

5.0 GENERAL

Computational fluid dynamics (CFD) and Static dynamics is one of the branch of Engineering, Finding numerical solutions of governing equations, using high-speed digital computer. CFD uses numerical methods to solve the fundamental nonlinear differential equations that describe fluid flow, for predefined geometries and boundary conditions. The result is a wealth of predictions for flow velocity, temperature, density and chemical concentrations for region inside the casing, where flow occurs. Static dynamics uses numerical methods to solve static behavior of the material respect to load on the casing.

Computational fluid dynamics (CFD) has become an integral part of the engineering design and analysis in an environment of many companies that need to predict the performance of new designs or processes before they are manufactured or implemented. ANSYS CFX combines advanced solver technology with a modern user interface and an adaptive architecture to make CFD accessible to both designers with general engineering knowledge and fluid dynamics specialists requiring in-depth model control and options. It is used in a vast array of industries to provide detailed insight into equipment and processes that increase efficiency, improve product longevity and optimize processes.

A key advantage of CFD is that it is a virtual modeling technique with powerful visualization capabilities such as flow circulation inside the casing and therefore engineers can evaluate the performance. Static dynamics analysis gives the detail study of material to be used for manufacturing of casing. With different material and thickness of material to be used can be analyzed for cost reduction and safety point of view. Figure: 5.1 shows casing with rib support as per requirement and suction pipe which is overhung. So, after analyzing

the CFD, we can do the static dynamics of the casing. Computational Fluid Dynamics (CFD) provides a qualitative or quantitative prediction of fluid flows inside the casing by means of:

Mathematical modeling (partial differential equations)

Numerical methods (discretization and solution techniques)

Software tools (solvers, pre- and post processing utilities)

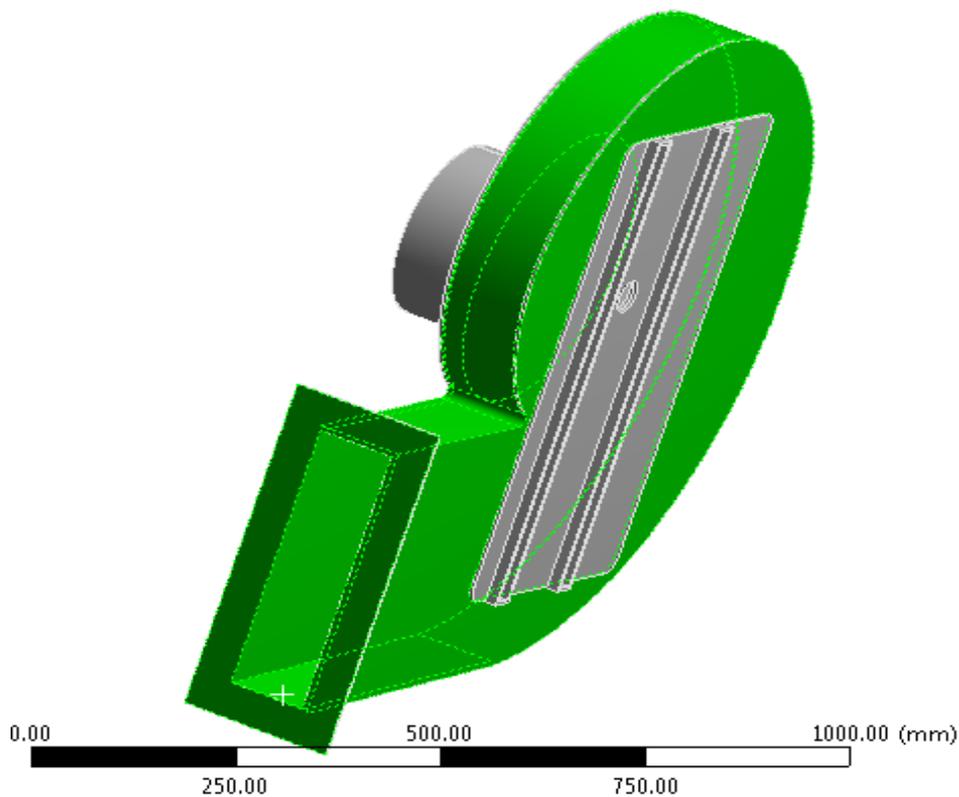


Figure: 5.1

To obtain an approximate solution numerically, we have to use a discretization method which approximates the differential equations by a system of algebraic equations, which can then be solved on a computer. The approximations are applied to small domains in space and/or time so the numerical solution provides results at discrete locations in space and time. Much as the accuracy of experimental data depends on the quality of the tools used, the accuracy of numerical solutions is dependent on the quality of discretization used. CFD is finding its way into process, chemical, civil, and

environmental engineering. Optimization in these areas can produce large savings in equipment and energy costs.

In Blower, Consists of rotating impeller and stationary casing, so there is interfacing between the two domains. To get the approximate analysis of flow in the casing Computational Fluid Dynamics is now an established design tool, helping to reduce design timescales and improve process. To get the detail circulation zone at any point in the casing with clear visualization can be very helpful by CFD. CFD provides a cost-effective and accurate alternative to scale model testing, with variations on the simulation being performed quickly, offering obvious advantages in analysis of flow in volute casing. CFD analysis contains some procedure step to evaluate the analysis. As we have used ANSYS CFX for analysis of flow in volute casing, which follows the procedure as per given below.

5.1 OVERVIEW OF CFD IN BLOWER

ANSYS CFX is a general purpose Computational Fluid Dynamics (CFD) code, combining an advanced solver with powerful pre- and post-processing capabilities. The next-generation physics pre-processor, ANSYS CFX-Pre, allows multiple meshes to be imported, allowing each section of volute casing geometries to use the most appropriate mesh. ANSYS CFX includes the features with an advanced coupled solver to solve all the equation in the casing, which is both reliable and robust. Well define parts of the geometry are defined in the software as per principle. In blower, we have rotating and stationary parts, so to handle the various equations and interface them properly to get the solution and results which are to be presented as per requirement. Ansys CFX solve the problem of Casing of blower in few steps, which require basic knowledge and principle working of blower, this all the information can be put in structure way of the software. All these steps are linked with one another.

5.2 STRUCTURE OF CFD ANALYSIS IN BLOWER

ANSYS CFX consists of five software modules which are linked by the flow of information required to perform a CFD analysis.

