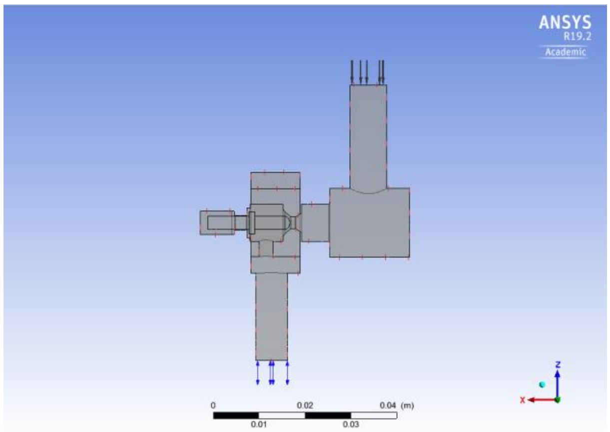
5	0,3	50	50	808,6	2,21E-06

- Total Druck – 0.3 bar
- Diesel Temperature - 50ºc
- Density – 808.6 kg/m^3
- Kinematic Viscosity – 2.21e-06

In materials option I have chosen diesel for the flow of the fluid in the flange and the diesel properties are mentioned above are used in this simulation to find out the result.

Then the inlet, outlet, symmetry and body geometries are selected to start the fluid flow from inlet to outlet of the flange.



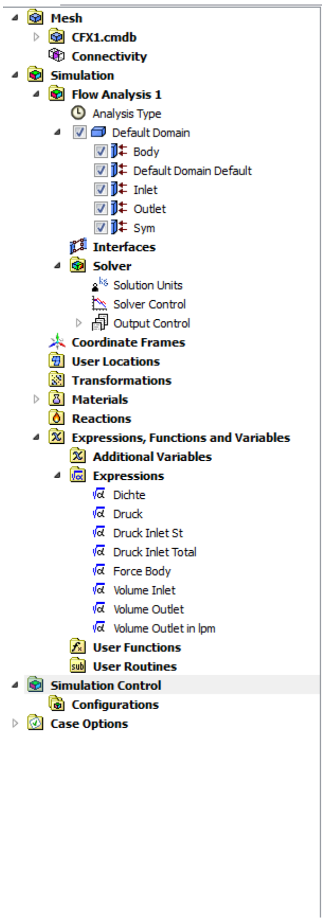


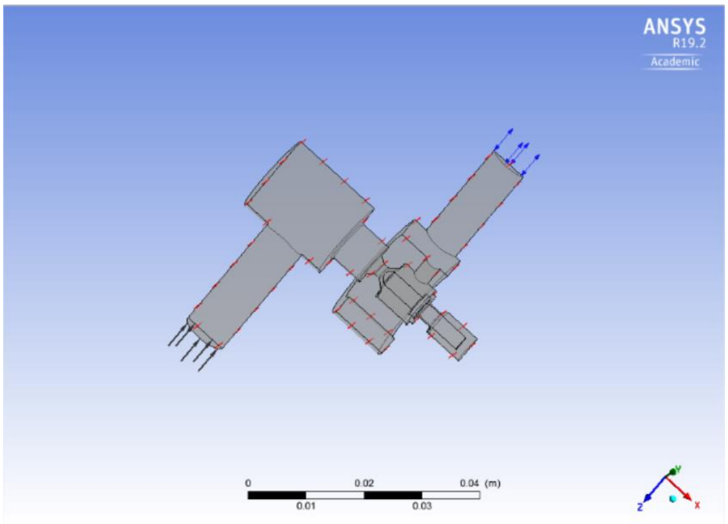
The black lines in the geometry shows the inlet in which the fluid pass through in and the blue lines shows the outlet of the flange.

Then after giving the boundary conditions, the setup is opened to give the parameters and expressions for the simulation to run.

Setup Data

In this method, the setup file is modified for my values like providing expressions, changing solver values and analysis type so as to run the simulation. Below shows the setup type in which the values are changes according to my preference.

