INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

ISSN: 2454-132X
Impact Factor: 6.078
(Volume 7, Issue 3 - V7I3-1938)
Available online at: https://www.ijariit.com

Analytical study of impeller blade using Ansys

Rutvik Sadanand Lagali
rutvik.21810903@viit.ac.in
Vishwakarma Institute of Information Technology, Pune, Maharashtra

ABSTRACT

FEM stands for finite element method. It is a numerical technique used to perform finite element analysis (FEA) of any given physical phenomenon. FEM had made modelling very easy with more detailing, accuracy, less time consuming, etc. It has many applications which includes stress analysis, slope stability analysis, analysis of dams, etc. Finite Element Analysis software, that is, FEA software, is a common tool used, today. 3D CAD, SolidWorks CAM, Composer, Electrical 3D, Electrical Schematics, Inspection, Simulation and Ansys are some of the best FEA software. In this paper, we are going to analyse total deformation and equivalent stress on the Impeller Blade. Model of an impeller blade was modelled on CATIA (stands for Computer-Aided Three-dimensional Interactive Application) and analysed on Ansys.

Keywords— *Ansys, Structural Analysis, Rotational Velocity, Thermal Conduction*

1. INTRODUCTION

Impeller blade is an important part of a centrifugal pump. It is a rotor used to increase the pressure and flow of fluid. It is opposite to the turbine. There are three types of impeller blades: open impeller, semi-open impeller and closed impeller. In 1839, Andrews introduced the proper volute casing and in 1846, used a fully shrouded impeller. Johnson was the first person to construct a three-stage centrifugal pump in the same year, that is, 1846. After three years, that is, 1849, Gwynne constructed a multi-stage centrifugal pump and began the first systematic examinations of these pumps. In 1960, Martin Stable (founder of Hidrostal) invented the patented single screw centrifugal impeller. Thus, this is the model of impeller blade. It is a semi-open impeller.

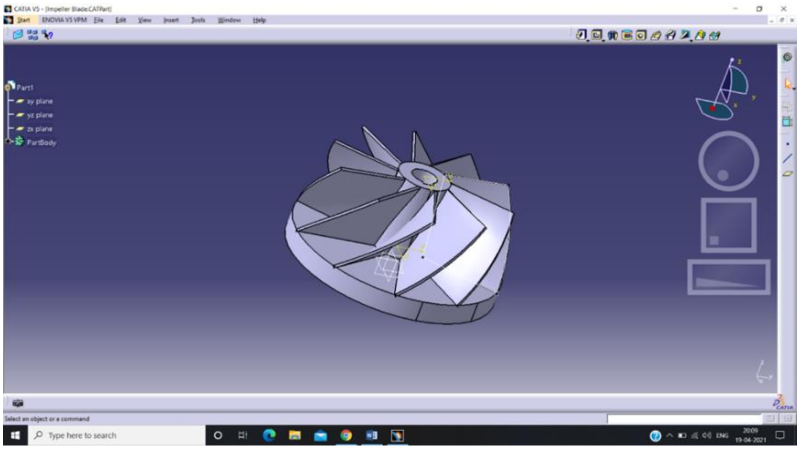

Fig 1: CATIA model of impeller blade

2. METHODOLOGY

2.1 Modelling

Purpose of this paper is to design and analyse the impeller blade. Modelling is done on CATIA V5R20. A circle is drawn of *80mm* of diameter and pad for *8 mm*. Again, another circle is drawn from the same centre of *20mm* of diameter and pad for *30 mm*. By using **helix curve definition,** a curve is drawn vertically on *Z axis* with *250mm* **type patch**, *30mm* **height**, *10 degree* **tape angle** and *inward* **way**. By using line definition vertical drawn curve is joined and a closed figure is constructed. By using **multi-sections surface definition,** the closed figure is turned into a curved sheet and by using **thick-surface definition**, the curved sheet is made 1 mm thick. Now, multi-section surface is hidden from part body. By using **circular pattern definition**, *complete crown* **parameters**, *10* **instances** and **reference element** *Z-axis.* At last a circle is drawn on the top of the model of *10mm* diameter and half rectangle is drawn at the outside of the circle by touching the curve of the circle. The curve of the circle is erased between the half-rectangle. Formed new structure is pocket till the end. This figure is formed.