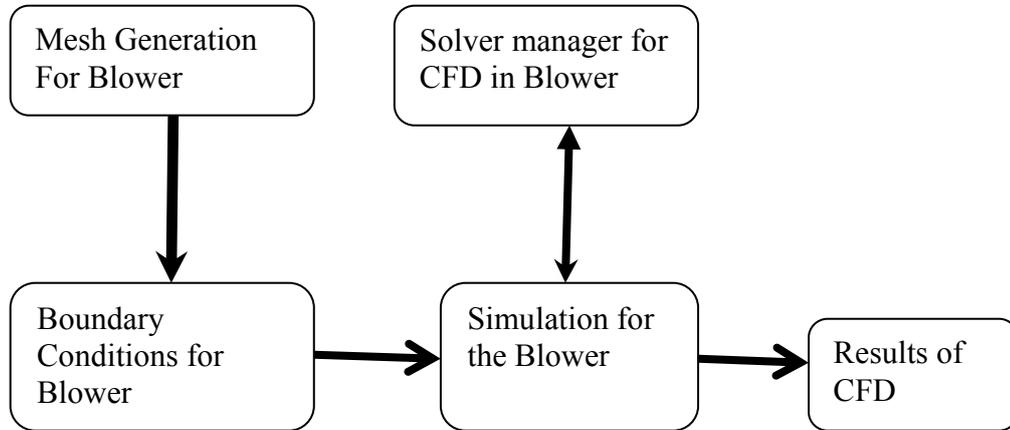


Figure: 5.2-A Structure of CFD

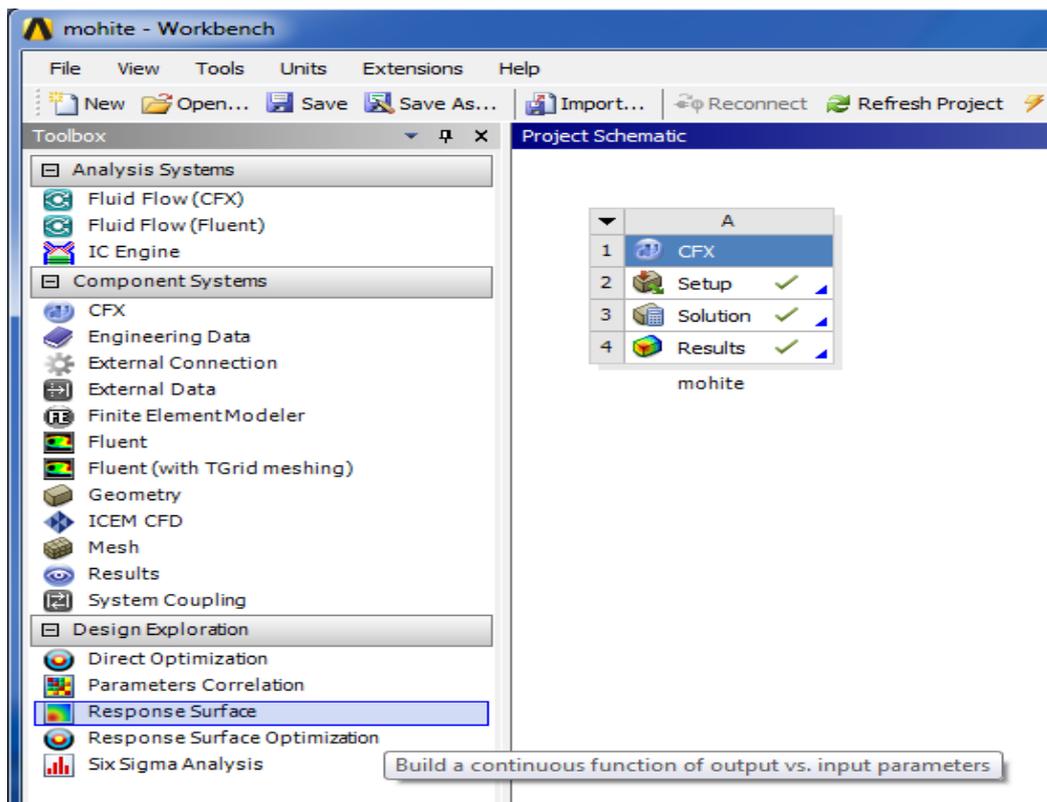


Figure: 5.2-B Structure of ANSYS CFX- Workbench

Above figure: 5.2-A & B shows that structure of CFX contains setup as CFX-Pre, Solution as ANSYS CFX and Result-CFX-Post. Complete structure is interlinked with each other. First is CFX-Pre which contains geometry and mesh.

5.3 MESH GENERATION FOR BLOWER

CFX-Mesh is a mesh generator aimed at producing high quality meshes for use in computational fluid dynamics (CFD) simulations. CFD requires the use of meshes which can resolve boundary layer phenomena and which satisfy more stringent quality criteria than structural analyses. CFX-Mesh produces meshes containing tetrahedral, prisms and pyramids in standard 3D meshing mode, and additionally can include hexahedra in the Extrude 2D meshing mode. It produces output in the form of a CFX-Pre Mesh File, suitable for importing directly into CFX-Pre, the CFX- pre-processor.

The steps to create a mesh are as follows:

1. Create the Geometry
2. Define Regions
3. Define the Mesh Attributes
4. Create the Surface Mesh
5. Create the Volume Mesh

5.3.1 CREATE THE GEOMETRY

You can create geometry for CFX-Mesh from sketch in Design Modeler within ANSYS Workbench or import it from an external CAD file. CFX-Mesh requires you to construct Solid Bodies (not Surface Bodies) to define the region for the mesh. A separate Solid Body must be created for each region of interest in the CFD simulation: for example, a region in which you want the CFD solver to solve for heat transfer only must be created as a separate Solid Body. Multiple Solid Bodies are created in Design Modeler by use of the Freeze command as shown in figure: 5.3. Considering the various design conditions casing geometry are generated, having dimensions as given in the table: A below.

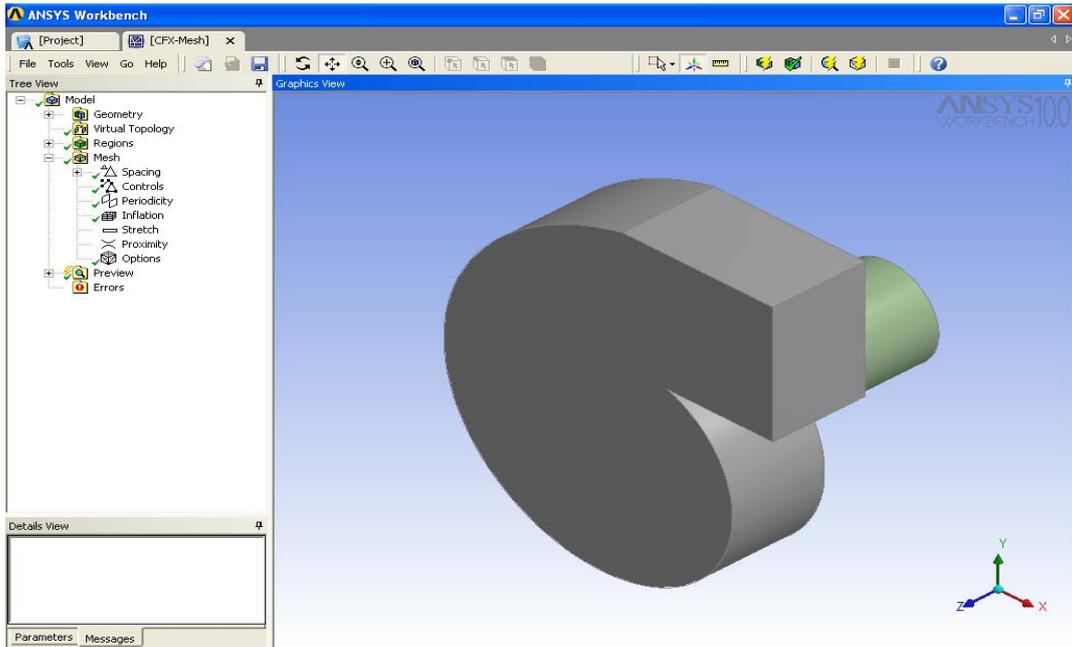


Figure: 5.3 Solid Geometry (.igs file)

