Design and Analysis of Multistage Centrifugal Pump

Project Title

Design And Analysis Of Multi Stage Centrifugal Pump

Guided By:
Prof. M.P.Rajpara

Prepared By:
Karan G. Patel(110780119005)
Vishal M. Patel(110780119022)
Chintan K. Darji(110784119001)
Mayur S. Patel(100780119045)

Contents

- Introduction
- Literature Review
- Methodology
- Introduction and Design of Pump Parts
- Efficiency Calculations
- CFD Work
- Conclusion
- References

Introduction

- Our Group is doing project at NIRMA Ltd.,Mandali
- In this project the design is made in CREO PARAMATRIC 2.0
- And then the gained design will be analyzed in ANSYS 15.0
- Whatever the results from this analysis is gained on the basis of that the redesign of the pump for increment in the efficiency will be done.

Literature Review

- **Title:-** About Designing the Flow Part of a Multi-Stage Pump with a Minimum Radial Dimensions
- **Authors:-** Igor Tverdokhleb, Elena Knyazeva, Aleksander Birukov, Svetlana Lugovaya
- **Conclusion:-**

➢ i) As a result of investigations, there was revealed the possibility to diminish dimensions of a flowing part at the expense of reducing the external radius of the impellor D2 and ratio DGV/D2, and reducing D2 is ensured at the expense of increasing the number of vanes while applying them in the impellor of two-row vane system.

➢ ii) The comparative analysis of the theoretical head pressure curves of the two-row impellor, counted by PT, NS and Euler's equation results corrected for the finite number of vanes, showed that for accurate forecasting head pressure performance, in this case, it is required to specify the formula of correction for the finite number of vanes.

➢ iii) Operations on researching flow in the two-row impellor for the purpose of creating a technique for designing highly effective flowing parts should be continued.

Literature Review

- **Title:-** Revisited Designing of Intermediate Stage Guide Vane of Centrifugal Pump
- **Authors:-** Svetlana Lugovaya, Pavel Olshtynsky, Andrey Rudenko, Igor Tverdokhleb
- **Conclusion:-**

➢ i) One of the ways to reduce the cost of a multistage centrifugal pump at the constant rotational speed is to reduce its weight and size characteristics using the stage, whereat there is applied an impeller with ITZ.

➢ ii) The analysis of various references and the review of existing flowing parts has displayed that the main attention at designing guide vanes had been paid to the elements of the spiral portion and GVC. Designing was carried out with the use of the one-dimensional theory of a fluid flow and did not take into account the flow spatiality.

➢ iii) The state-of-the-art review of the flowing parts with the guide vanes of various types and also the numerical simulations of the flow in the guide vane with ITZ have displayed that the recommendations known from the art and used at designing the guide vanes with CTC should be specified for the guide vanes with ITZ.

➢ iv) At designing the guide vanes with ITZ, it is necessary to pay attention not only to the GVC optimization, but also to the determination of the geometry of the space after GVC, the shape and curvature of the IC vanes and to the definition of the angles for installation of the IC vanes at the inlet and outlet.

Literature Review

- **Title: -** LATERAL CRITICAL SPEED ANALYSIS OF MULTI- STAGE CENTRIFUGAL PUMP ROTOR USING FEA
- **Authors: -** Naveena, Dr. Suresh P
- **Conclusion: -** The main objective of this work is to build and perform rotordynamics analysis of Multi-stage centrifugal pump rotor using ANSYS. The results obtained from ANSYS, Analytical calculations and RBTS (Rotor Bearing Testing Software) are closer with each other. The critical speeds for the centrifugal rotor obtained from Campbell diagram. From the Campbell diagram of 1x API, there is no resonance observed for all the operating speeds from 3600 RPM to 5130 RPM. But the first reverse mode for 2xAPI is closer to the pump operating speed, 5130 RPM. The maximum amplitude of deflection of rotor for the applied unbalance loading is determined. The maximum amplitude of deflection is 7.12% which is less than 35% of the diametric clearance is obtained from unbalance response analysis as per API 610 guidelines. New clearance calculations are within the acceptable region as per API 610 stand

Literature Review

- **Title: -** Numerical Investigation of a First Stage of a Multistage Centrifugal Pump: Impeller, Diffuser with Return Vanes, and Casing
- **Authors: -** Nicolas La Roche-Carrier, Guyh Dituba Ngoma, and Walid Ghie
- **Conclusion:** - In this study, a steady-state liquid flow in the three- dimensional first stage of a multistage centrifugal pump was numerically investigated. A model of a centrifugal pump stage composed of an impeller, diffuser, and casting was developed to analyze the impacts of the height of the impeller blades and diffuser vanes, the number of impeller blades, diffuser vanes and diffuser return vanes, and the wall roughness height on the pump stage head, brake horse power, and efficiency. The results obtained demonstrate, among other things, that the pump stage head and brake horsepower increase as the height of the impeller blades and diffuser vanes and the number of impeller blades increase. Moreover, the head and efficiency increase for large volume flow rates with increasing numbers of diffuser vanes and diffuser return vanes. The brake horsepower hardly varies at all regardless of the number of diffuser vanes and diffuser return vanes. Furthermore, higher wall roughness heights of the impeller and diffuser negatively affect the head, brake horsepower, and efficiency. In all, the numerical curves obtained for the head, brake horsepower, and efficiency follow the trend of the experimental results

Literature Review

- **. Title: -** Research on Three-Dimensional Unsteady Turbulent Flow in Multistage Centrifugal Pump and Performance Prediction Based on CFD
- **Authors: -** Zhi-jian Wang, Jian-she Zheng, Lu-lu Li, and Shuai Luo
- **Conclusion: -**

➢ (i) Complicated three-dimensional flow model is built including inlet region, impeller flow region, guide-vane flow region, and exit region to simulate flow in 20BZ4 multi- stage centrifugal pump. The method of multi reference frame (MRF) is used to model rotating blades and stationary blades by FLUENT.

➢ (ii) The simulation results show that the flow in impellers is mostly uniform, no eddy, backflow, and separation flow. The Jet-wake along some blades influences the efficiency. There is secondary flow at blade end and exit of guide vanes. The pressure on pressure surface is higher than that of suction surface and the pressure difference causes the moment of resistance on rotating axis. At the inlet of suction surface the pressure is lowest and cavitations may happen there.

➢ (iii) Besides design condition, six off-design conditions are set to predict the external characteristics of hydraulic performances. The comparison between experimental data and simulation data shows that the experimental curve agrees well with the simulation curve under design condition, but under off-design conditions the unsteady factors of flow field influence the precision. The actual losses cause the efficiency of numerical simulation to be higher than that of experiment

Literature Review

- **Title: -** CFD STUDY FOR ASSESSMENT OF AXIAL THRUST BALANCE IN CENTRIFUGAL MULTISTAGE PUMPS
- **Authors: -** Stefania DELLA GATTA, Simone SALVADORI, Paolo ADAMI Laura BERTOLAZZI
- **Conclusion:-** A numerical investigation of a multistage centrifugal pump has been carried out. Driving the correct design and dimensioning of the axial bearings to balance the hydraulic thrust is the present study main target. To reach the task, both the main, the impeller chambers and the balancing drums flow have to be considered because of their relevance to the problem of axial load unbalance. Separated CFD analysis have been carried out for stage (with a mixing plane approach) and side chambers. 2D analysis to characterize the central drum and the lateral balancing drum have been developed. A methodology to predict the residual axial thrust for a complex machine has been presented and pump sensitivity to the different variables which influence axial thrust have been pointed out. Actual results show the calculated characteristics maps of the pump and how the axial thrust depends on the operating conditions of the pump as well as its mechanical wear conditions, because of effects on the main flow head and on the pressure distribution inside the side chambers.

Literature Review

- **Title: -** Numerical Analysis On The Performance Characteristics Of The Centrifugal Pump
- **Authors:** - Shalin P Marathe, Mr.Rishi R Saxena, Mr. Chetan H Solanki
- **Conclusion**:-The results obtained from the analysis shows that the characteristic curve for different outlet blade angles are completely matched with the numerical results and it can be concluded from the results that for low head operating conditions, the impeller with backward bladed works efficiently and the problem of the cavitations, which reduces the performance of the pump is less.

Literature Review

- **. Title: -** MULTISTAGE CENTRIFUGAL-PUMPS: ASSESSMENT OF A MIXING PLANE METHOD FOR CFD ANALYSIS
- **Authors: -** P.Adami, S.Della Gatta, F.Martelli, L.Bertolazzi, D. Maestri, G.Marenco , A.Piva
- **Conclusion:-** An improved CFD approach is presented for the analysis of complex pump stages performances. The method consists in steady simulations which use the mixing plane technique to couple impeller and diffuser. Many secondary effects such as the presence of leakage flows, wall friction and ventilation losses are not to be neglected in order to reach an high level of accuracy. The computational procedure gives results that fit very well the experimental data thus confirming its effectiveness. Furthermore, it is successfully applied to two centrifugal pumps which are equipped with different diffuser configurations and are designed for different operating conditions. Finally a quick and reliable prediction of head-rate and efficiency-rate curves in a wide operating condition range is obtainable with the present approach. However the additional parasitic effects has to be more deeply investigated at low flow rates where greater is their influence.

Literature Review

- **Title: -** EXPERIMENTAL INVESTIGATION OF OPEN WELL CENTRIFUGAL PUMP PERFORMANCE
- **Authors: -** Mr. Ashok Thummar, Mr. Vijay F. Pipalia, Tushar V. Javiya
- **Conclusion:** - The experiment shows some looses of centrifugal pump with the values Q, H, Power, Speed etc. are determined for the various operating points. In centrifugal pumps, the delivery head depends on the flow rate. This relationship, also called pump performance, is illustrated by various graphs. In a today competitive and sophisticated technology, Centrifugal pump is more widely used than any other Applications because the advantages of following factors are effect on the centrifugal pump. 1. Its initial cost is low 2. Efficiency is high 3. Discharge is uniform and continuous flow 4. Installation and maintenance is easy. 5. It can run

  at high speeds without the risk of separation of flow.

Literature Review

- **Title: -** COMPUTATIONAL ANALYSIS ON PERFORMANCE OF A CENTRIFUGAL PUMP IMPELLER
- **Authors: -** P.USHA SRI, C. SYAMSUNDAR
- **Conclusion: -** A centrifugal pump impeller was designed and analyzed with the aid of computational flow dynamics. The flow patterns through the pump, performance results, static pressure contours, absolute velocity vectors, blade loading charts at 50% span, streamwise variation of mass averaged total pressure and static pressure, streamwise variation of area averaged absolute velocity, and pressure contours on blade to blade plane are predicted for five different flow coefficients. The CFD predicted value of the head at the designed flow rate is approximately H=9.4528 m. There is 5.78% of difference between the theoretical head and the predicted numerical head. The increase of the designed flow rate causes a reduction in the total head of the pump. With the increase of mass flow rates drop in static pressures are observed from pressure contours on mid span for different flow coefficients.At designed or more than designed mass flow rate, the fluid flows smoothly along the blade walls. The blade curvature exhibits a weak vortex at the pressure side of the blade. On pressure side of the blade static pressure drop is observed. At low mass flow rates a recirculation zone is established near the leading edge of each blade.

Pump Layout

- The Layout of the pump is shown bellow…

Grundfos CR 32-12

Methodology

1. Find out & study of Pump & Pump problems
2. Visit the Industry and locate the required pump
3. Take part wise complete physical dimensions
4. Create 3D model into CREO Parametric 2.0
5. Upload created model into ANSYS R15.0(Workbench)
6. Study the acquired results
7. Modify the pump on the basis of the acquired results
8. Analysis the modified pump
9. Study the acquired results
10. Conclusion
11. End

Introduction and Design of Pump Parts

1. Impeller :-

Definition:-

"An impeller is a rotating component of a centrifugal pump, usually made of iron, steel, bronze, brass, aluminum or plastic, which transfers energy from the motor that drives the pump to the fluid being pumped by accelerating the fluid outwards from the center of rotation."

Calculations for the Impeller

- Dynamic Pressure Head:-

Applying Bernoulli principle: The first force cause the absolute velocity of the object as circumferential speed which means Dynamic Pressure Head.

The equation of Dynamic pressure Head is,

$H_d = U^2{}_2/2g = 14.26m$

- Static Pressure Head:-

The second force creates the static pressure. If a mass moves radially outward along a vane of the impeller its orbit will be a spiral-shaped curve. We can easily calculate it's angular speed In two dimensions the angular velocity ω is given by

$\omega = 2\pi * 2960/60 = 309.97$ rpm

Calculations for the Impeller

- So during it movement the centrifugal force F_c always present as

$F_c = mrw^2 = 1132.61$ N

- The centrifugal acceleration increase linearity on the radius of rotary R(variable) In constant gravitational acceleration g, static pressure of a column of water h is

$$H_s = gh = 96.79 \text{ m}$$

- In the centrifugal acceleration increase linearity from R_1 position to R_2 position static pressure of a column of water $R_2 - R_1$ is

$$H_s = \frac{a_c1 + a_c2}{2} R_2 - R_1$$

Calculations for the Impeller

$$H_s = \frac{\omega^2 R_1 + \omega^2 R_2}{2} R_2 - R_1 = \frac{U_2^2 - U_1^2}{2}$$ =96.79 m

- In the case the discharge of the pump is 0 static pressure save it's original value In the outlet of the pump is open air of static pressure created by the impeller drop to 0 static pressure transfer all to the dynamic pressure in vector which is highest value.
- **Velocity diagram and work done by impeller:-**The velocity diagram and work done by impeller of centrifugal pump on the liquid may be derived same way as for a turbine, since radial flow centrifugal pump acts as a reversed of an inward radial flow reaction turbine.