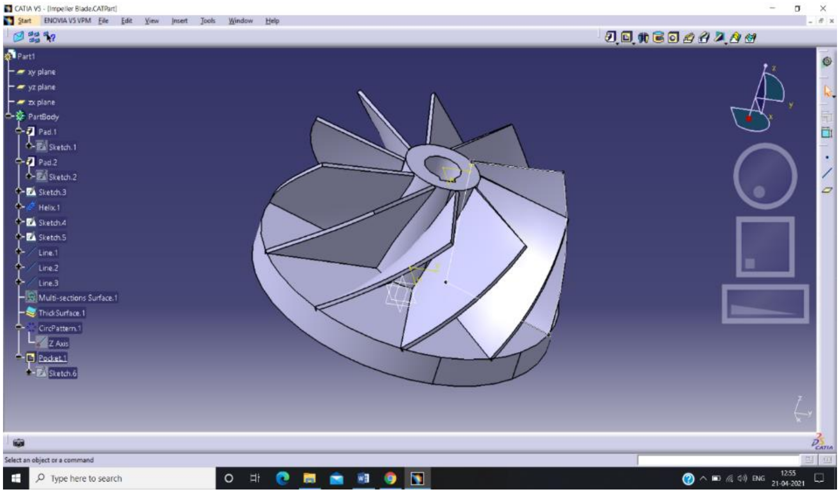

Fig 2: CATIA model of impeller blade with part body

2.2 Properties, Applications and Material

Impeller blade increases the velocity and the pressure and also directs it towards the outlet of the pump. Impeller blades are used in centrifugal compressors, water jets, agitated tanks, washing machines, firefighting rank badge, air pumps, etc. Stainless steel is used for making this model, because it is very easy to clean, low maintenance and resistant to corrosion and most chemical reactions.

3. ANALYSIS

Analysis is done by using ANSYS 2021 R1. The first model is made on the CATIA. **Static structural** is chosen from the **ANSYS system**. *Stainless steel* is chosen from the list of materials from **engineering data**. From **geometry**, the model is imported. Imported model is opened through **model**.

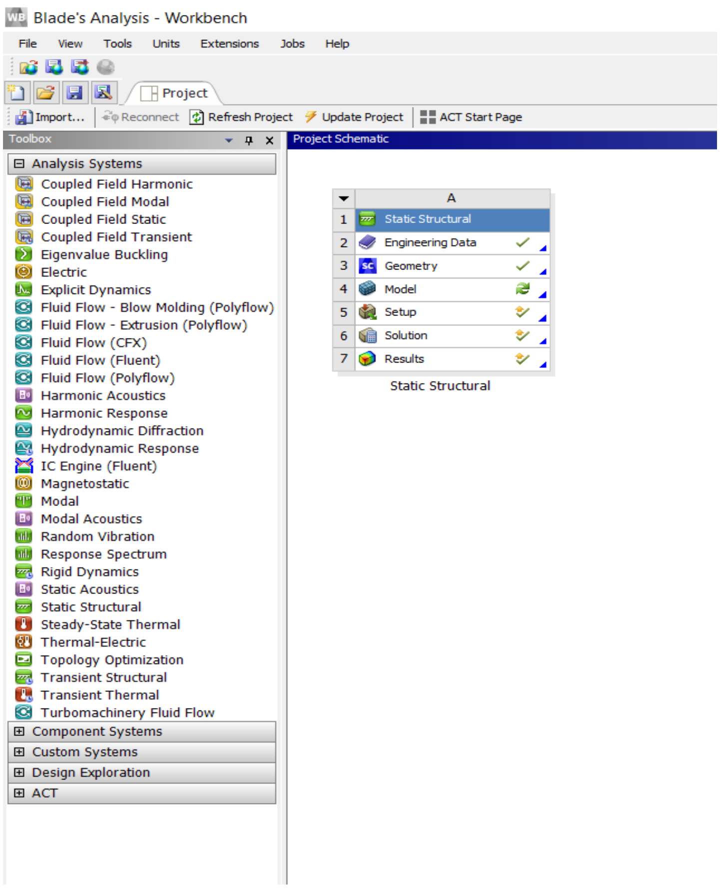

Fig 3: Static structural

1) Then geometry is opened in another window.