Table: A

Casing-1 Vortex method $D1 = 30$ cm, $D2 = 42.5$ cm, $b1 = 17.5$ cm, $b2 = 13$ cm, $\beta1 = 200$, $\beta2 = 480$, RPM (N) = 2900 and No. of blades (Z_r) = 12			
Volute angle (θ°)	Radius (r_θ Cm)	Volute angle (θ°)	Radius (r_θ Cm)
0	23.375	225	43.19
45	26.43	270	48.84
90	29.88	315	55.22
135	33.79	360	62.43
180	38.2		
Casing-2 Constant velocity method $D1 = 30$ cm, $D2 = 42.5$ cm, $b1 = 17.5$ cm, $b2 = 13$ cm, $\beta1 = 200$, $\beta2 = 480$, RPM (N) = 2900 and No. of blades (Z_r) = 12			
Volute angle (θ°)	Radius (r_θ Cm)	Volute angle (θ°)	Radius (r_θ Cm)
0	23.375	225	47.9118
45	28.2824	270	52.8192
90	33.1897	315	57.7265
135	38.0971	360	66.6339
180	43.0045		

Casing-3 forward impeller D1 = 30 cm, D2 = 42.5 cm, b1 = 17.5 cm, b2 = 13 cm, $\beta_1 = 200$, $\beta_2 = 1200$, Speed (N) = 1600 r.p.m and No. Of Blades (Zr) = 12 nos.

Volute angle (θ°)	Radius (r_θ Cm)	Volute angle (θ°)	Radius (r_θ Cm)
0	23.375	225	38.7611
45	26.4522	270	41.8383
90	29.5294	315	44.9155
135	32.6066	360	47.9927
180	35.6839		

5.3.2 DEFINE REGIONS.

When you come to set up your CFD simulation, you will need to define boundary conditions on which to apply specific physics. For example, you may need to define where the fluid enters the geometry as suction pipe of blower or where it leaves as exhaust of the casing. Although it would be possible to select the faces which correspond to a particular boundary condition in CFX-Pre, it is rather easier to make this selection in CFX-Mesh. In addition, it is much better to define the location of periodic boundaries before the mesh is generated to allow the nodes of the surface mesh to match on the two sides of the periodic boundary, which in turn allows a more accurate CFD solution.

5.3.3 DEFINE THE MESH ATTRIBUTES.

The mesh generation process in CFX-Mesh is fully automatic. However, you have considerable control over how the mesh elements are distributed, in order to ensure that you get the best CFD solution possible with your available computing resources. You can dictate the background length scale, and where and how it should be refined. In the casing, along the curve mesh over casing should be fine and special care should be taken. Existing meshes generated with a wide range of analysis packages or ANSYS CFX products can be imported into ANSYS CFX-Pre. The volume mesh can contain hexahedral,

tetrahedral, prismatic and pyramidal element types as shown in figure: 5.4

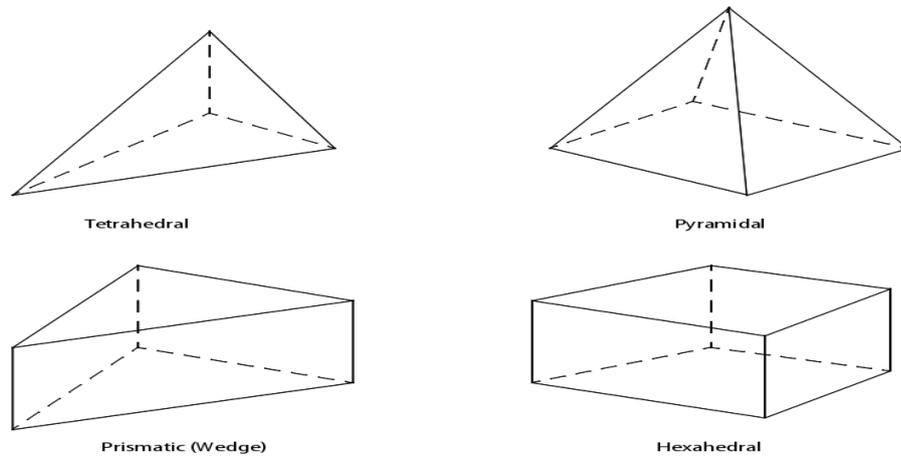


Figure: 5.4 Types of elements

The ratio of elements to nodes is approximately 5:1 for a tetrahedral mesh. For example, if 5 million tetrahedral elements are in a mesh, then there are approximately 1 million nodes. This is in contrast to a hexahedral mesh where the ratio of elements to nodes approaches 1:1 as the grid becomes large. Memory required for a tetrahedral mesh is about 0.4 times the memory required for a hexahedral mesh of the same number of elements. Alternatively a tetrahedral mesh has about 2 times the required memory of a hexahedral mesh with the same number of nodes.

In many simple cases, the need for mesh controls is removed by the setting of appropriate local face mesh spacing, Edge Proximity and Surface Proximity. These controls can be used in isolation, or in combination. Inflation is used to control the near-wall internal mesh distribution. CFX-Mesh uses all the current Mesh Control settings to determine the appropriate size of the mesh in a particular region. In general, the element size is determined by the minimum length scale from all Mesh Controls, the local length scale from surface mesh parameters and global length scale. In short of mesh should be check as per limits.

5.3.4 CREATE THE SURFACE MESH

The surface mesh will always be generated prior to the volume mesh generation. However, it is often helpful to explicitly generate at least part of the surface mesh before volume meshing, to view it and ensure that the chosen length scales and controls will have the desired effect. Included in the surface mesh generation process is a mechanism called Inflation for generating prism elements (and a small number of pyramids as required) near the walls. Inflation is used for resolving the mesh in the near wall regions to capture flow effects for viscous problems. The surface mesh can be previewed before generating the volume mesh by using the Preview function. Preview Groups can be used to view the surface mesh on selected faces or the whole surface mesh can be generated in one go.

5.3.4 CREATE THE VOLUME MESH.

The standard volume Mesher in CFX-Mesh is the Advancing Front Volume Mesher. It allows automatic tetrahedral mesh generation using efficient mesh generation techniques. An alternative volume Mesher, the Extruded 2D Meshing, is available for two-dimensional or simple extruded meshes. The CFX-5 software allows you to refine your mesh automatically as the solution to your CFD calculation is obtained. This helps to ensure that a fine mesh is used where the solution is changing most rapidly. The set-up for Mesh Adaption takes place in the CFX-Pre software, not in CFX-Mesh. Mesh Adaption can be used to improve a reasonable solution; it cannot be used to produce good solutions from a initial poor quality mesh, so you must still generate a reasonable mesh to begin with. The Global numbers of Nodes, Elements and Faces as shown in figure:5.5 for the present case are as under.