Calculations for the Impeller

- **The following assumption made for analysis.**

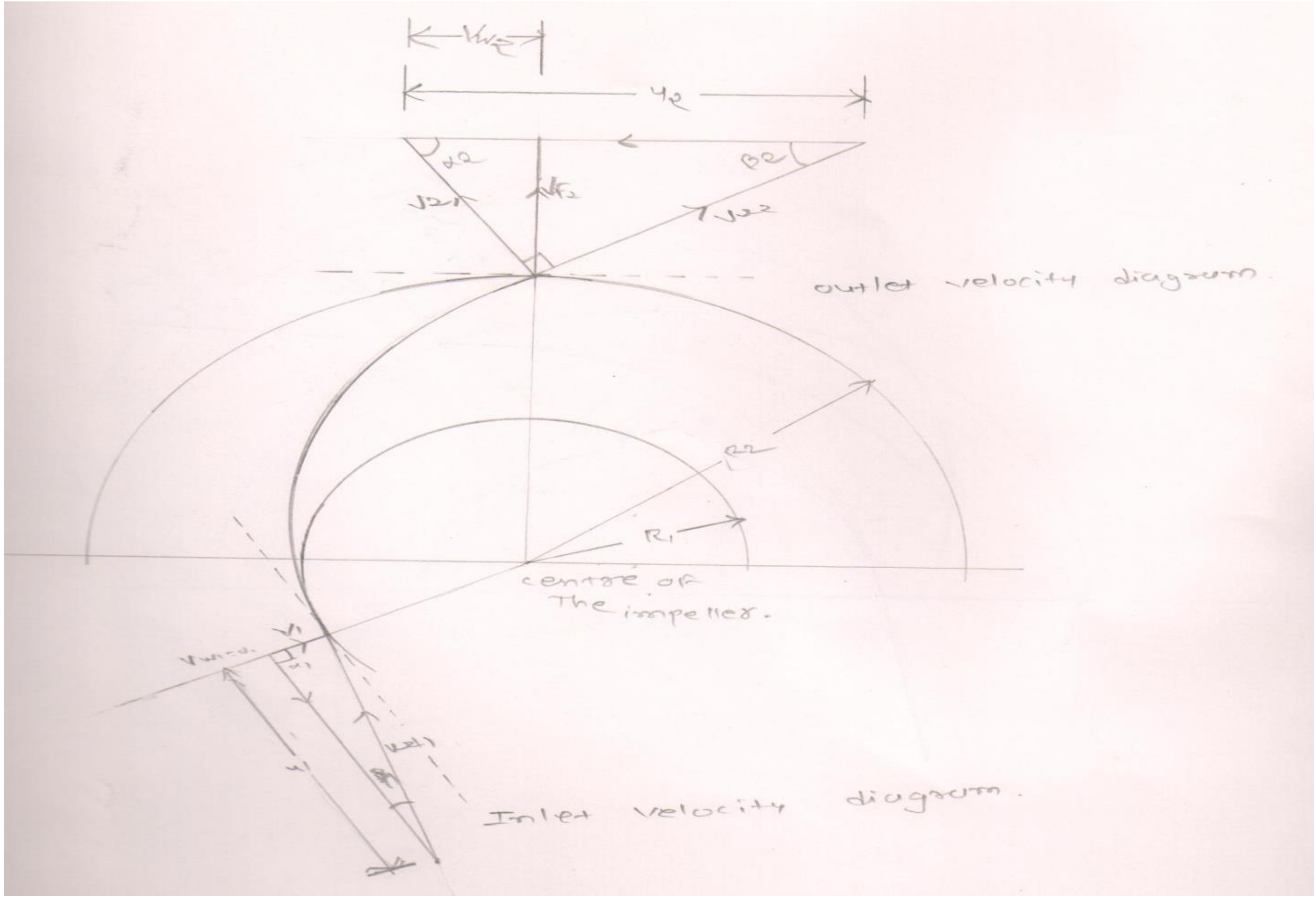
(1)Liquid enters the impeller in radial direction.

(2)No energy losses in impeller in impeller due to friction and eddy formation.

(3)Liquid enter without shock.

(4) Uniform velocity distribution in the narrow passages formed between two adjacent vanes.

Inlet and Outlet Diagram of Impeller



- Here, all dimension of our pump impeller is given below.
- At Inlet

D_1=60 mm
U_1= $\pi D_1 N/60$=9.29 m/s
β_1= 36^o
α_1=90^o
V_{r1}=7.52 m/s
V_1=4.42 m/s

- At Outlet

D_2=108 mm
U_2= $\pi D_2 N/60$=16.73 m/s
β_2= 16^o
α_2=71^o
V_{r2}=16.08 m/s
V_2=4.78 m/s
V_{f2}=4.52 m/s
V_{w2}=4.45 m/s

- Work done by impeller per sec per unit weight of liquid:-

  $= -1/g \times [V_{w1} \times u_1 - V_{w2} \times u_2]$

  $= -1/g \times [0 - V_{W2} \times u_2]$

  $= 1/g \times [V_{w2} u_2]$

  $= 9.30$

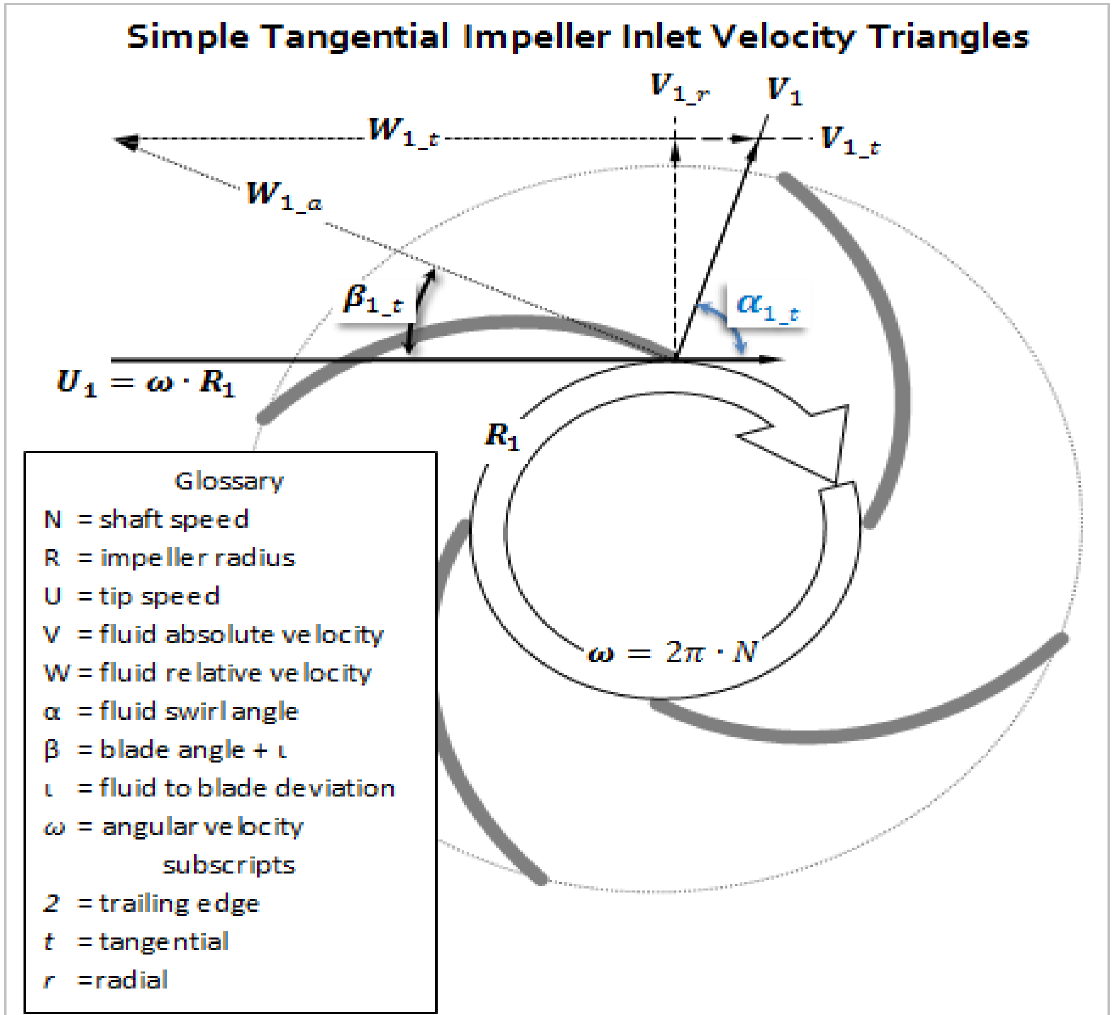
From the above data Manometric head is given by below calculation,

$Hm = (u_2^2 - V_{f2}^2 \times \mathrm{cosec}^2 \beta_2)/2g$

$= (16.73^2 - 4.5262 \times \mathrm{cosec}^2 16^{o})/2 \times 9.81$

$= 0.56$ m

Simple Tangential Impeller Velocity Triangles



Calculation

- Calculation On The Basis Of above Mentioned Figure…

At Inlet

- Cos $\beta1$ = U1 / Vr1

  Vr1 = 9.29/cos 360=4.22 m/s
- Sin $\beta1$ = V1 / Vr1

  V1 =Vr1× Sin $\beta1$ =3.65 m/s
- Sin $\beta2$ = V2 / U2

  V2=16.74 × Sin $\beta2$ =10.75 m/s
- Sin $\alpha2$ =Vf2 / V2

  Vf2 = 12.82 m/s
- D1=60 mm
- U1= πD1N/60=9.29 m/s
- $\beta1$= 36o
- $\alpha1$=90o
- Vr1=4.22 m/s
- V1=3.65 m/s

Calculation

At Outlet

- D_2=108 mm
- $U_2 = \pi D_2 N/60$=16.74 m/s
- β_2= 40 o
- α_2=50 o
- V_{r2}=8.23 m/s
- V_2=10.75 m/s
- V_{f2}=12.82 m/s
- V_{w2}=10.33 m/s
- Work done by impeller per sec per unit weight of liquid:-