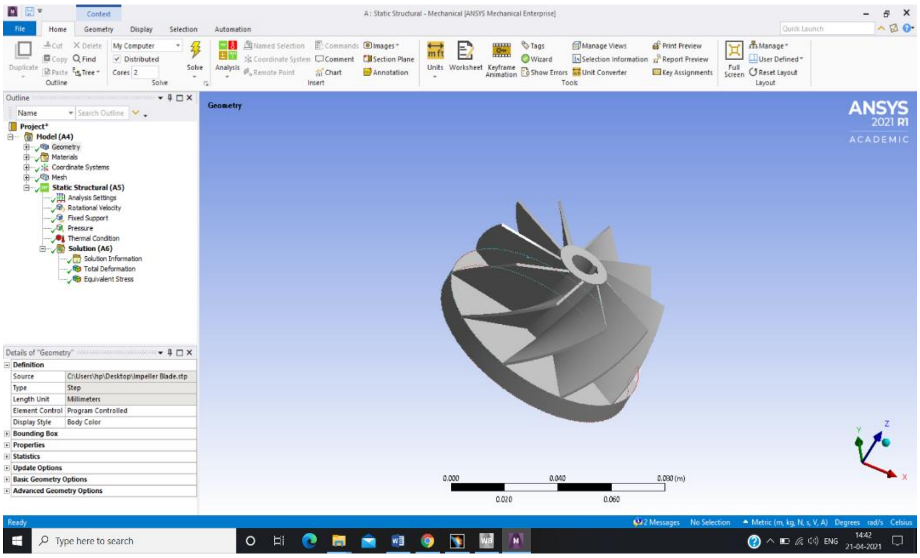

Fig 4: Geometry is opened in ANSYS

2) Again, material is chosen, *stainless steel*.

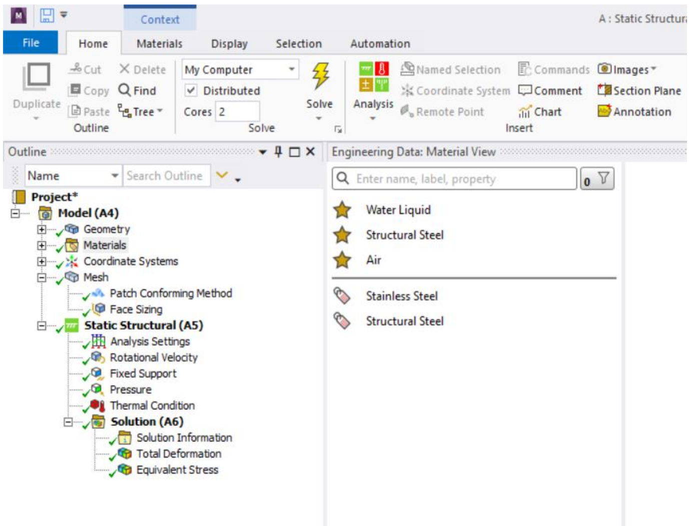

Fig 5: Materials

3) Coordinate system is selected.