Global number of nodes: 56,643

Global number of elements: 2, 95,574

Global number of faces: 20,918

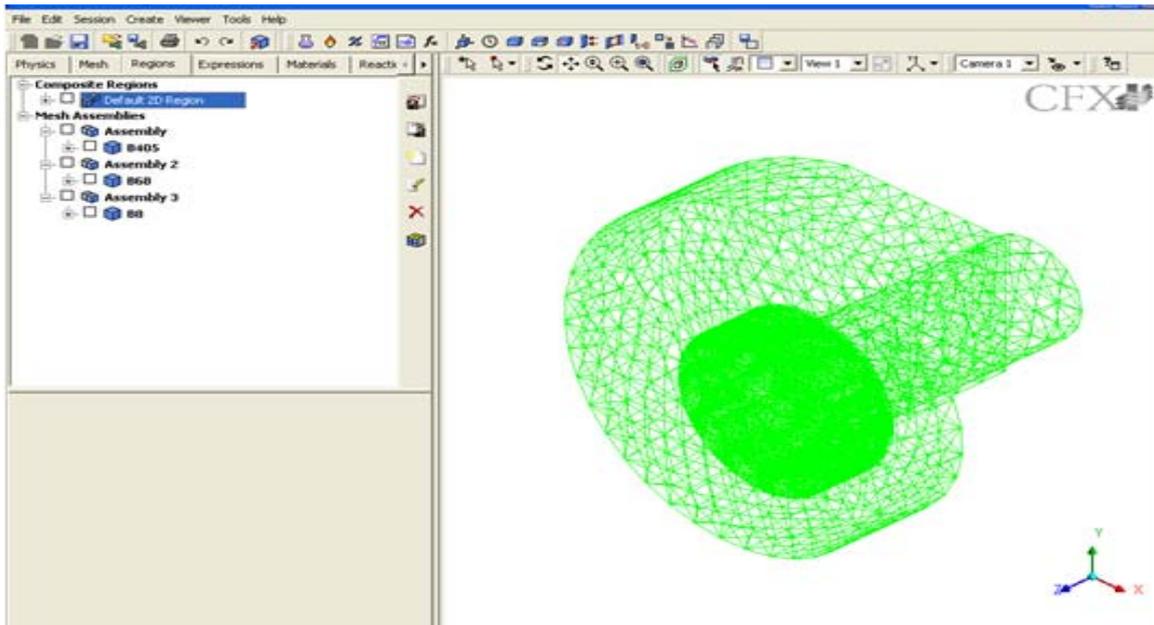


Figure 5.5 Meshing (.gtm file)

5.4 BOUNDARY CONDITIONS FOR BLOWER CASING

ANSYS CFX-Pre, can after generating mesh over the Blower or import mesh files produced by a range of mesh generation software packages. It can be generated in Ansys also by selecting Mesh option. When specifying domain interfaces in CFX-Pre, you must select the type of analysis that will be carried out in the solver. The choices are Frozen Rotor, Stage and Transient Rotor Stator. The Frozen Rotor model treats the flow from one component to the next by changing the frame of reference while maintaining the relative position of the components. The Frozen Rotor model is used for non-axisymmetric flow domains, in Impeller / volute. It can also be used for axial compressors and turbines. The Frozen Rotor model has the advantages of being robust, using less computer resources than the other frame change models, and being well suited for high blade counts. The drawbacks of the model include inadequate prediction of physics for local flow values and sensitivity of the results to the relative position of the rotor and stator for tightly coupled components. Blower system contains Domain1 as inlet pipe, Domain2 as impeller and Domain3 as casing. As per path of flow from suction

inlet pipe to impeller GGI interface1 is created and between impeller and casing GGI interface2 is created.

Flow physics, boundary conditions, initial values and solver parameters such as number of iterations and residual type are specified in ANSYS CFX-Pre as shown in figure: 5.6-A & B. A full range of boundary conditions, including inlets, outlets and openings, together with boundary conditions, are applied in ANSYS CFX through ANSYS CFX-Pre as shown in figure:5.7. There are several different types of boundary condition that can be applied at the inlet boundary. For example, the velocity (or mass flow rate) can be specified or total pressure can be specified. The velocity is specified over the entire inlet face of the impeller for most computational cases .and at the outlet static pressure can be applied. For present case following boundary conditions are applied.

Casing-1	At inlet, At outlet,	static pressure= -588.6 Pascal. Static pressure= 1000 Pascal.
Casing-2	At inlet, At outlet,	velocity=18.20 m/s. Static pressure=1633 Pa.
Casing-3	At inlet, At outlet,	velocity=18.20 m/s. Static pressure=2000 Pa.

The volute casing is stationary and the impeller is rotating with a rotational speed of 2900 rev/min. Additionally, the k-ε turbulence model in ANSYS CFX 14.0 applied for the turbulence intensity. The computations for the present work run with 5% turbulence intensity.