  $= -1/g \times [V_{w1} \times u_1 - V_{w2} \times u_2]$

  $= -1/g \times [0 - V_{w2} \times u_2]$

  $= 1/g \times [V_{w2} u_2]$

  $= 17.6273$

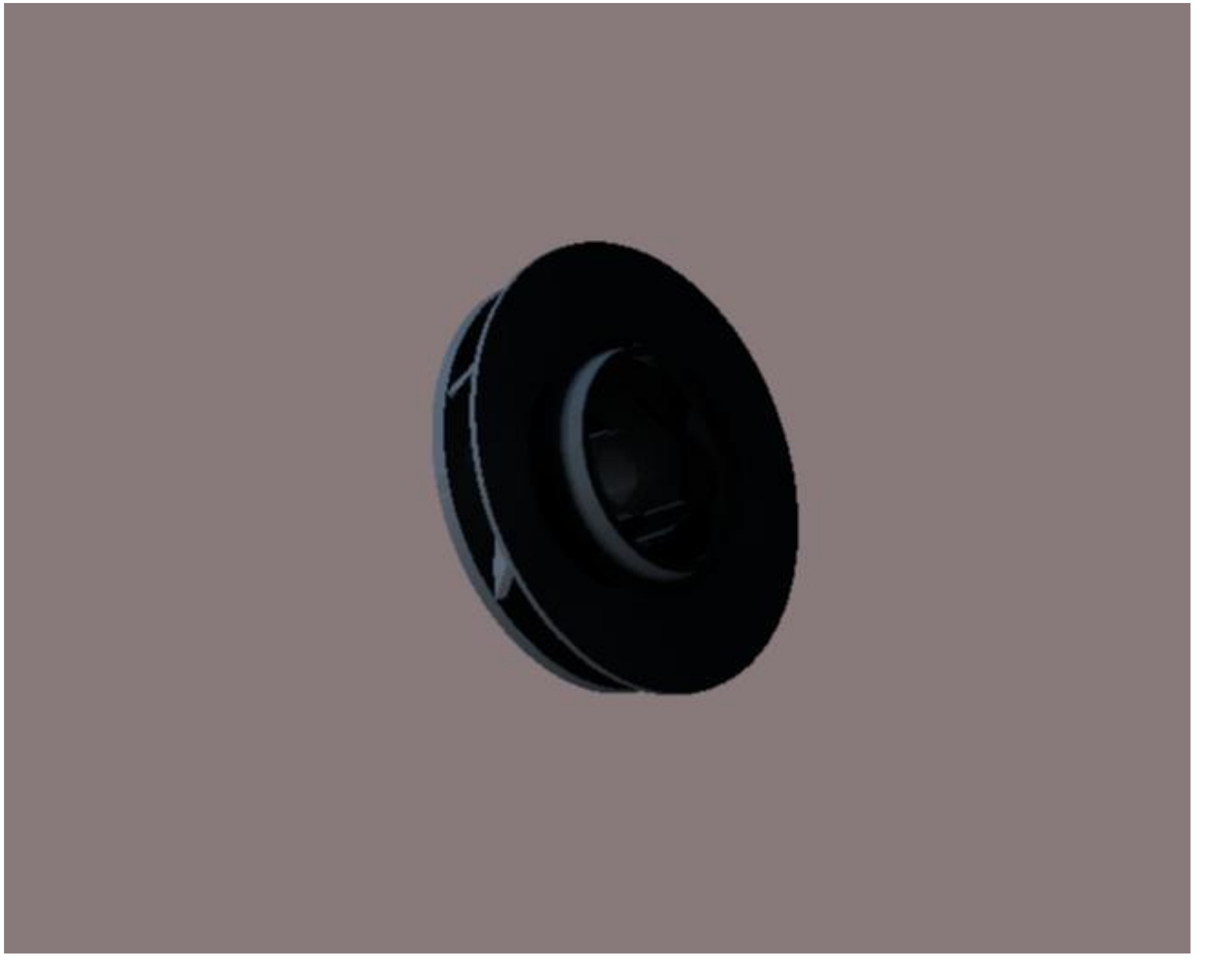
Calculation

- Hm= Total Head/No. Of Stages

  = 212 m/14

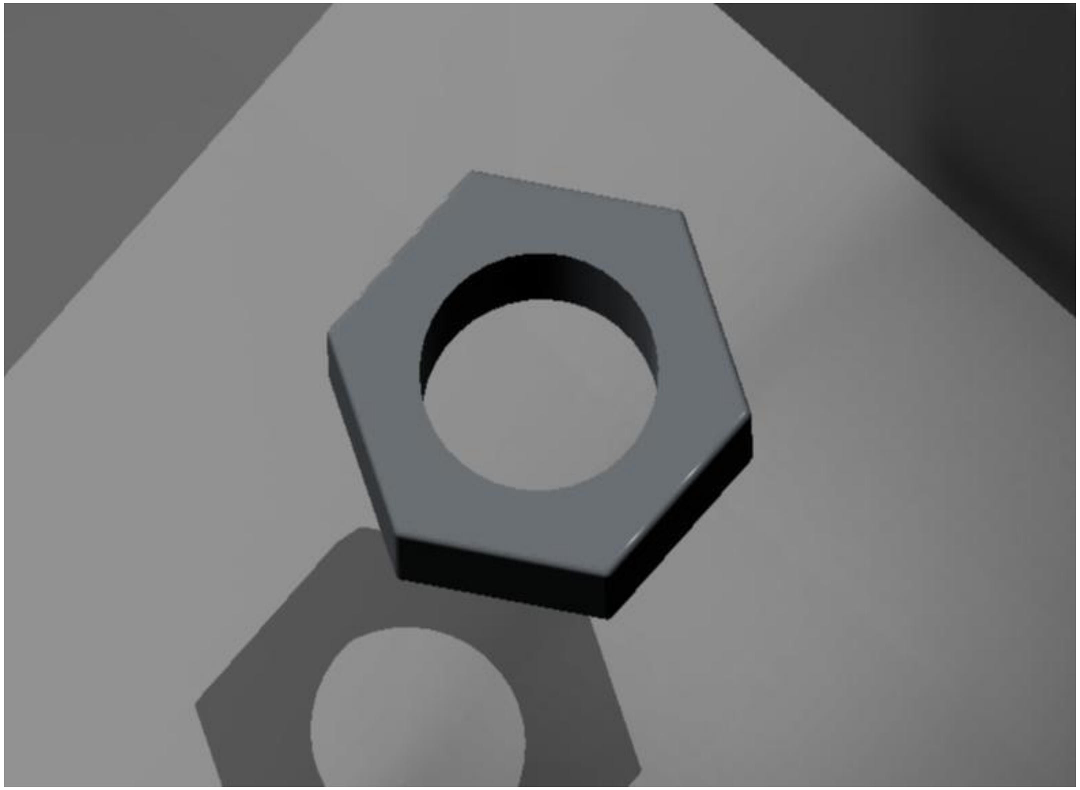
  = 15.14 m

Impeller in CREO



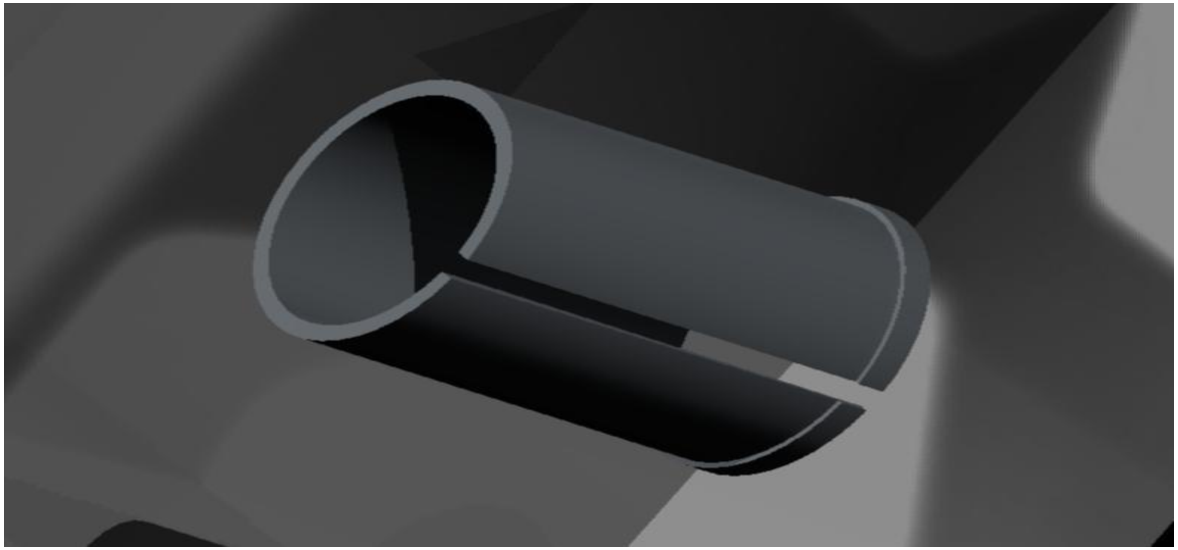
Impeller Part Design

1. Spilt Cone Nut:-



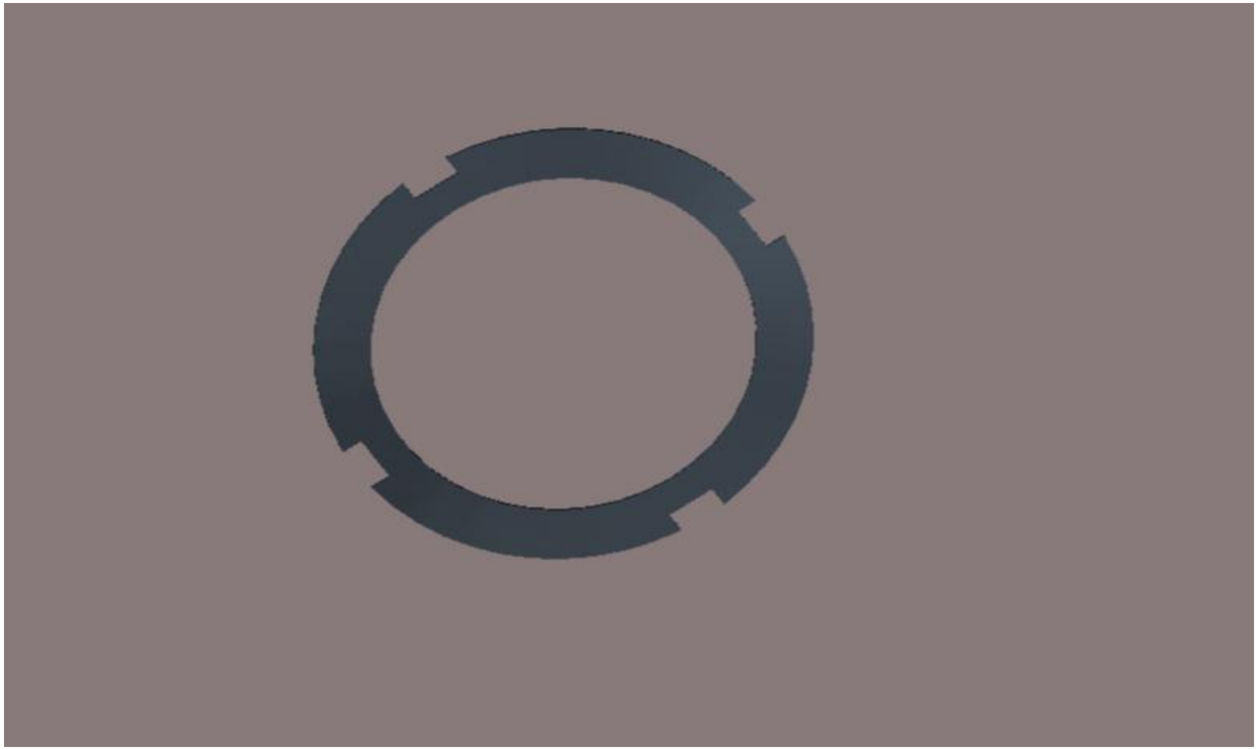
Impeller Part Design

2. Sleeve:-



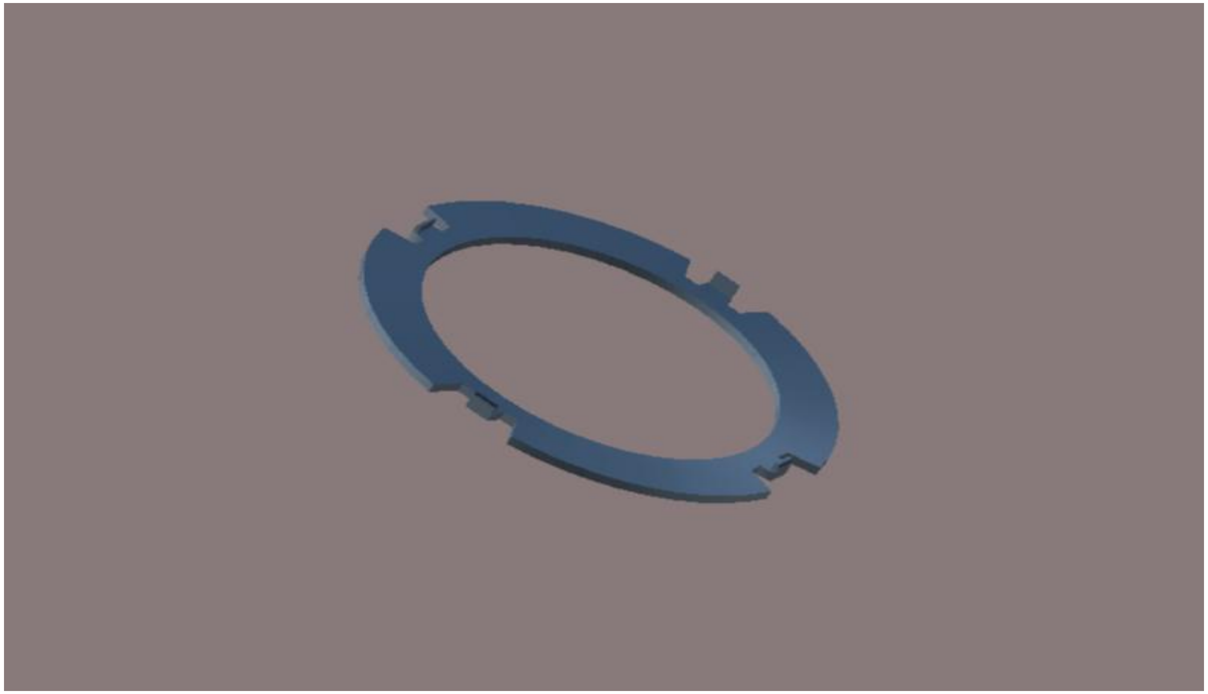
Chamber(Casing) Part Design

1. Neck Ring:-



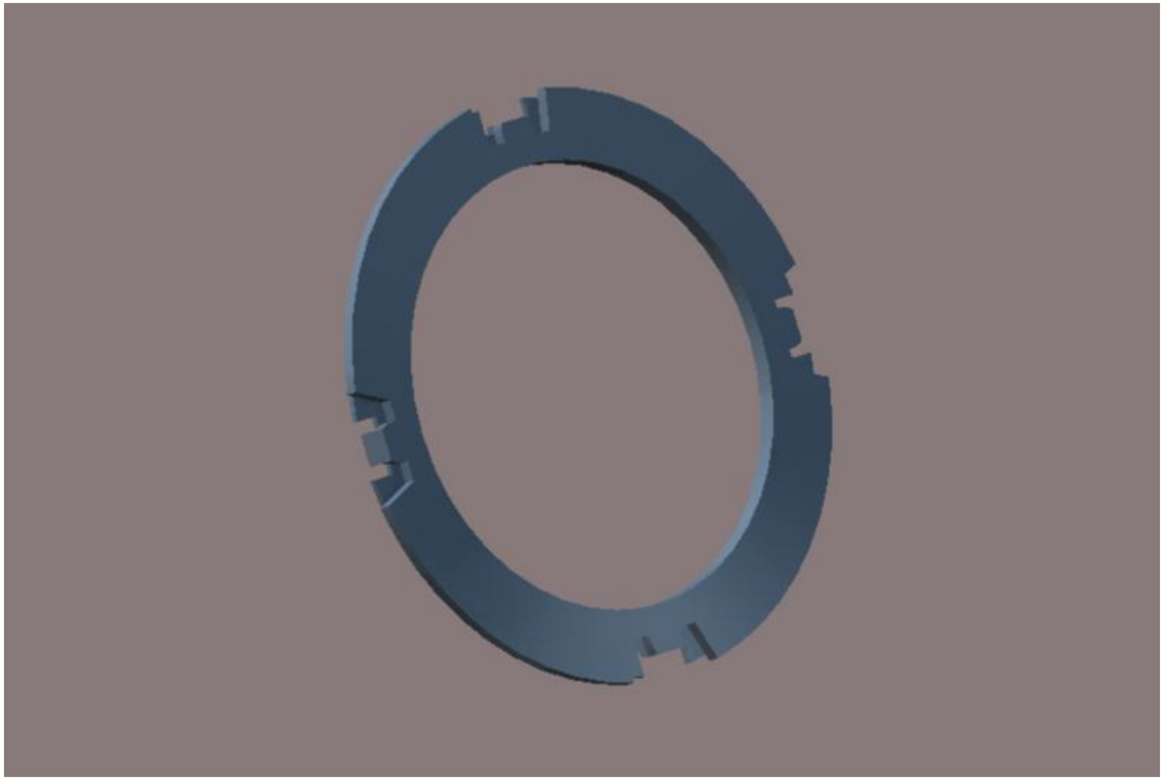
Chamber(Casing) Part Design

2. Neck Ring Retainer:-



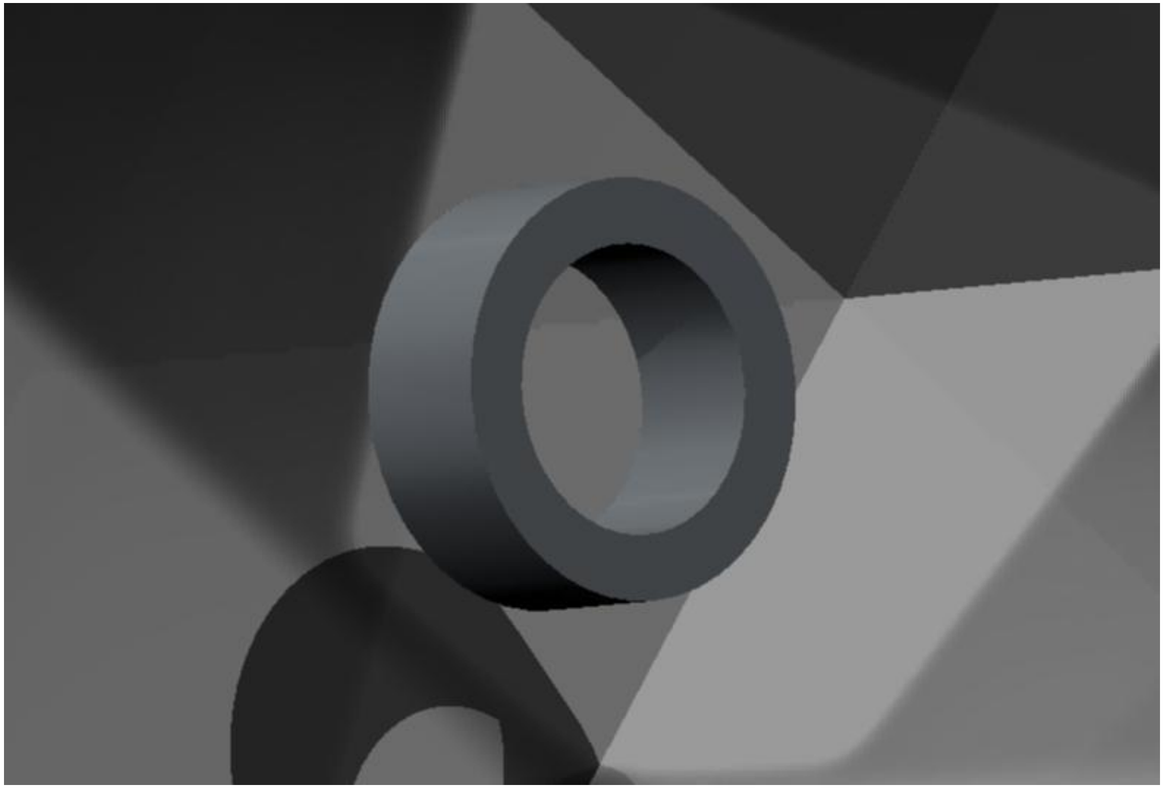
Chamber(Casing) Part Design

3. Inlet Part Complete:-



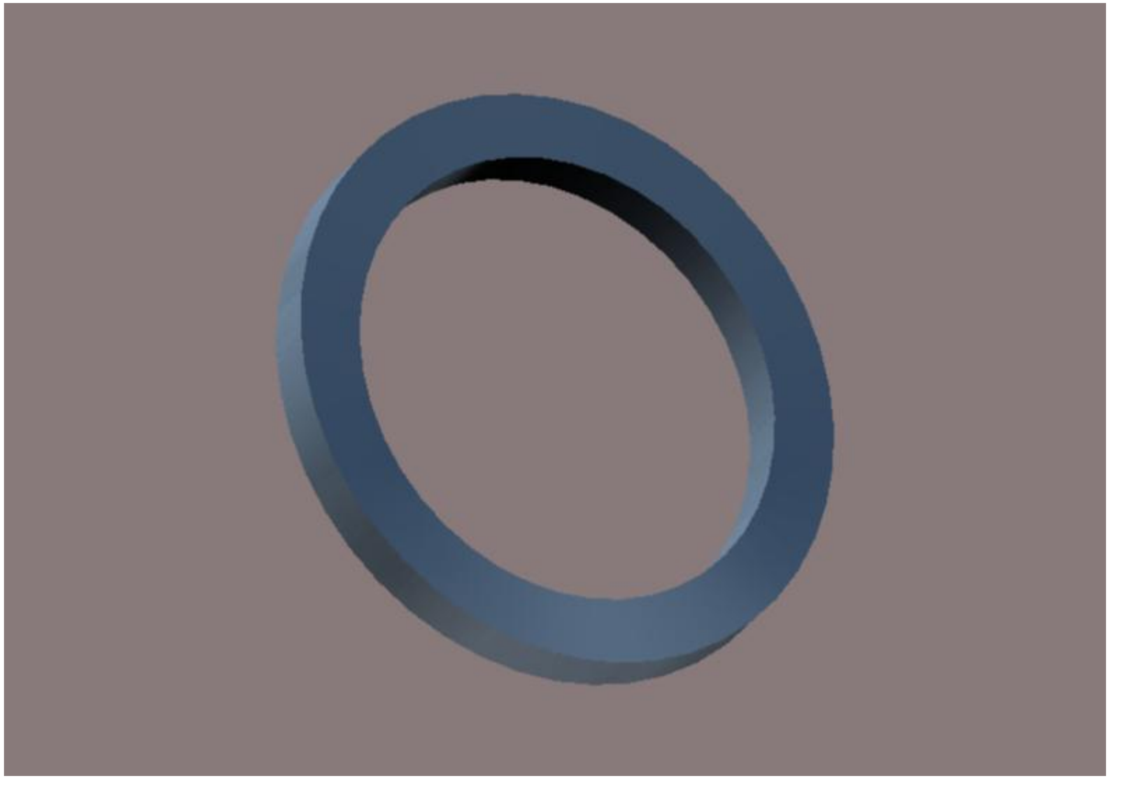
Chamber(Casing) Part Design

4. Bush:-



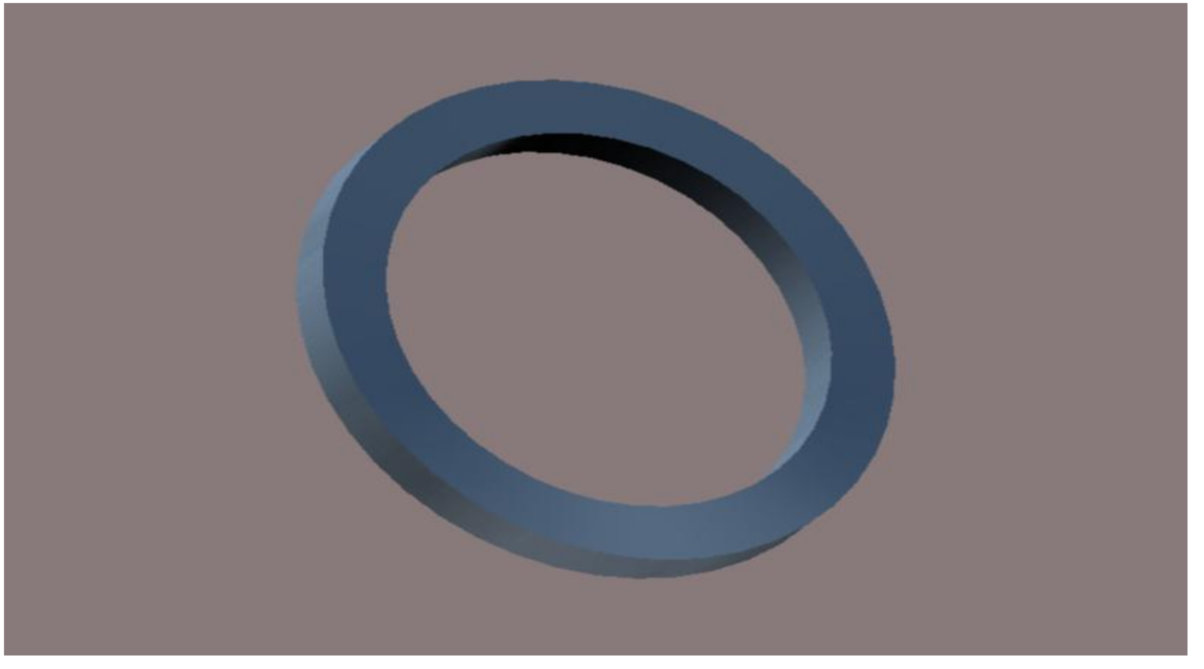
Chamber(Casing) Part Design

5. Retaining Ring:-



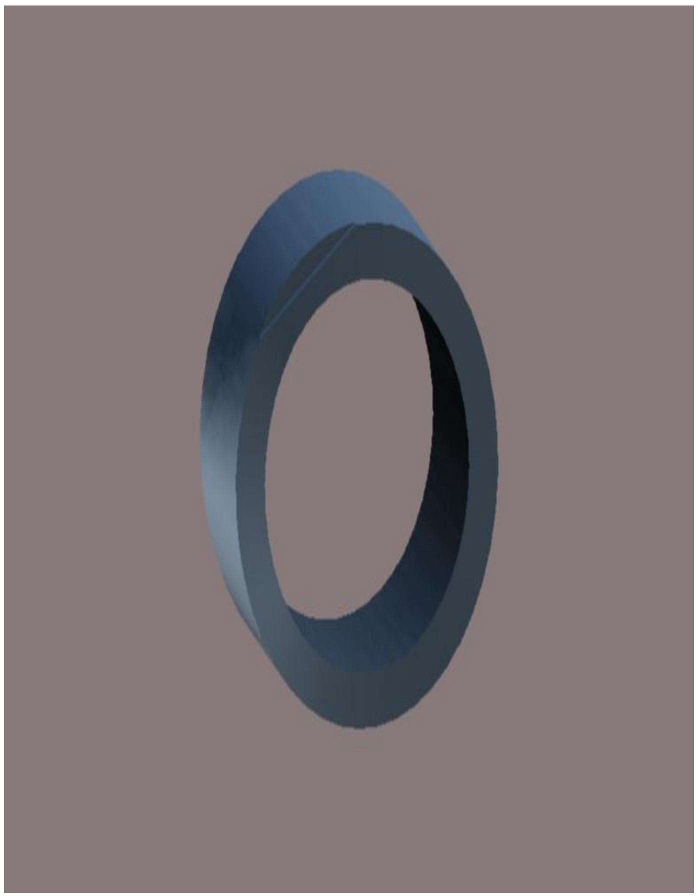
Chamber(Casing) Part Design

6. Wear Ring:-

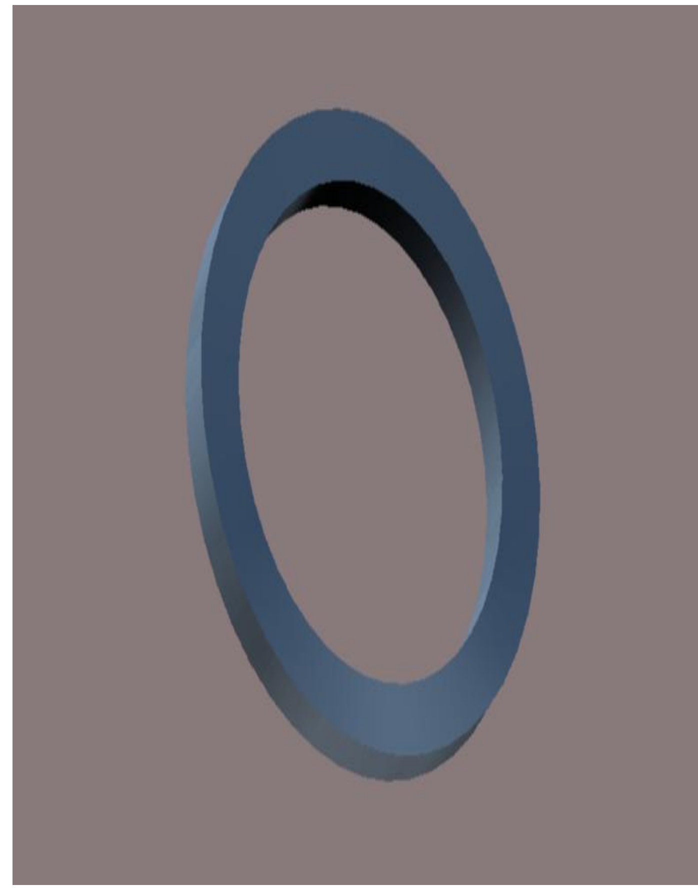


Bearing and Part Design

Bearing



Bering Rotating



Shaft Design

- **Definition**:-

"A shaft is a rotating machine element which is used to transmit power from one place to another."

- Material used for shaft

➢ It should be have high strength.

➢ It should good machinability.

➢ It should have low notch sensitivity factor.

➢ It should have good heat treatment properties.

➢ It should have high wear resistant properties.

- **Design of shaft**

➢ Shaft may be designed on the basis of

(i) Strength

(ii) Rigidity and stiffness

- **Strength**

➢ shaft subjected to twisting moment or torque only.

➢ shaft subjected to bending moment.

➢ shaft subjected to bending and twisting moment.

➢ But here shaft is subjected to twisting moment only.

$$\tau/J = T/r \quad \text{..........} \ (1)$$

Where,

T=twisting moment acting upon shaft, N.mm

J=polar moment of inertia of the shaft about the axis of rotation

τ=torsion shear stress.

r=d/2

here, $T=0.3\sigma_{el}$ or $T=0.18\sigma_u$

σ_u=ultimate tensile strength MPa.