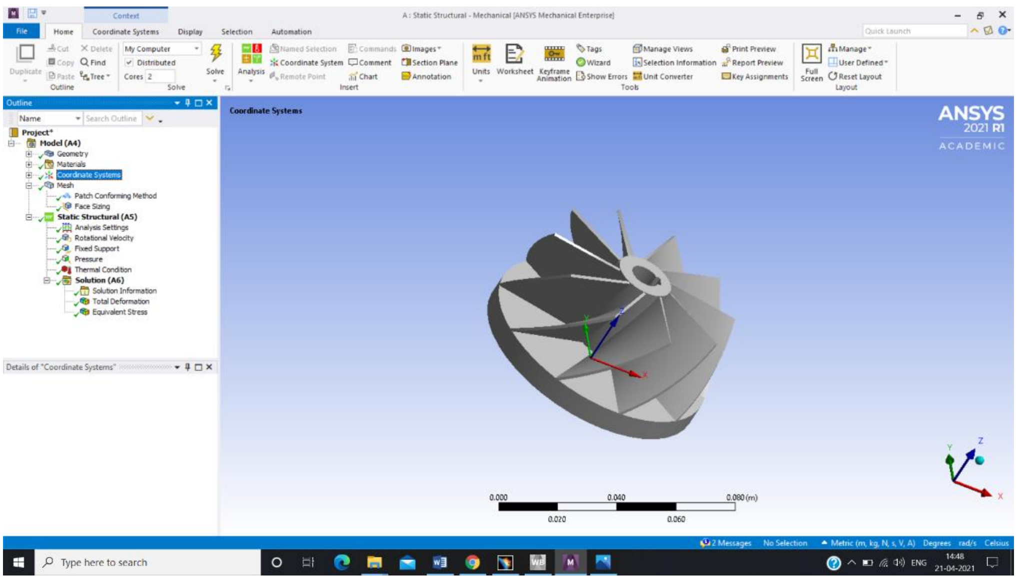

Fig 6: Coordinate system

4) Under mesh, in path conforming method, **scoping method** is *geometry selection* and **method** is *tetrahedrons* and **geometry** is *1 body*.

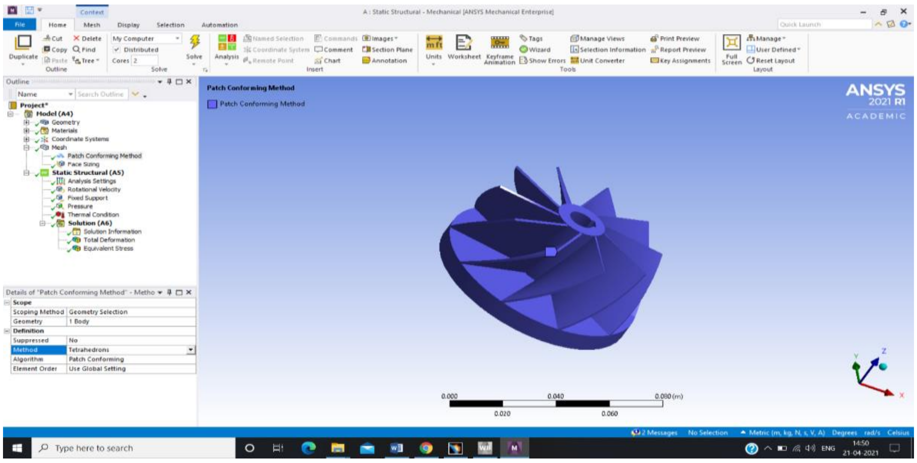

Fig 7: Path conforming method

Scoping method	Geometry	Supressed	Method	Algorithm	Element order
Geometry selection	1 body	No	Tetrahedrons	Patch confirming	Use global setting

5) Under mesh, path facing is done. Scoping method is geometry selection, geometry is 4 faces and element size is 5.e-003 m.