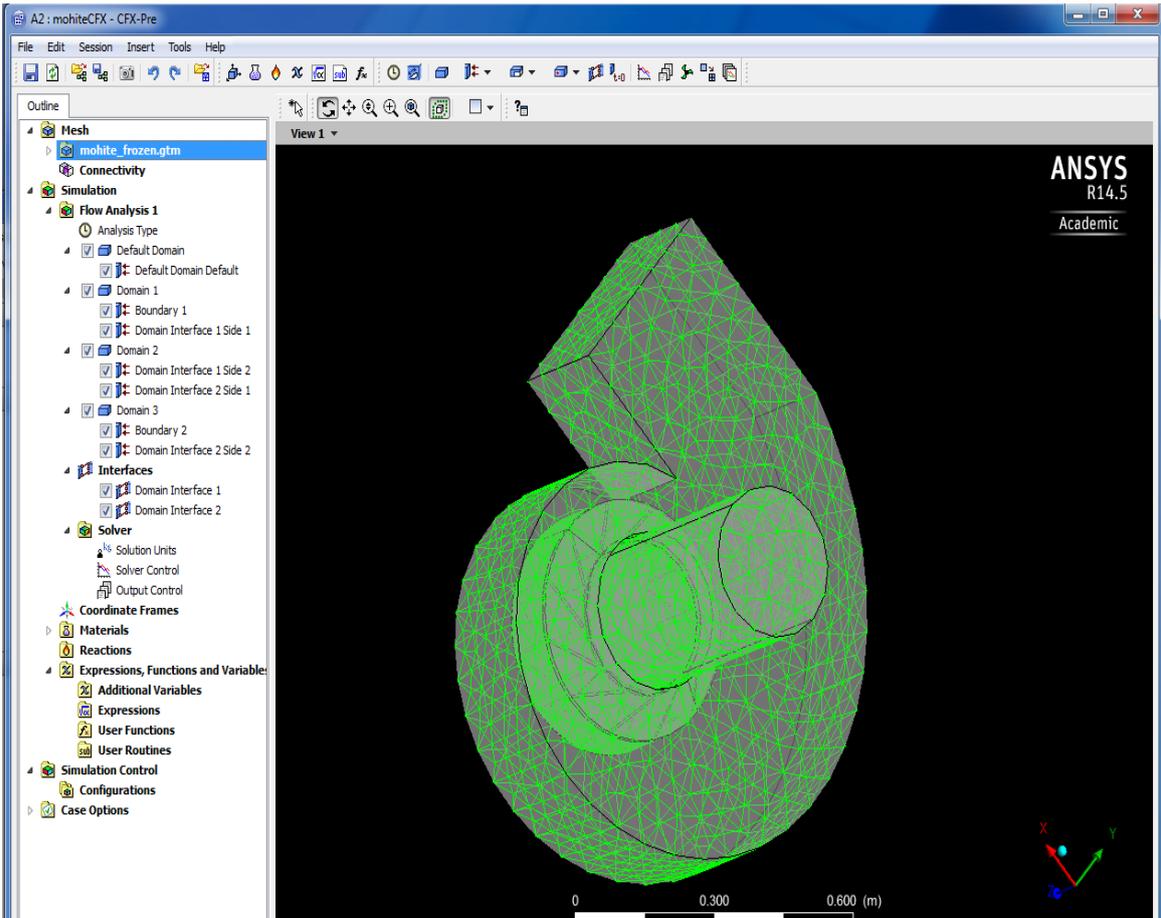


Figure: 5.6-A ANSYS CFX- Pre

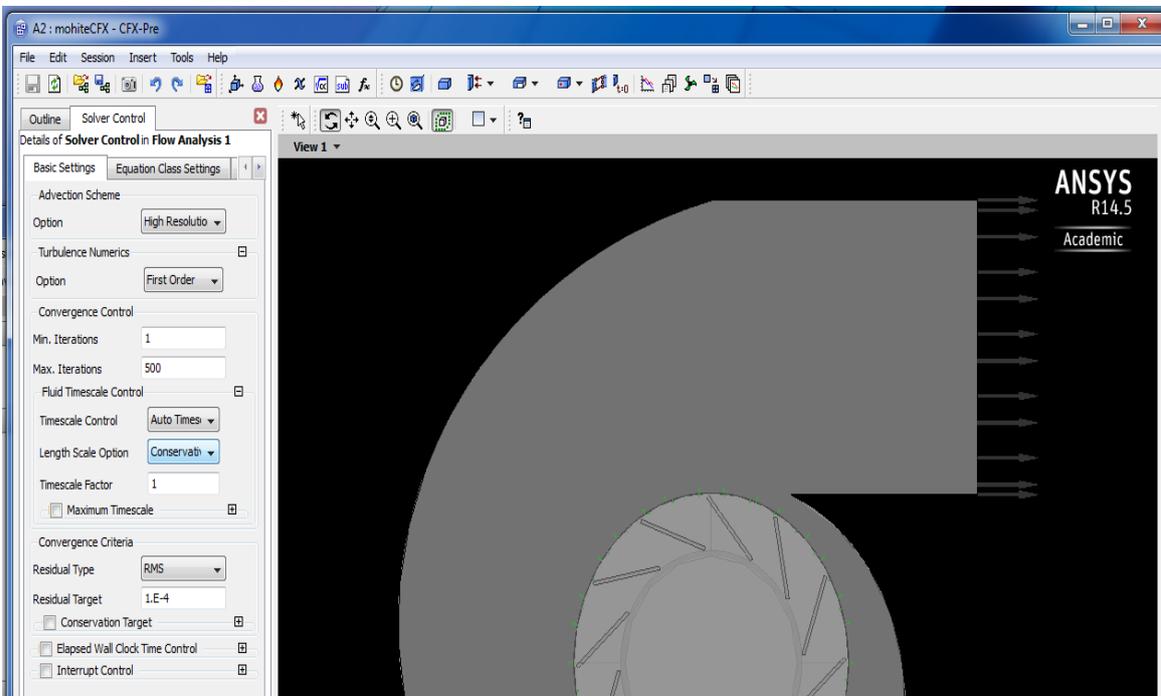


Figure: 5.6-B ANSYS CFX- Pre with solver control

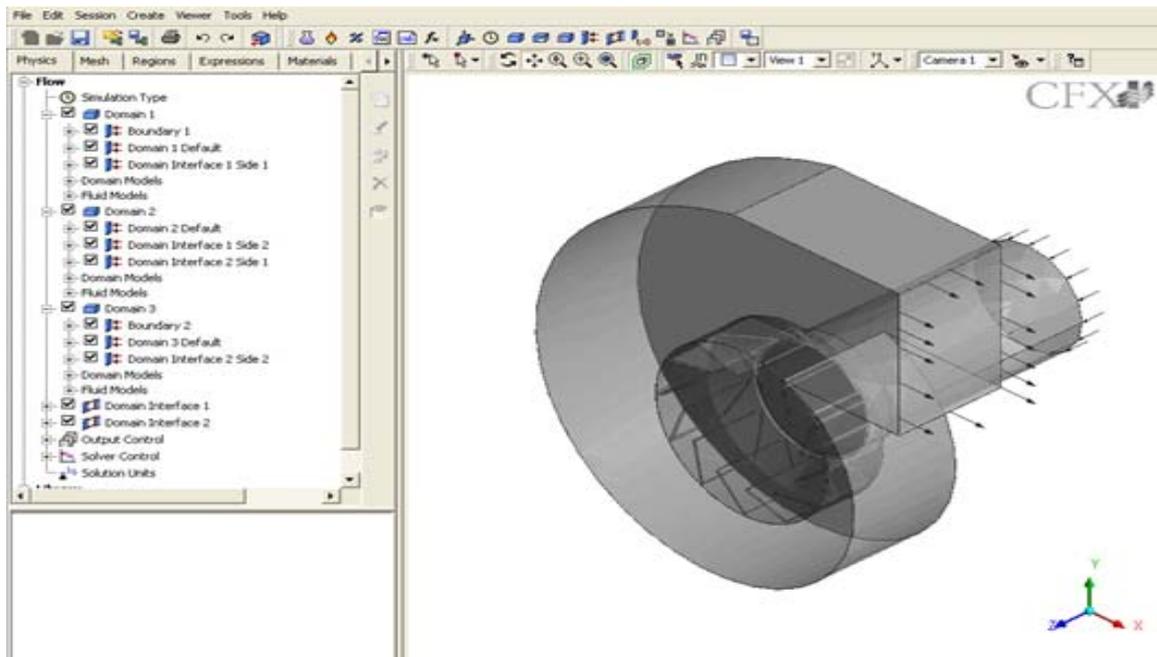


Figure: 5.7 Boundary conditions (.cfx file)

5.5 SIMULATION FOR THE BLOWER CASING

ANSYS CFX-Solver solves all the solution variables for the simulation for the problem specification generated in ANSYS CFX-Pre. One of the most important features of ANSYS CFX is its use of a coupled solver, in which all the hydrodynamic equations are solved as a single system. The coupled solver is faster than the traditional segregated solver and less iteration are required to obtain a converged flow solution. Default parameters are used for the steady state simulation. The present case was run on Intel (R), Dual core processor with 6GB of RAM hardware configuration. It's taken almost seven hours for 1000 iterations for a run to converge down to maximum residuals between E-02 and E-03. As shown in figure: 5.8 RUN command is given to solve the solution. Finally, results are generated in post-processor.