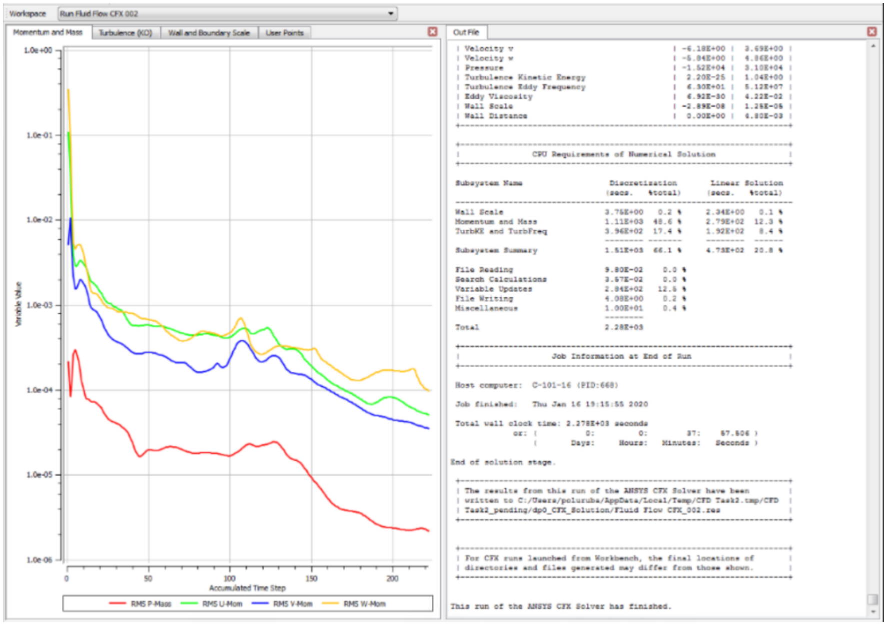




The arrows in the diagram shows the place where the fluid flow takes place. Those expressions are given act as an initial condition for the flow simulation to takes place.

Final Result

The simulation is run from time step of 0-200, as a result the graph which shows the fluid flow inside the flange is shown below.



Now that we have looked for 1e-4-time step and their results. The next task is to run the same simulation with 5e-5 step time to see the result and compare the same.

For this 5e-5, the same method is followed as in the case for 1e-4 expect changing the time step value in the solver output and the results are saved and run for the simulation.

The below pictures show the graph of the final result of the simulation.

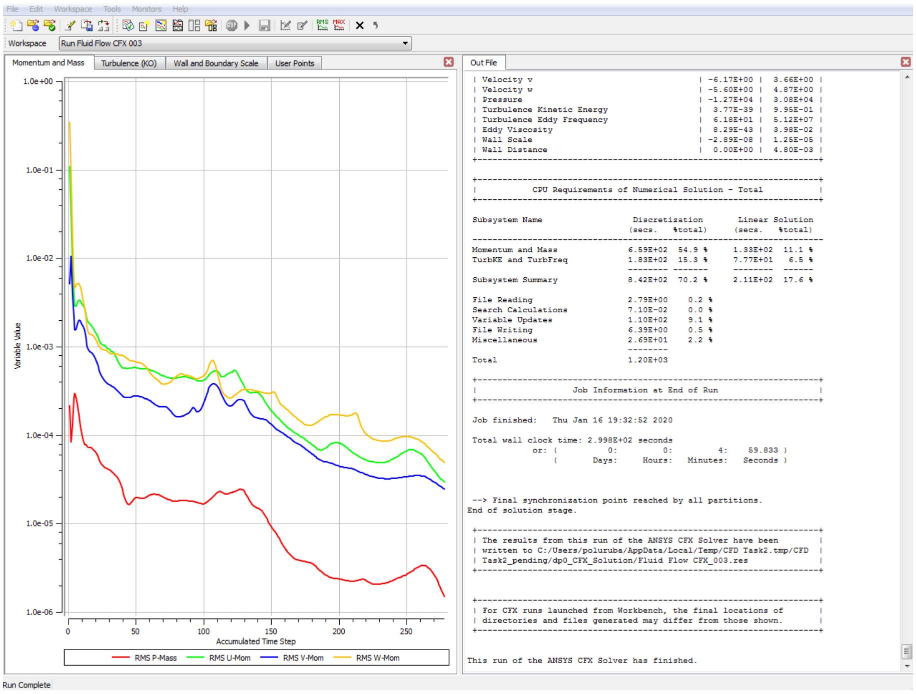
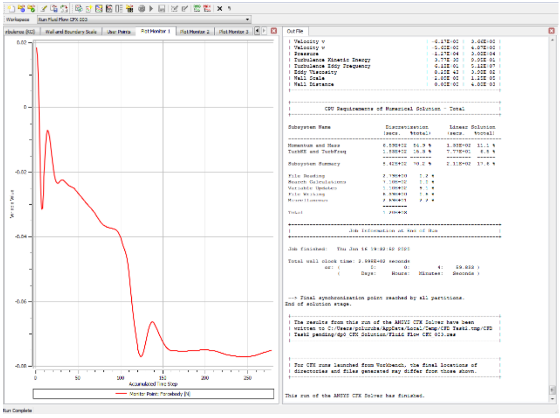