- We know that round solid shaft's polar moment of inertia

  $J=(\pi/32)\times(d^4)$

- Equation (i) may be now be written as,

  $T/(\pi/32)*(d^4) = ꞇ/(d/2)$

  $T=(\pi/16)\times ꞇ\times d^3$ ……………….(2)

  $= (3.14\times21\times(21.6^3))/60$

  $=41.5\times10^3$ N.mm

- The twisting moment (T) may be obtained by using the following relation,
- Power transmitted by the shaft,

  $P=2\pi N\times T/60$ Kw or $T=p\times60/2\pi N$

  $P=(2\times3.14\times2960\times41.5)/60$

  P=4492.29 W

  P=4.5 Kw

- N=speed of the shaft in r.p.m

- Design of shaft on the basis of rigidity.

- Torsional rigidity:-

  Torsional deflection may be obtained by using the torsion equation.

  $T/J=G.\theta/L$

Where, θ=torsional deflection in rad.

G=modulus of rigidity for the shaft material.

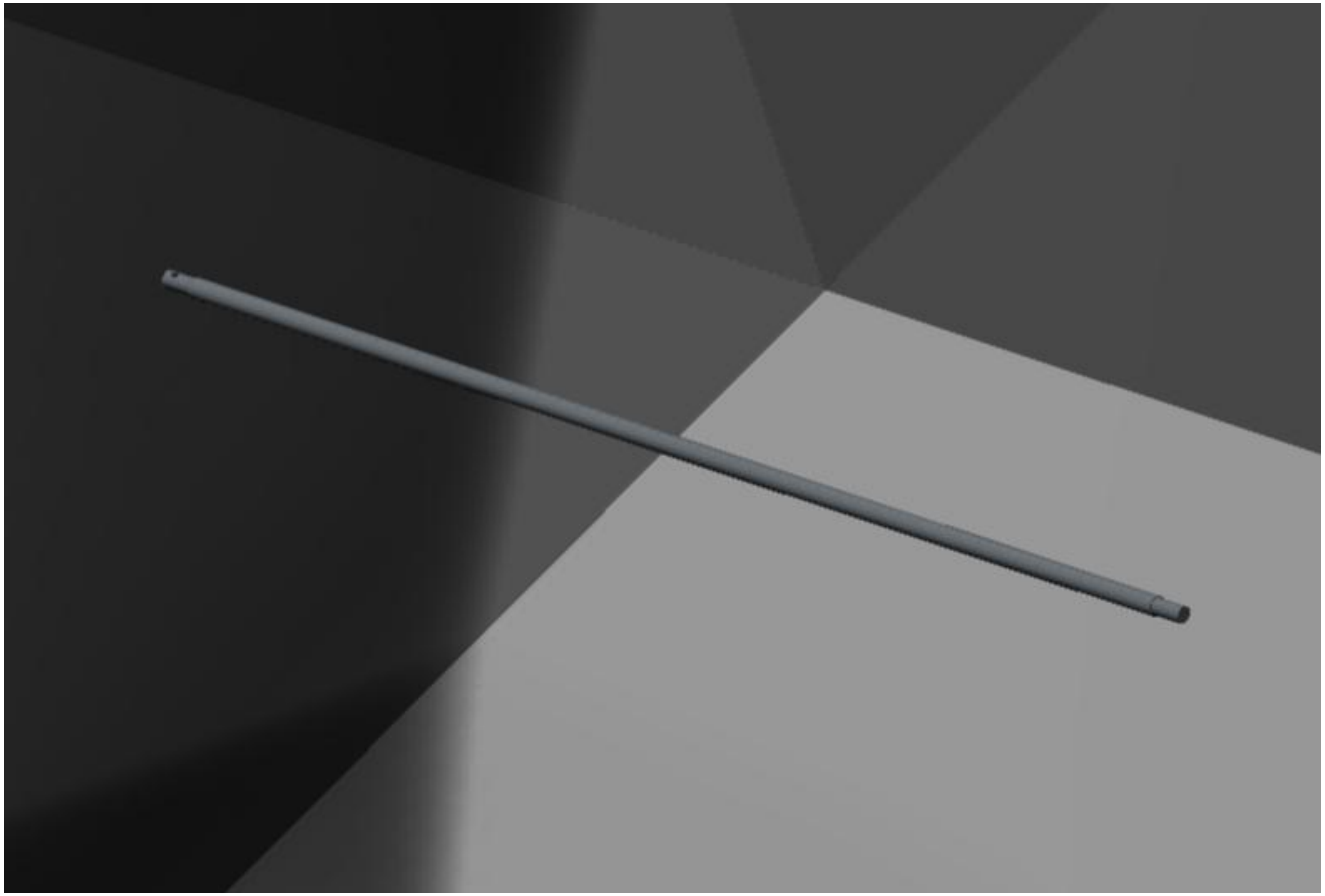
L=length of the shaft.

- Now, $\theta= (T\times L)/ (G\times J)$

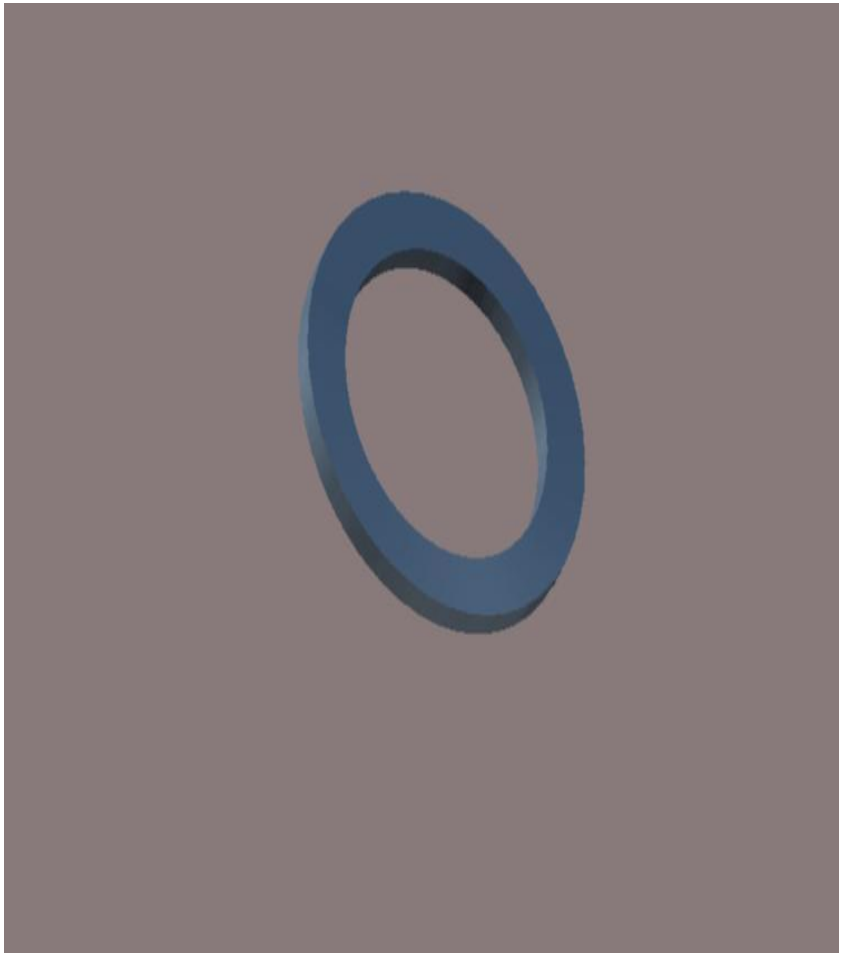
  $= (41.5\times10^3\ N.mm\times1225\ mm)/ (3.14/4\times(21.6^4 mm^4)\times84\ N/mm2)$

  $=0.0034$ rad.

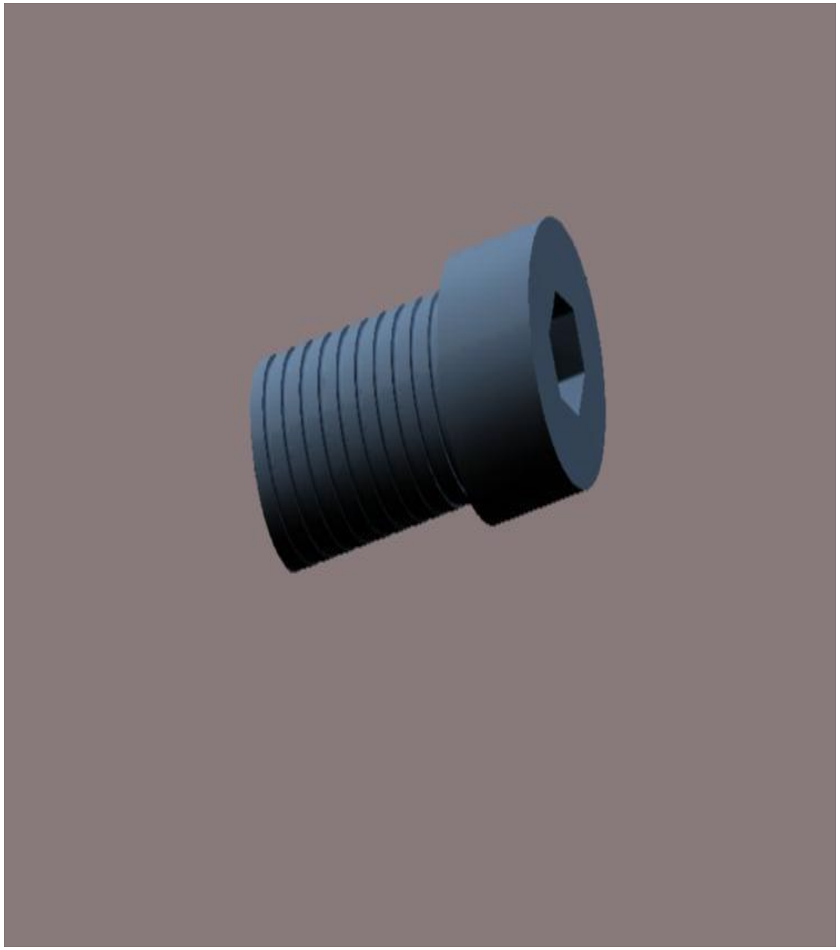
Shaft Design



Lock Washer Design

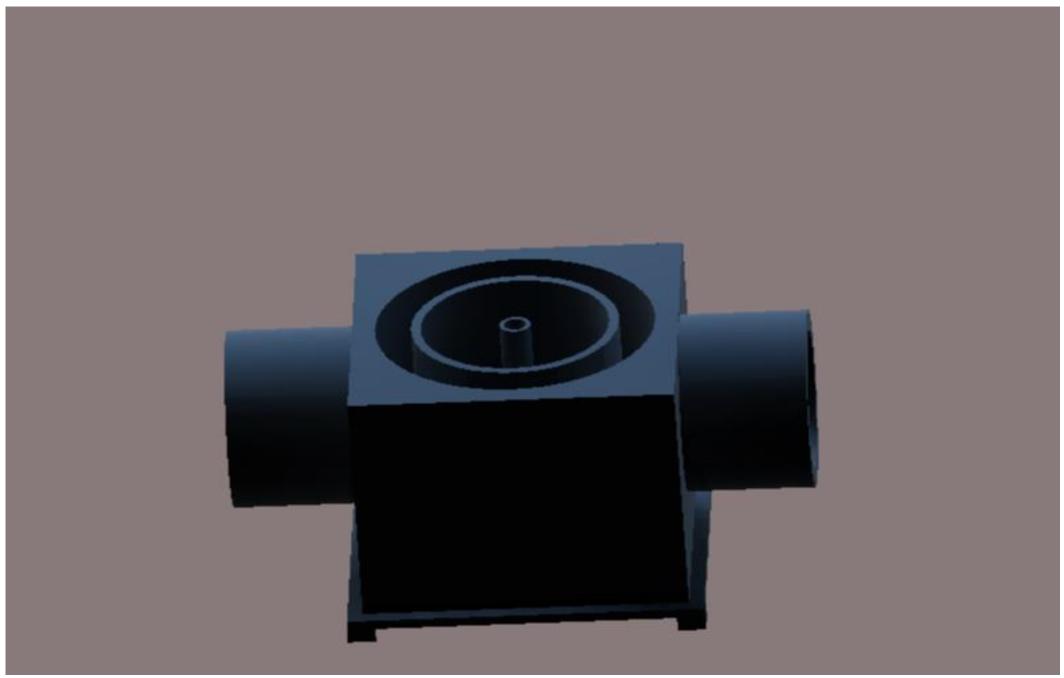


Lock Nut



Name Of Parts And Their Designs

5. Base



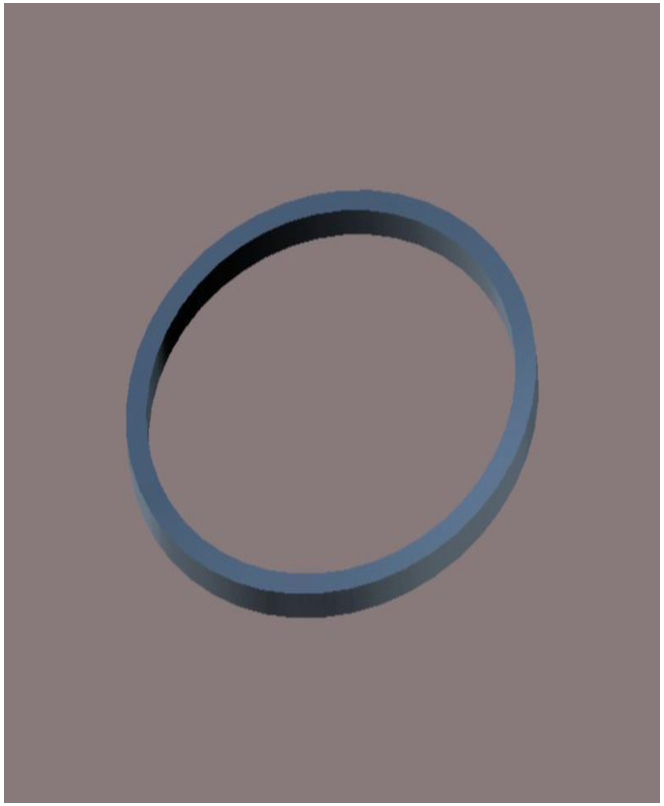
Function Of Base

- It is support the whole assembly of pump. In this pump suction and discharge are generate.
- At suction, water is entering to the pump and it is discharge in outline though the base. It is made of cast iron.
- Head
- (i) Suction head:-It is the vertical height of the free surface of liquid in sump to the centre of the pump.
- (ii) Delivery head:-It is vertical distance between the centre of pump and the water surface in the tank to which liquid is delivered.
- (iii) Static head:-It is sum of suction head and delivery head.