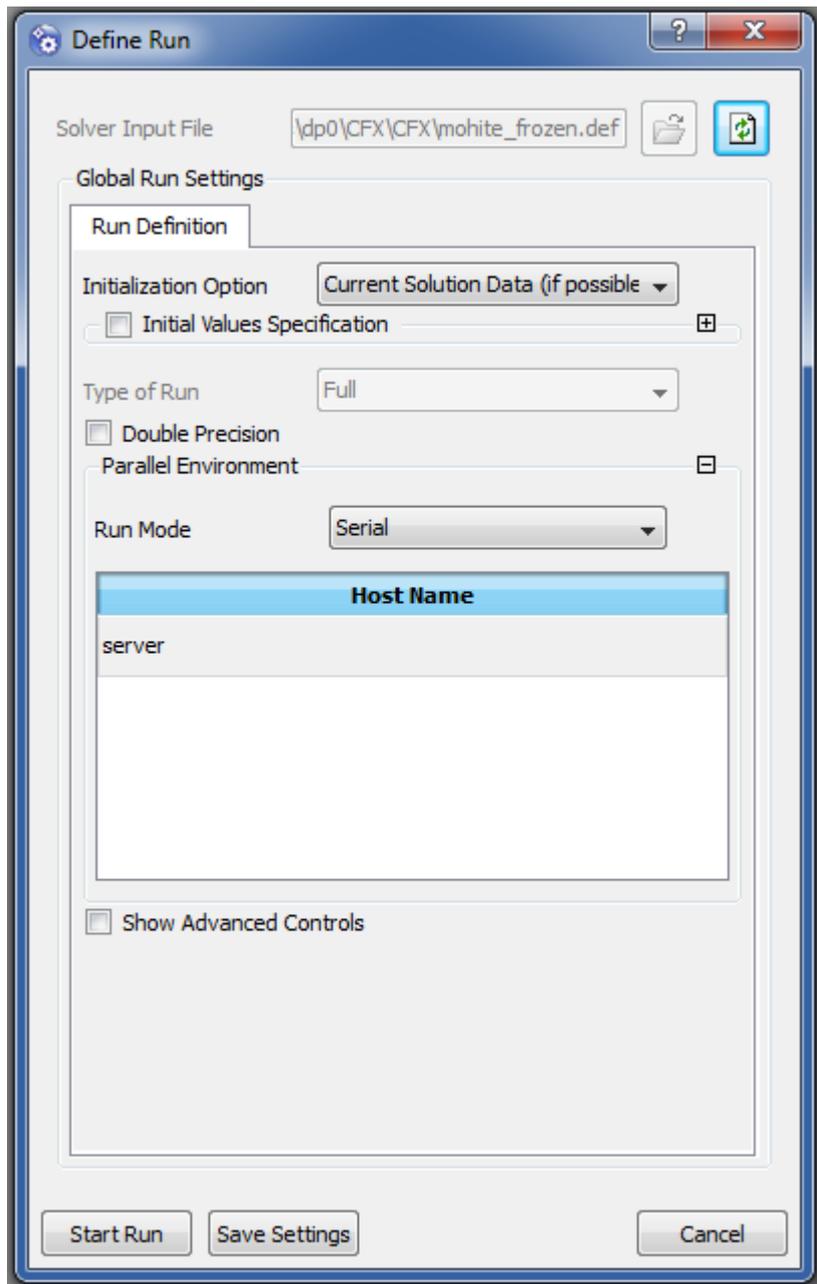


Figure: 5.8 Solver to run

5.6 SOLVER MANAGER FOR CFD IN BLOWER

The ANSYS CFX-Solver Manager is a module that provides greater control to manage the CFD task as shown in figure: 5.9. Its major functions are:

- Specify the input files to the ANSYS CFX-Solver.
- Start/stop the ANSYS CFX-Solver.
- Monitor the progress of the solution.
- Set up the ANSYS CFX-Solver for a parallel calculation.

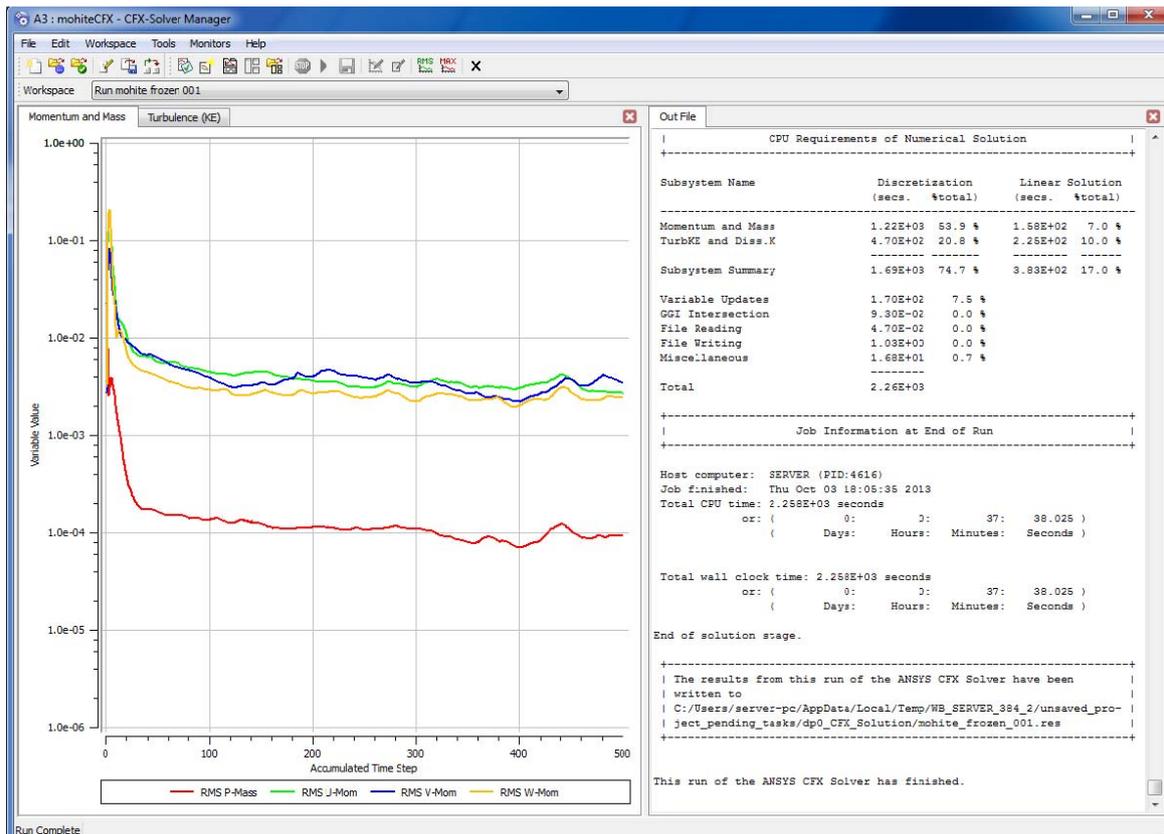


Figure: 5.9 Solver Manager

5.7 RESULTS OF CFD SIMULATION

ANSYS CFX-Post provides state-of-the-art interactive post-processing graphics tools to analyses and presents the ANSYS CFX simulation results. Important features include:

- Visualization of the geometry and control volumes
- Vector plots showing the direction and magnitude of the flow at various plane in volute casing.
- Visualization of the variation of scalar variables (variables which have only magnitude, not direction, such as pressure) through the domain as mention in blower.
- Quantitative numerical calculations if any extra output is required.
- Animation-In casing this very important to visualize the whole effect.
- Charts showing graphical plots of variables
- Hardcopy output

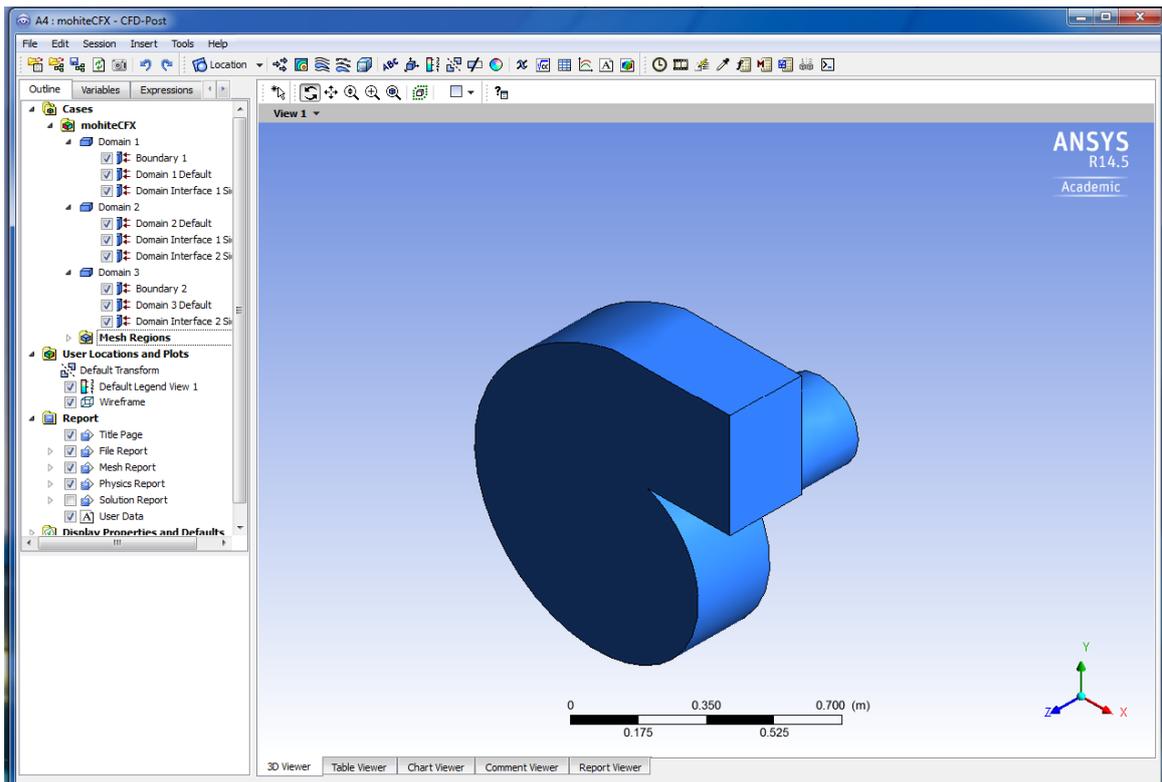


Figure: 5.10 CFX- Post

Finally, results are generated in post-processor. After CFD analysis static analysis is required to get strength of the casing for material selection and providing the support for holding the casing.

5.8 STATIC ANALYSIS OF BLOWER CASING

After CFD analysis, geometry and result file can be input of the static analysis. In static solution, we have to mention material properties Poisson ratio or selection the material from the material library of ANSYS and load direction if any. In this analysis, we can vary the thickness of the material, Size of the support of the casing and position of support of the casing. This analysis is helpful in the selection of material as per the requirement. In figure: 5.11 show selection of material with various properties as per requirement. Output of CFX result is the input to the static analysis. In figure: 5.12 show the arrangement of analysis.