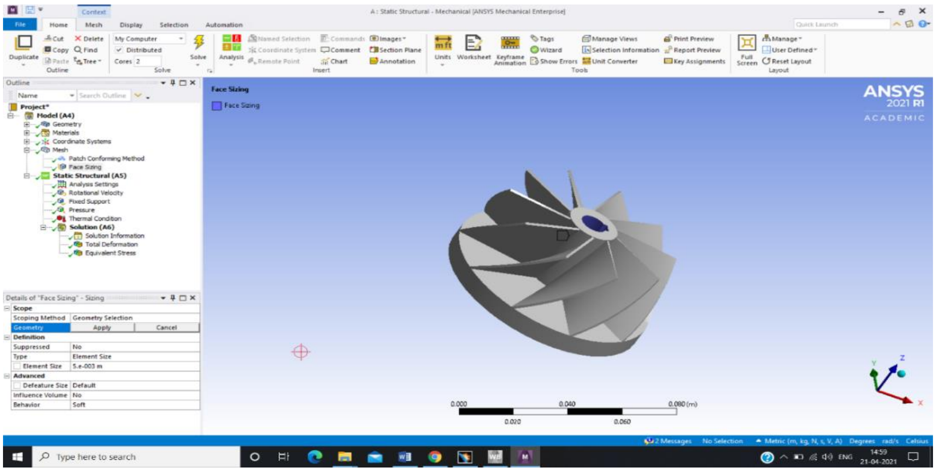

Fig 8: Face sizing

Scoping method	Geometry	Supressed	Type	Element size	Defeature size	Influence volume	Behaviour
Geometry selection	4 faces	No	Element size	5.e-0003 m	Default	No	Soft

6) Meshing is done.