$$Hs = hs + hd.$$

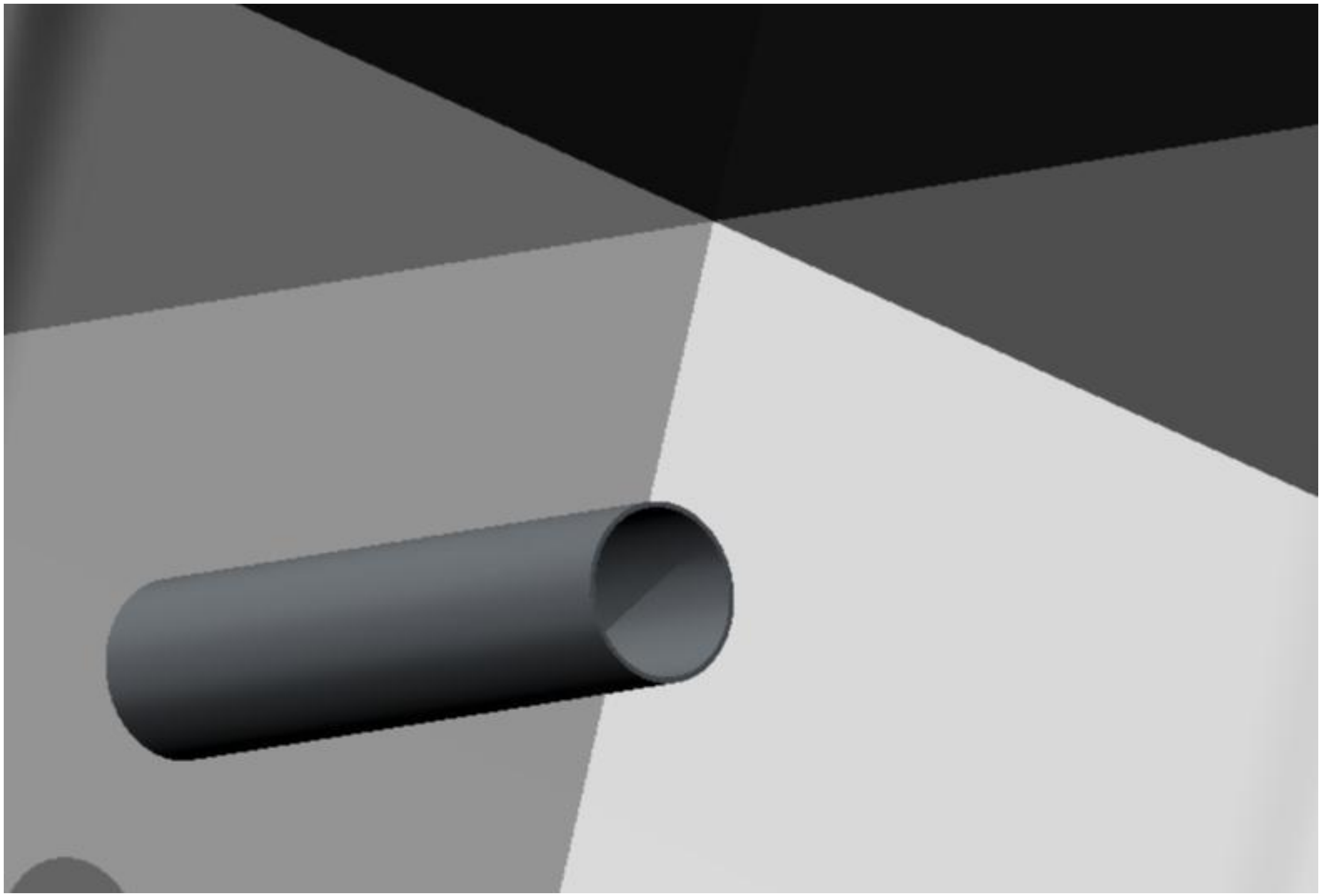
Base Part and Design

1. O-Ring



2. Drain Plug

Outer Sleeve



CFD Work

Introduction to CFD Work:-

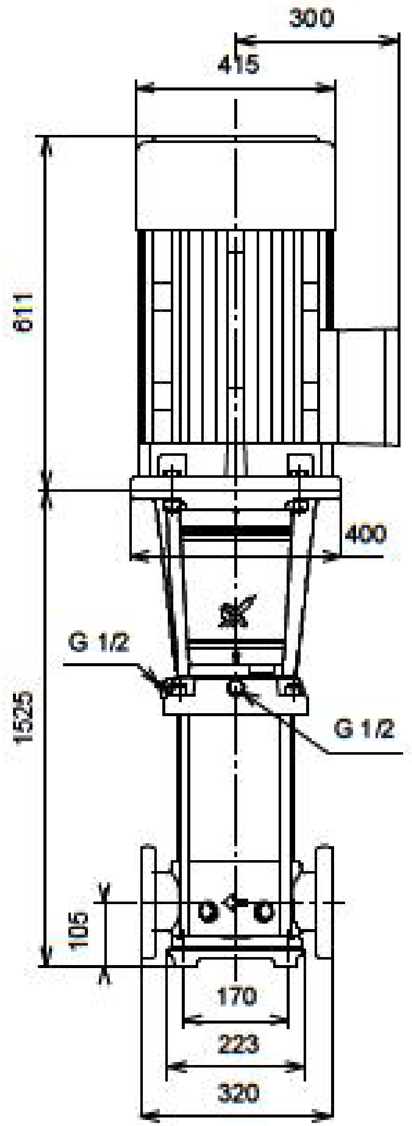
- CFD Simulation procedures for the PAT in the Ansys presents here. It would be particularly helpful for modifying or improving the existing designs of the PAT.
- Simulation facilitated to reduce cost and time of the experiment. If once PAT is simulated.
- Then it would be easy to check the performance of the PAT at any condition. Estimating the simulated performance of a PAT is an indispensable tool in design.

CFD Work

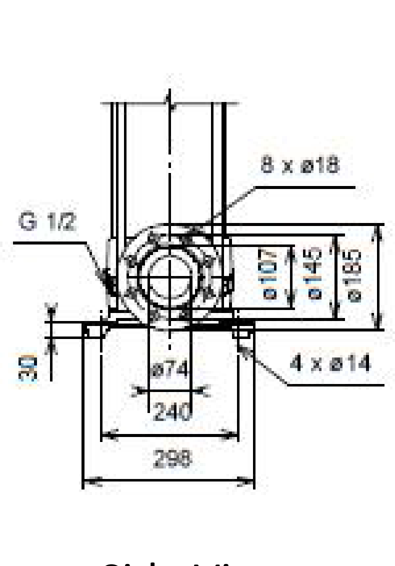
Generation Of PAT Model

- PAT model is generated with commercial available modeling software PROENGINEER 2.0 (CREO 2.0)
- PATs generated using this software is shown in Fig

Generation Of PAT Model



Front View



Side View

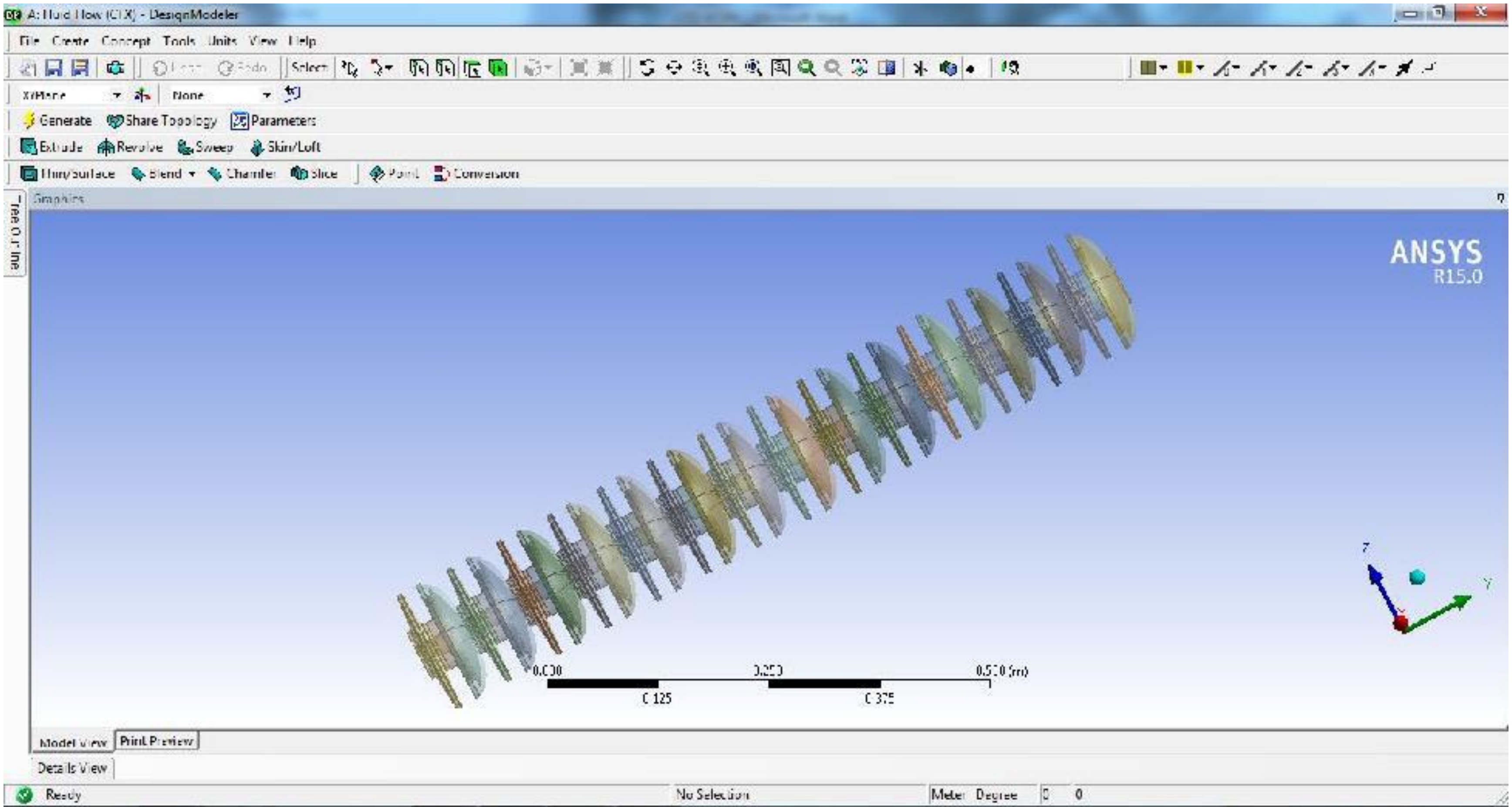
Generated PAT Models

CFD Work

Generation Of Fluid Models:-

- Fluid model is considered for numerical solution. Fluid model is having different topology compared to actual model.
- In fluid model area of interest for numerical study is only working domain of fluid.
- This fluid model is generated using software ANSYS 15.0. Fluid model for both PATs are generated using same method.