Figure 25: *Shows the final result for 5e-5 step time.*



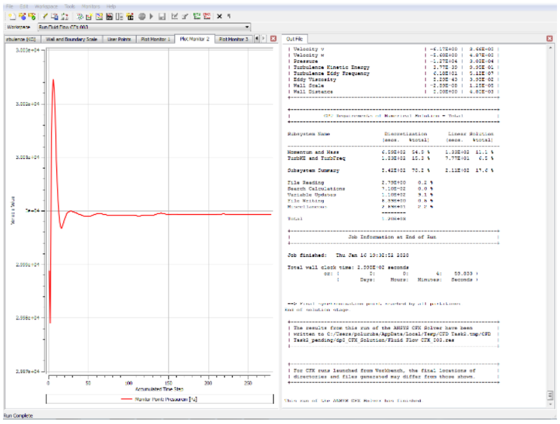


Figure 26: *Shows the plot monitor 1 and 2 for 5e-5*

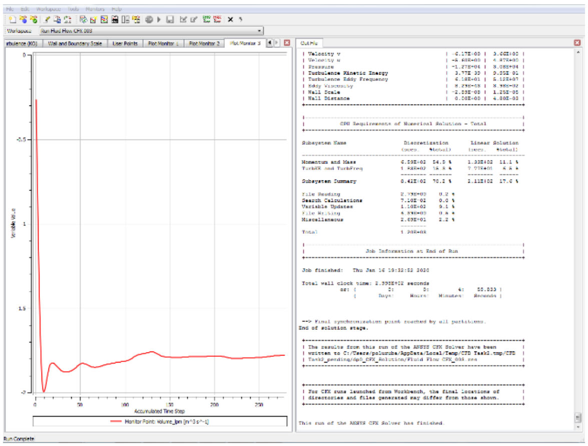
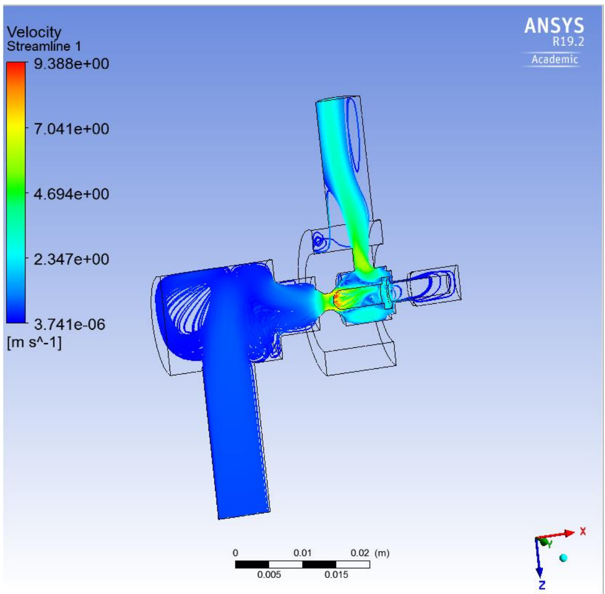
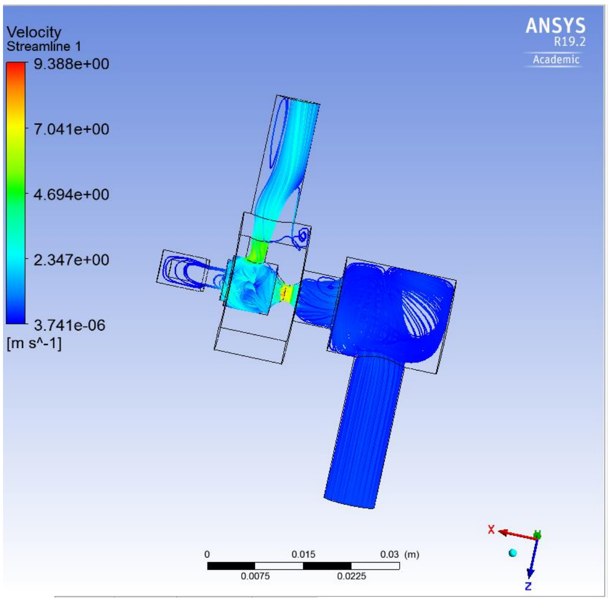


Figure 27: *Shows the plot monitor 3.*

Now that we have seen the graph for the final simulation of flange with 5e-5, now in order to find the real simulation setup, after finishing the simulation the result section is opened and the streamline option is added and the time step is given as 250 as the simulation is run for 250-time step.





The above images show the fluid flow in the flange and see how the laminar flow changes into turbulent as it enters the bend part of the pipe.

Conclusion

The simulation is run successful and the results shows the fluid flow inside the flange from inlet to the outlet as it moves from laminar to turbulent.