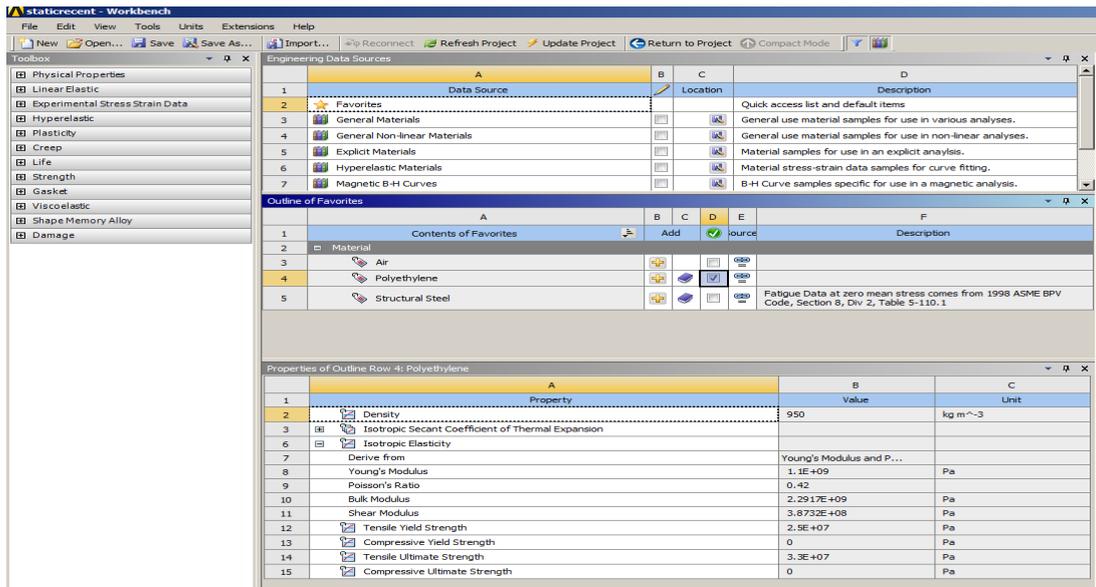


Figure: 5.11 material available in ansys Library

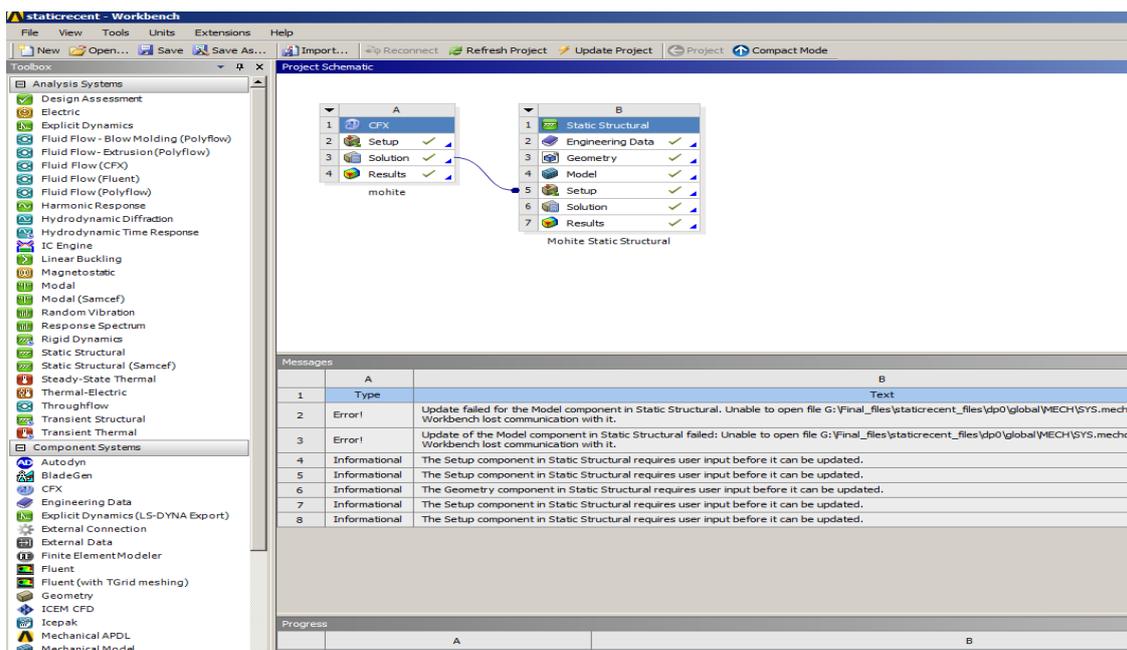


Figure: 5.12

Creating the model in any software or in ansys workbench with the design condition, can be input to the CFD software of ansys. After generating a proper mesh over a casing as shown in figure: 5.13 and with the boundary conditions final analysis is done.

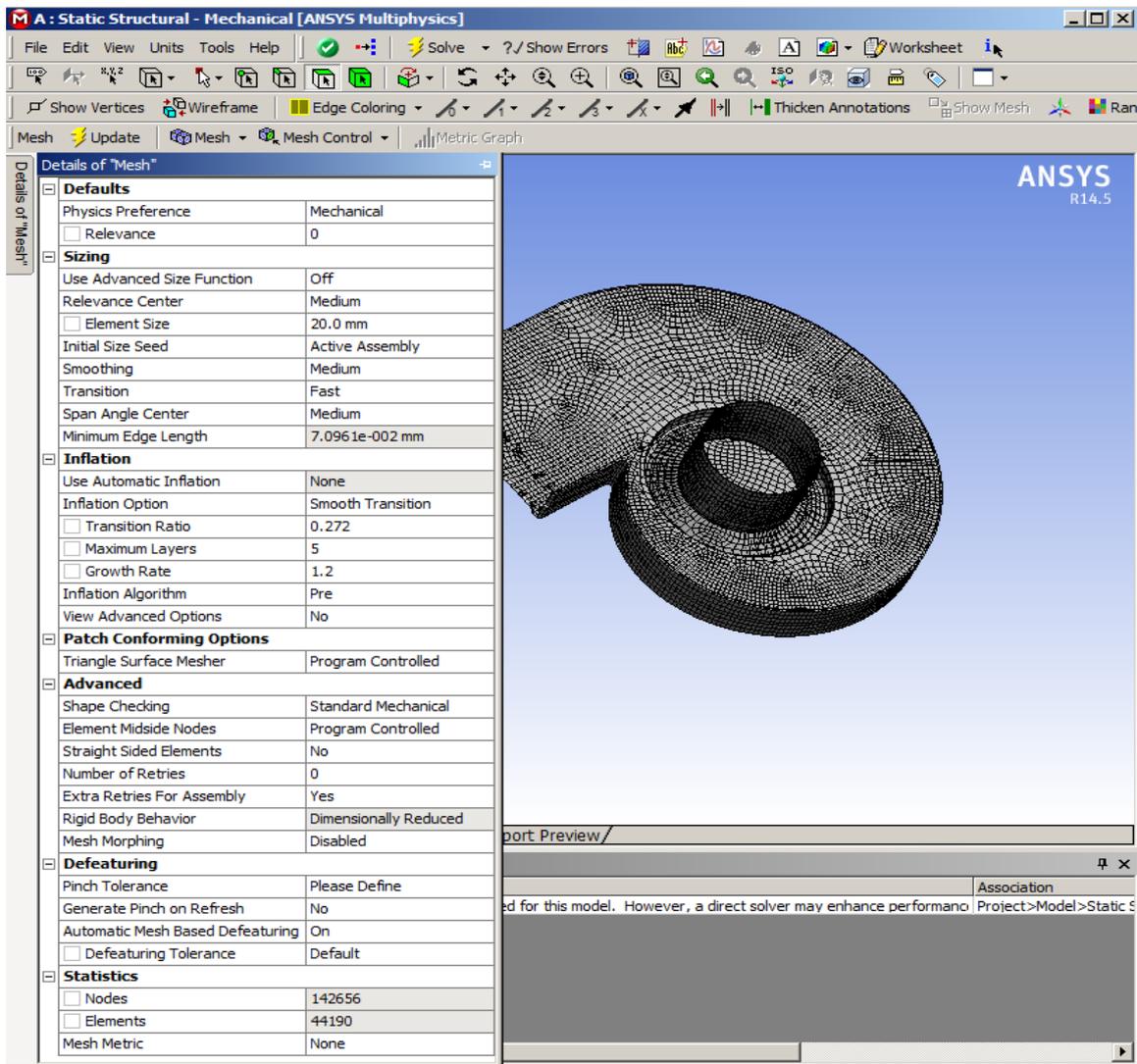


Figure: 5.13 shows selection to improve the quality of mesh.

By using various materials such as FRP, steel and sheet metal with properties shown in below table as per ASME BPV code, section 8, Div 2, Table 5-110.1, with this analysis is done with change in thickness.

Table: B

	Structural Steel	Polyethylene	FRP
Density	7850 Kg/m ³	950 Kg/m ³	1700 Kg/m ³
Young's Modulus	2E+11 Pa	1.1E+09 Pa	2.06E+10 Pa
Poisson's Ratio	0.3	0.42	0.5
Bulk Modulus	1.6667E+11 Pa	2.2917E+09 Pa	
Shear Modulus	7.692E+10 Pa	3.8732E+08 Pa	
Tensile Yield Strength	2.5E+08 Pa	2.5E+07 Pa	
Compressive Yield Strength	2.5E+08 Pa	0 Pa	
Tensile Ultimate Strength	4.6E+08 Pa	3.3E+07 Pa	

5.9 SUMMARY

CFD analysis is done by Using ANSYS CFX code on two versions 14 & 14.5. Model is generated in different cad software and in ANSYS workbench also. After importing the cad file in ANSYS, repair of some joints and exact overlapping of the surface are checked. This joint can create problem in creating the mesh. Mesh generation fails many times due to wrong selection of surface. Boundary conditions are applied as per the various analysis required. Static and dynamic analysis of volute casing, before and after the CFD analysis is done and CFD analysis is done with various experimental conditions.

Finally, experimental and CFD analysis results are generated at same boundary conditions. These results are plotted in contours of pressure and velocity. The velocity vectors showing flow direction at various locations in the volute casing. Result obtained is to be understood with various effects, to evaluate the analysis of flow in volute casing of blower.