Generation Of Fluid Models



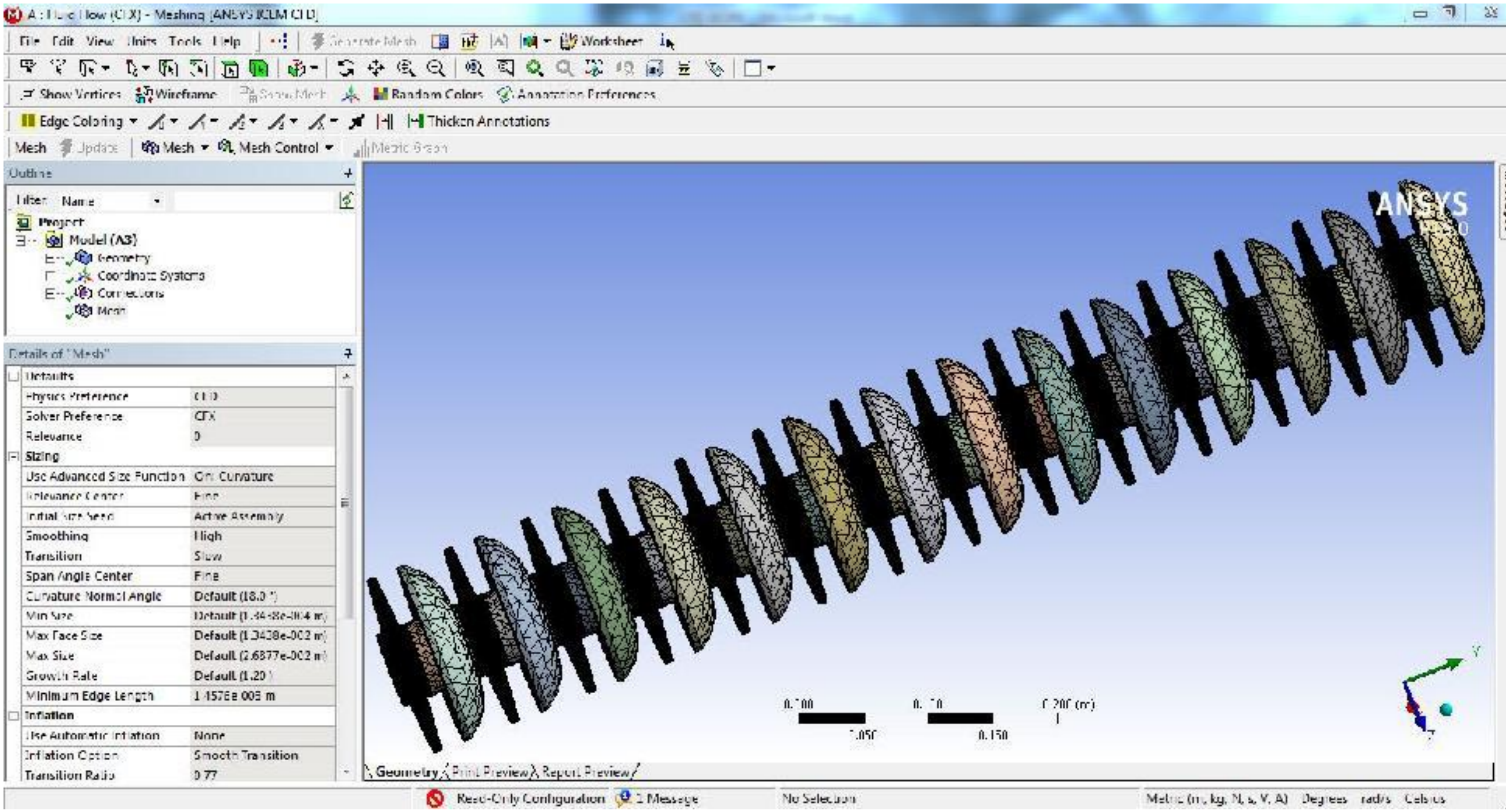
Generated Fluid Model in Ansys R 15.0

CFD Work

Meshing of Fluid Model

- In the numerical solution the working domain is divided into small sub domains. These sub domains are called as mesh. Governing equations are then discredited and solved in each of these sub domains.
- There are two types of mesh; structured and unstructured. A structured mesh is characterized by regular connectivity that can be expressed as a two or three dimensional array. This restricts the element choices to quadrilaterals in 2D or hexahedra in 3D.
- Additional classification can be made upon whether the mesh is conformal or not. An unstructured mesh is characterized by irregular connectivity is not readily expressed as a two or three dimensional array in computer memory. Compared to structured meshes, the storage requirements for an unstructured mesh can be substantially larger since the neighborhood connectivity must be explicitly stored.
- Results obtained by numerical study are dependent on mesh size and it's quality. Finer mesh size and good quality mesh leads to accurate results. Meshing of PAT model is done in ANSYS 15.0 Meshing for given PATs can be done by direct meshing and meshing by areas.
- In this study, meshing is divided into three areas of fluid model. Meshing of these areas is shown in fig

Meshing of Fluid Model

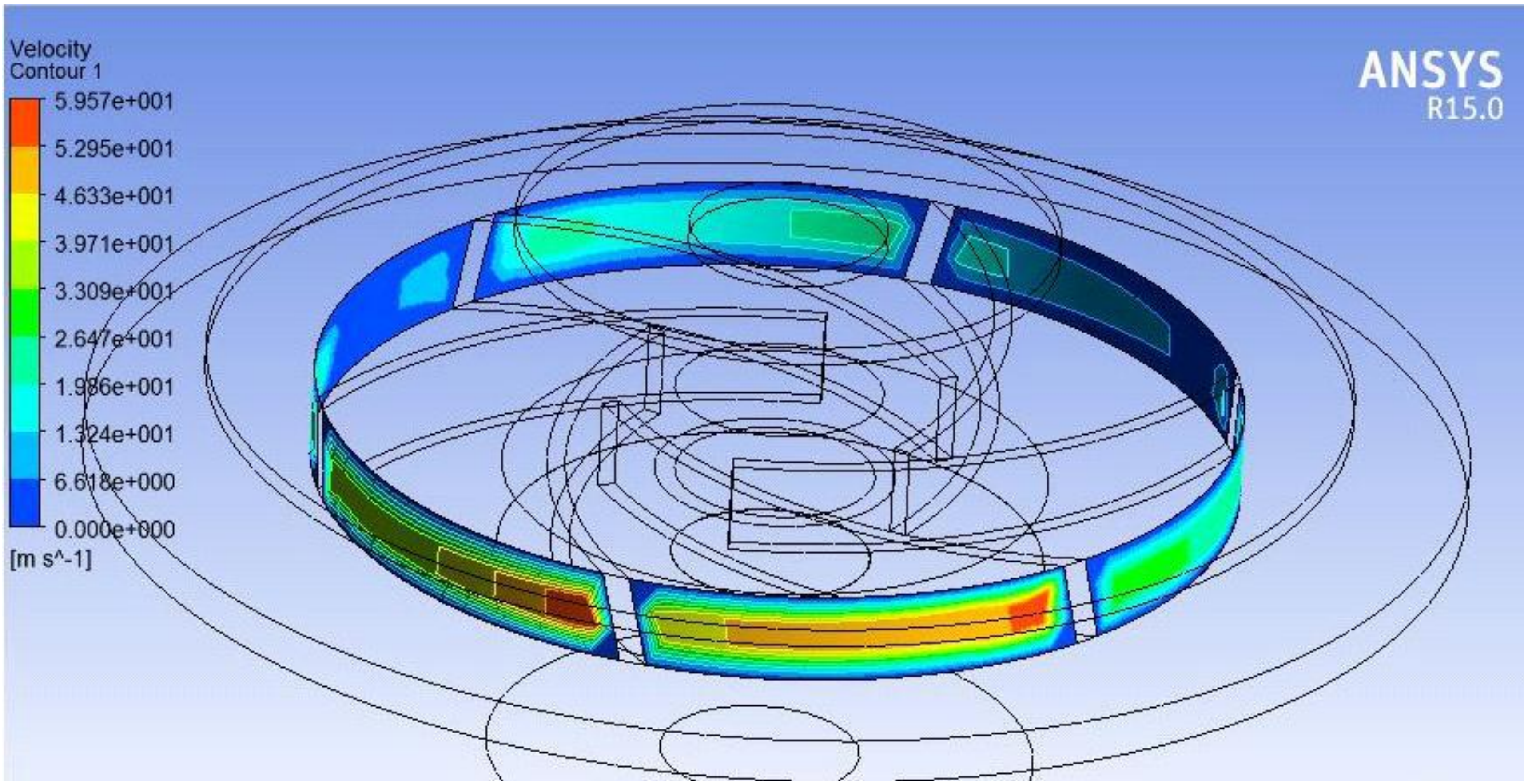


Meshing Of Fluid Model

Solution Procedure

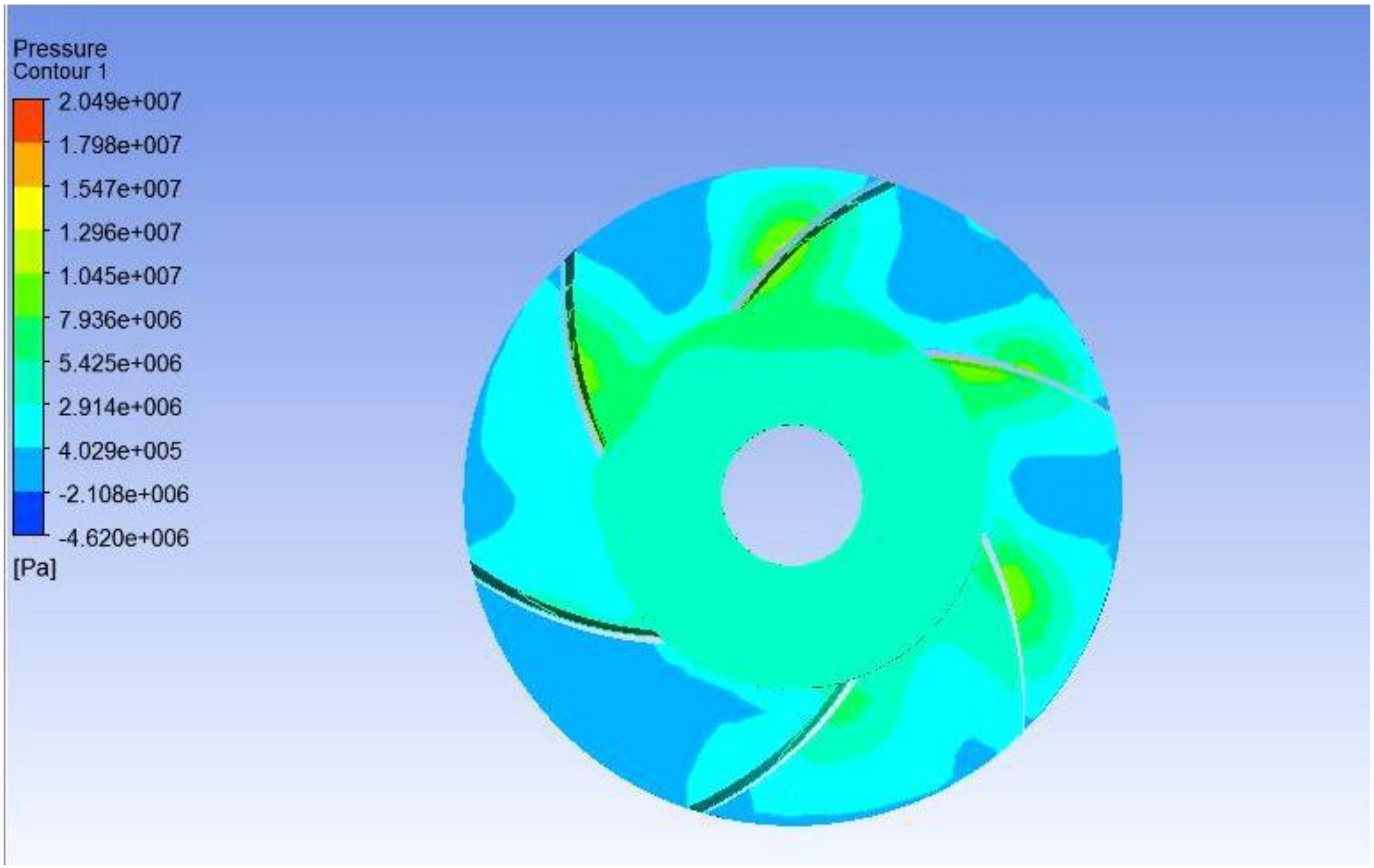
- **Step 1 Flow Analysis**
- **Analysis type**: for given fluid model steady state analysis type is considered. Define domain. Present fluid model has three fluid domains i.e. impeller, casing and outlet path . For casing and outlet path , stationary domain motion is considered.
- For impeller, rotating type domain motion is considered. No heat transfer, No slip condition and smooth wall function is considered for three fluid domains. Default settings are adopted for domain initialization.
- **Boundary conditions:** Boundary conditions are defined on inlet of casing and outlet Of draft tube.
- **Step 2 Interface Conditions**
- There are two interface conditions are defined in this section. First interface is defined in between casing and impeller.
- In this outlet of casing and inlet of impeller are interfaced to each other. In this interface impeller has rotating motion whereas casing is stationary part, so frozen rotor is considered for mixing model. Second interface is defined in between impeller and outlet. Here interfacing is done between outlet of impeller and inlet of outlet.
- Frozen rotor is considered for mixing model as impeller has rotary motion and outlet is stationary. General connection and automatic pitch change is considered for both the interfaces.

Velocity Contour on Boundary 2



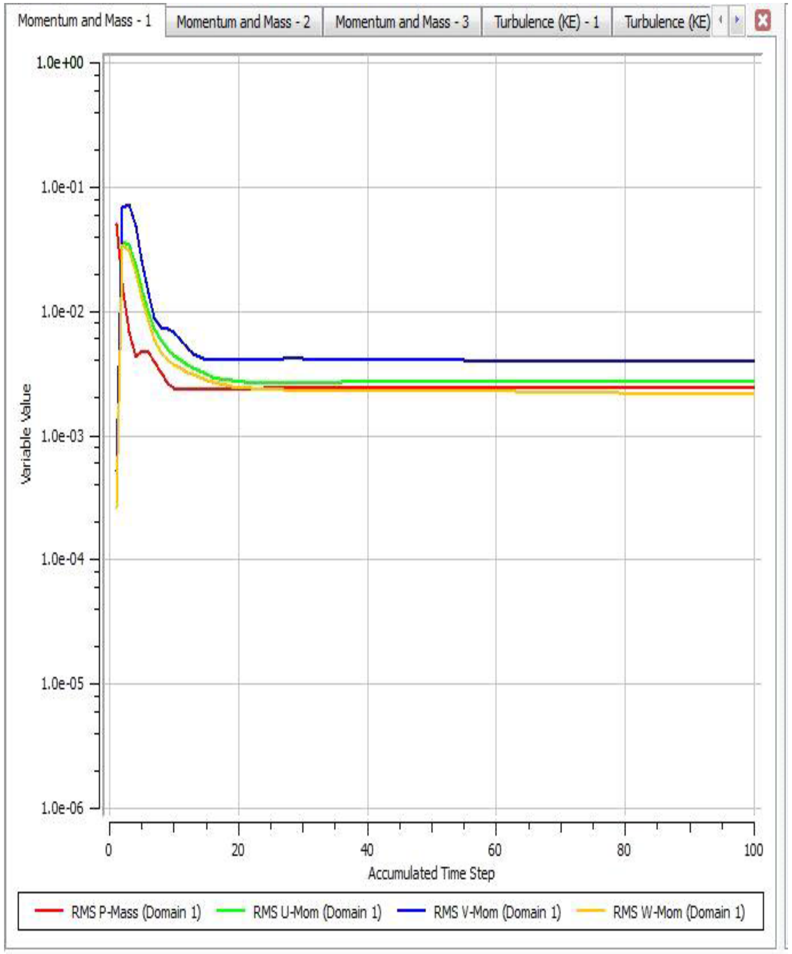
Velocity Contour on Boundary 2 in Ansys R 15.0