- Linear

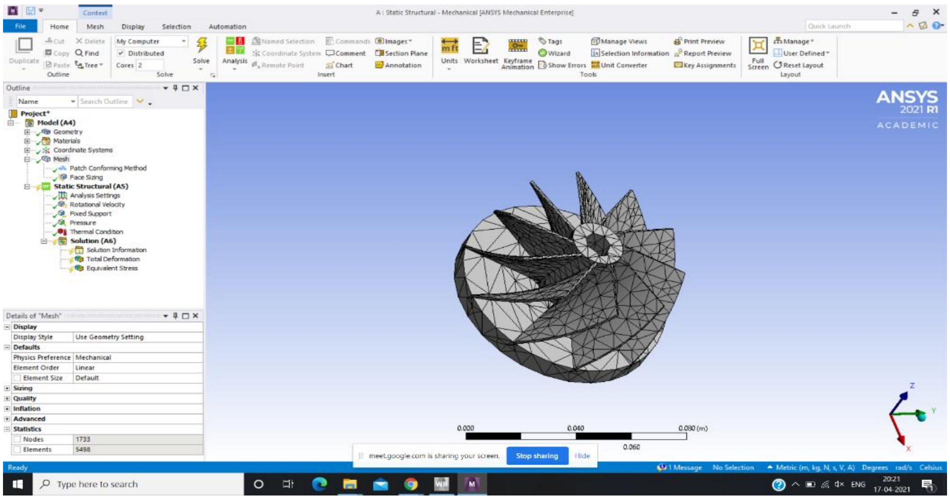

Fig 9: Linear

Nodes	1733
Elements	5498

- Quadratic

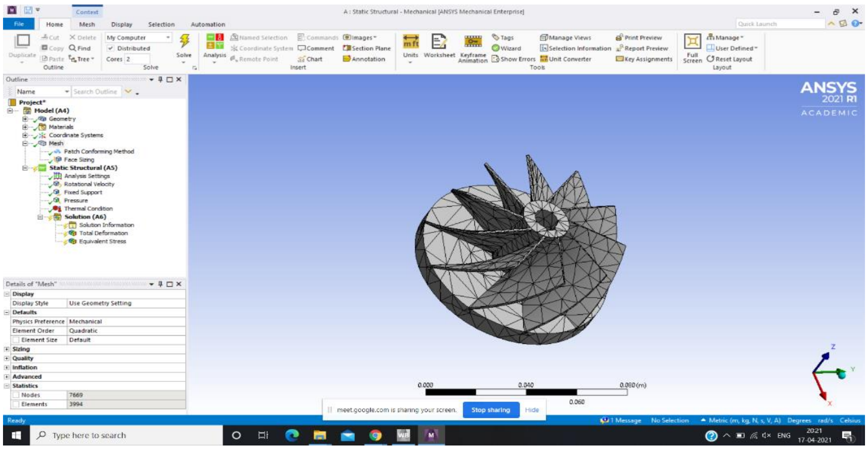

Fig 10: Quadratic

Nodes	7669
Elements	3994

- Program controlled

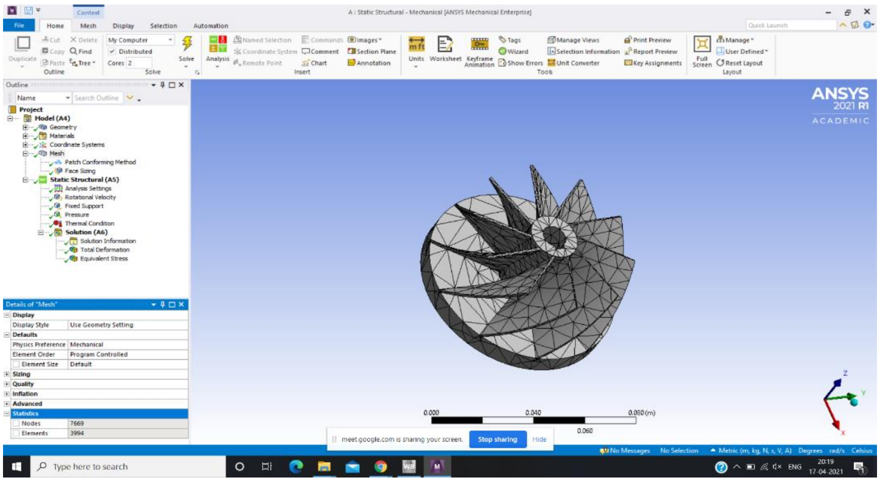

Fig 11: Program controlled

Nodes	7669
Elements	3994

7) In static structural, boundary conditions are added.
   - Rotational velocity is added. (12566 rad/s or 200 FPS) (Geometry is all bodies)

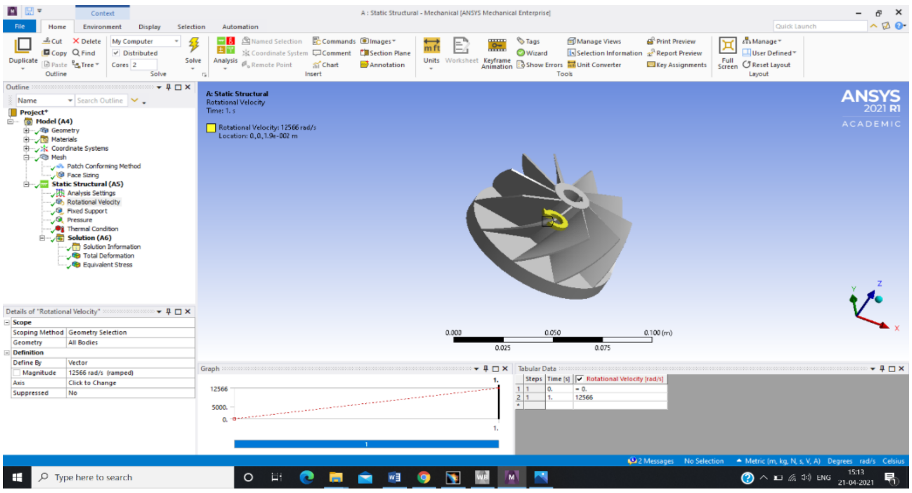

Fig 12: Rotational velocity

Scoping method	Geometry	Defined by	Magnitude
Geometry selection	All bodies	Vector	12566 rad/s

- Fixed support is added. (Geometry is 16 faces)