Pressure Contour on Domain 2

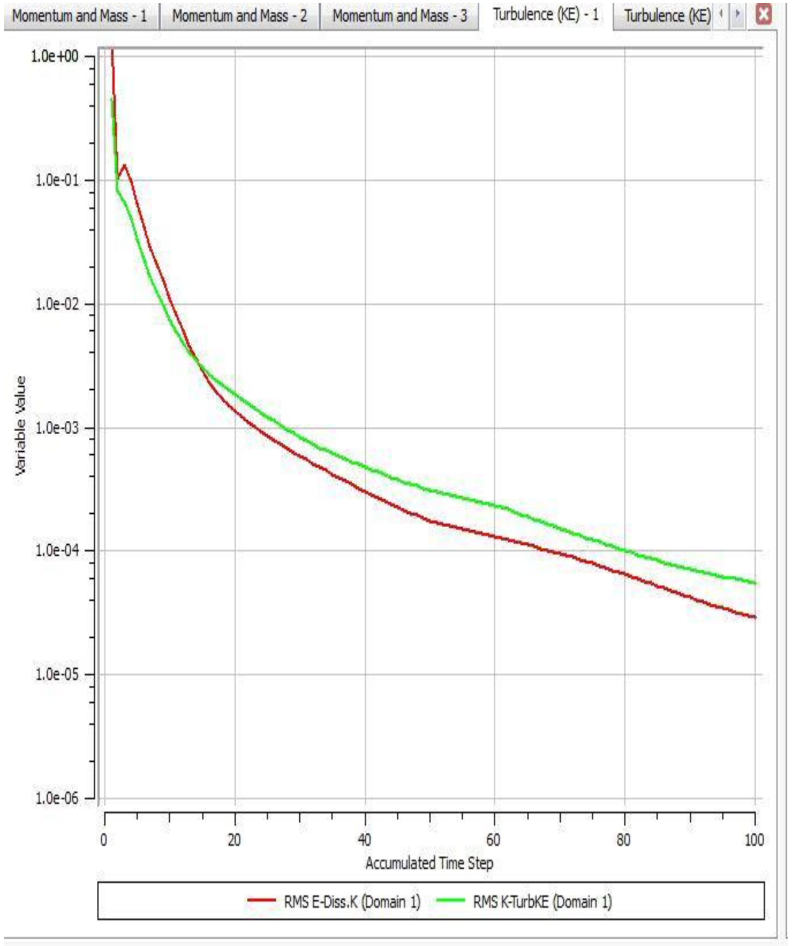


Pressure Contour on Domain 2 In Ansys R 15.0

Graphs Generated

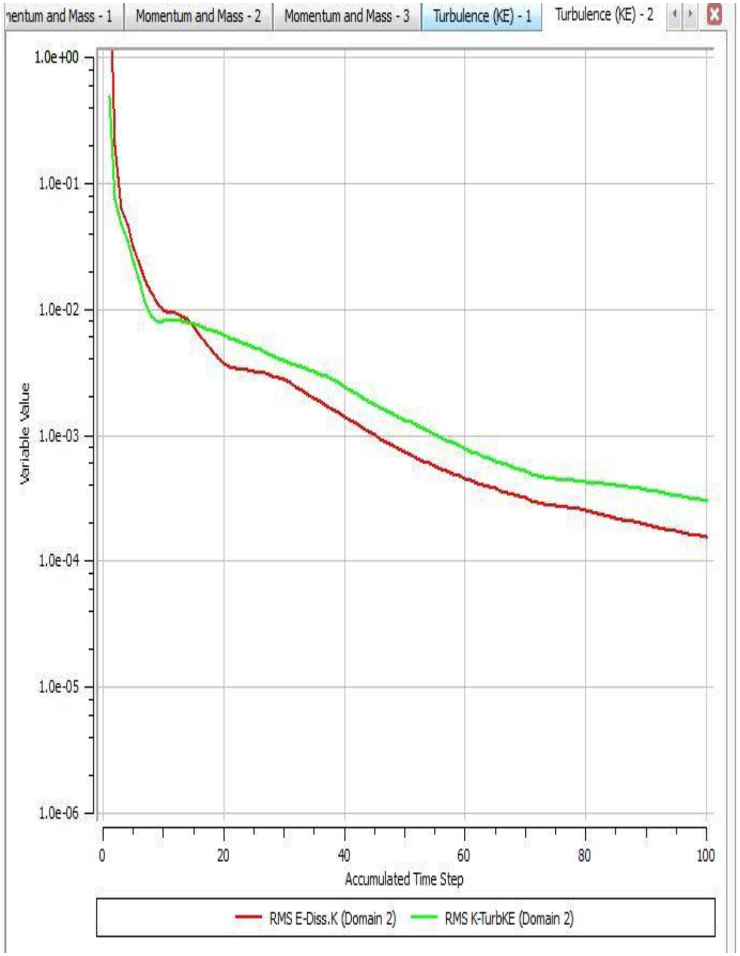


Momentum and Mass in Domain 1

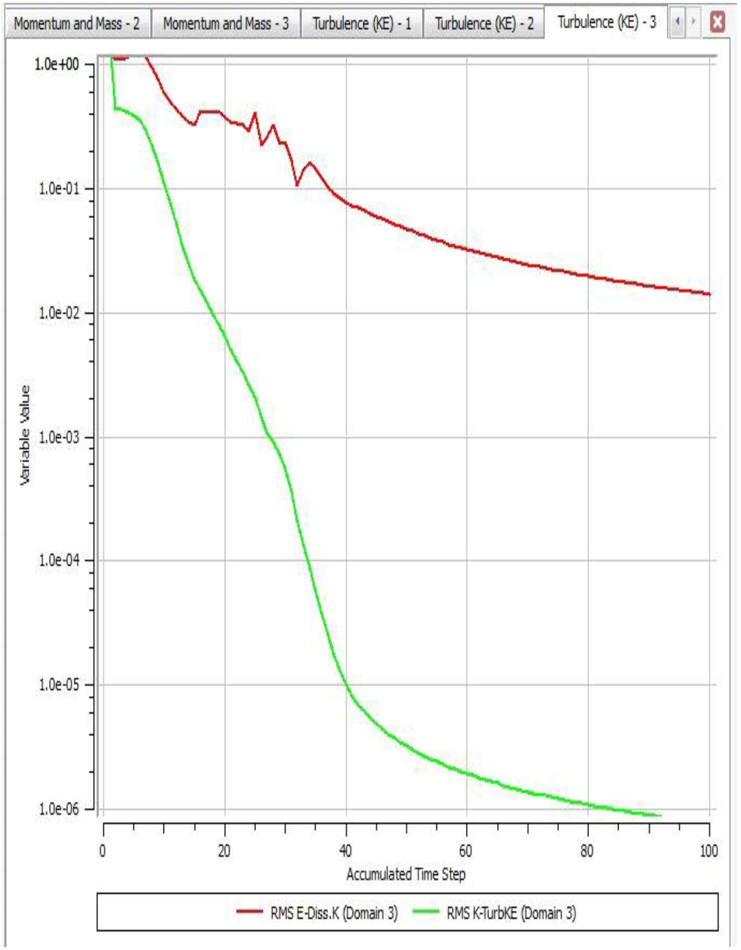


Turbulence in KE in Domain 1

Graphs Generated



Turbulence and KE in Domain 2



Turbulence in KE in Domain 3

Efficiency Calculation

- **Mechanical efficiency (ηm):-**

$\eta m = \rho(Q+q) \times (Vw2 \times u2)/P$

$= 1000\times(0.014)\times(8.25\times16.74) / (4.5\times1000)$

$= 0.25575$

$=25.575\%$

- **Manometric efficiency ($\eta mano$):-**

$\eta mano = Hm/Hi$

$= Hm/(Vw2 \times u2/g)$

$= g \times Hm/Vw2 \times u2$

$=(9.81\times15.14)$ **$/ (10.33\times16.74)$**

$= 0.8589$

$= 85.89\%$

- **Volumetric Efficiency (ηv)**

$\eta v = Q / (Q+q)$ ($q=0$)

$=1$

$=100\%$

- **Overall efficiency (ηo):-**

$\eta o = \rho g Q Hm/P$

$= (1000\times0.00834\times9.81\times15.14) / (45\times1000)$

$=0.2750$

$=$**27.50%**

Assumptions In The Project

Assumptions:-

- **Assumptions in Design (CREO Modeling):-**

1) Ignore the difficult curvature to draw in software.
2) Blades in the Casing is Ignored due to difficult in Analysis because number of domains are unnessacerily increased.

- **Assumptions in Analysis (Ansys):-**

1) Static Pressure and Outlet Pressure value is assumed same atmospheric pressure 1 bar.
2) Velocity of one stage is multiplied by number of stages and gained final outlet velocity.

Results

Results:-

- **Comparison of Working Model and New Model:-**

Working Model	New Changed Model
D1=0.06 m	D1=0.06 m
D2=0.108 m	D2=0.114 m
V=18.2857 m/s	V=26 m/s
P=30 bar	P=4.29 bar

Conclusion & Future Scope

Conclusion:-

- We change the outlet diameter of Impeller (from 108 mm to 114 mm), and then we get increment in velocity as per requirement of the boiler (from 18.2857 m/s to 26 m/s) in which the pump is utilized for its purpose is satisfied. Due to this, the Performance of boiler is increased.

Future Scope:-

- We should minimize the curvature angel and we will try to solved out of problem of blades in casing and get perfect solution in analysis.

References

- [1] Yvon Gardana, A methodology for a function-to-shape translation tool in foundry, aCMCAO/IFTS, 7 Boulevard Jean Delautre, 08000 Charleville-MeÂzieÁres, France 2000.
- [2] Tycho K. Fredericks, “An ergonomic evaluation of a manual metal pouring operation”, Department of Industrial and Manufacturing Engineering, Human Performance Institute, College of Engineering and Applied Sciences, Western Michigan University, Kalamazoo, MI 49008-5336, USA 2007.
- [3] J.C. Ferreira, Integration of reverse engineering and rapid tooling in foundry technology, Department of Mechanical Engineering, Instituto Superior Técnico, Av. Rovisco Pais, P-1049-001 Lisbon, Portugal 2002.
- [4] Ken’ichi Yano, Motion control of liquid container considering an inclined transfer path, Department of Production Systems Engineering, Toyohashi University of Technology, Hibarigaoka 1-1, Tempaku-cho, Toyohashi, 441-8580, Japan 2001.
- [5] Dr. Thoguluva Raghavan Vijayaram, FET Faculty (Engineering and Technology) MMU Multimedia University, Malaysia 2012
- [6] Yoshiyuki Noda, Precision Analysis of Automatic Pouring Machines for the Casting Industry, Department of Production Systems Engineering, Toyohashi University of Technology 1-1 Hibarigaoka, Tempaku-cho, Toyohashi, Aichi 441-8580, Japan 2008.

References

- [7] Makoto Tsuji, Automation of Die Casting, Die Casting Machine Division, Toshiba Machine Co., Ltd. 4-29-1 Hibarigaoka, Zama-shi, Kanagawa 228-0003, Japan 2008.
- [8] Rhythm-Wadhwa, Electromagnet Gripping in Iron Foundry Automation Part I:Principles and Framework, Department of Production and Quality Engineering, NTNU Trondheim, 7051, Norway 2007.
- [9] Dr. O. P. Khanna, A Text book of foundry technology, 1996, Ch:1, pp. 5-6
- [10] Dr. O. P. Khanna, A Text book of foundry technology, 1996, Ch:1, pp. 6-7
- [11] Dr. O. P. Khanna, A Text book of foundry technology, 1996, Ch:1, pp. 7
- [12] Dr. O. P. Khanna, A Text book of foundry technology, 1996, Ch:1, pp. 7-9
- [13] Noopur Enterprise, Manufacturer of all grade of ferrous metals, www.nainitagroup.com
- [14] Dr. O. P. Khanna, A Text book of foundry technology, 1996, Ch:2, pp. 46-47
- [15] R. K. Jain, Production technology, 2009, 17th Edition, Ch:3, pp. 85
- [16] S.K. Hajra Chaudhary, Elements of workshop technology, vol:1, 2007, 14th Edition, Ch:11, pp. 307

References

- [17] P. L. Jain , Principles of foundry technology, 2003, 4^{th} Edition, Ch: 2, pp. 6-15
- [18] P. N. Rao, Manufacturing technology, vol:1, 2010, 3^{rd} Edition, Ch:3, pp; 60-66
- [19] S.K. Hajra Chaudhary, Elements of workshop technology, vol:1, 2007, 14^{th} Edition, Ch:11, pp. 322
- [20] T. V. Ramana Rao, Metal casting, 1996, 1^{st} Edition, Ch:2, pp. 21-26
- [21] R. K. Jain, Production technology, 2009, 17^{th} Edition, Ch:3, pp. 91
- [22] A. K. Chakrabarti, Casting technology & cast alloys, 2005, Ch:2, pp. 6
- [23] Dr. O. P. Khanna, A Text book of foundry technology, 1996, Ch:9, pp. 152-153
- [24] P. N. Rao, Manufacturing technology, vol:1, 2010, 3^{rd} Edition, Ch:4, pp. 117-128
- [25] Dr. O. P. Khanna, A Text book of foundry technology, 1996, Ch:11, pp. 220-234
- [26] Dr. O. P. Khanna, A Text book of foundry technology, 1996, Ch:12, pp. 238
- [27] Electronic circuit by millman j. & taub H. Tata Mc Graw Hill
- [28] J. S. Katre, A Basic Electronics Tech-max Publication, 2009,ch:4,pp.4-1 to 4-5
- [29] Digital Logic design by Morris Mano
- [30] J. S. Katre, A Basic Electronics Tech-max Publication, 2009,ch:6,pp.6-2 to 6-10
- [31] R. B. Patil, Computer integrated manufacturing Tech-max Publication, 2009,ch:2, pp.2-40 to 2-45

Thank You...