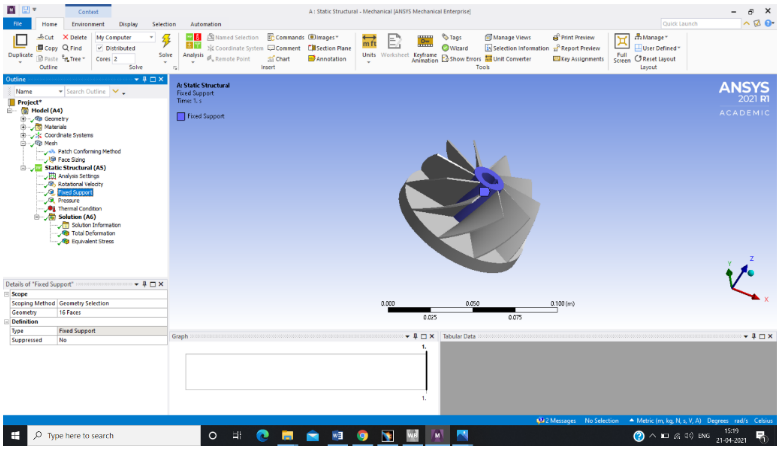

Fig 13: Fixed support

Scoping method	Geometry
Geometry selection	16 faces

- Pressure is added. (57380 Pa) (Geometry is 16 faces)

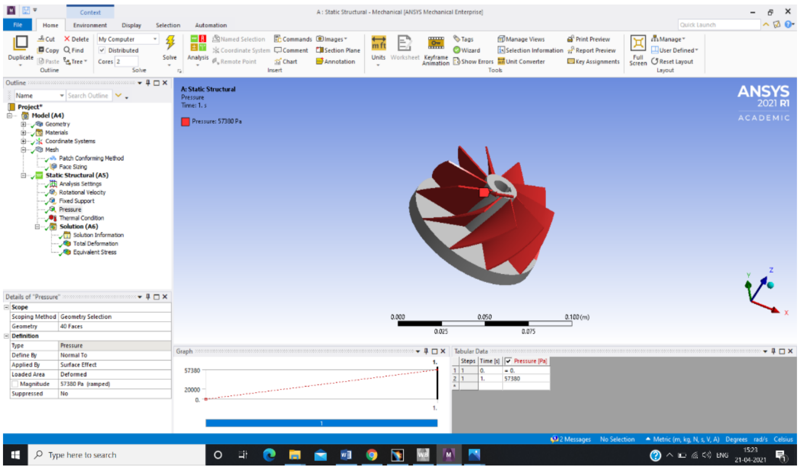

Fig 16: Pressure

Scoping method	Geometry	Type	Defined by	Applied by	Loaded area	Magnitude
Geometry selection	16 faces	Pressure	Normal to	Surface effect	Deformed	57380 Pa

- Thermal condition (22 degree C) (Geometry is 1 body)

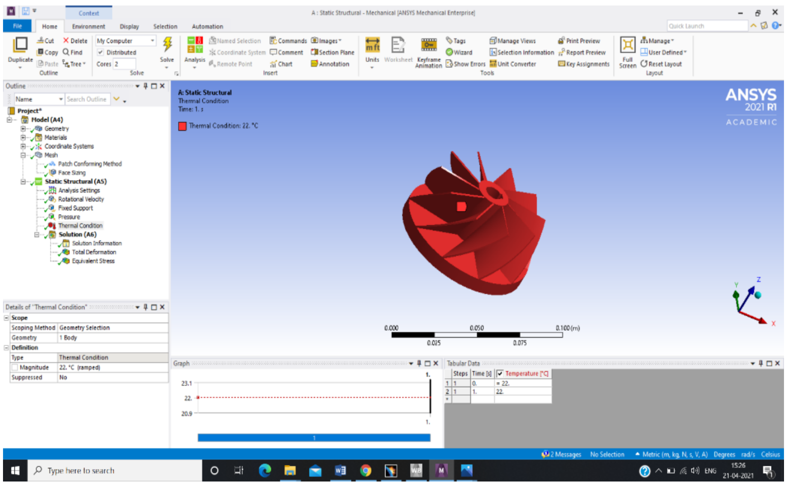

Fig 17: Thermal condition

4. RESULT

1) Total deformation

Minimum	Maximum	Average
1. m	9.3915e-004 m	1.197e-004 m

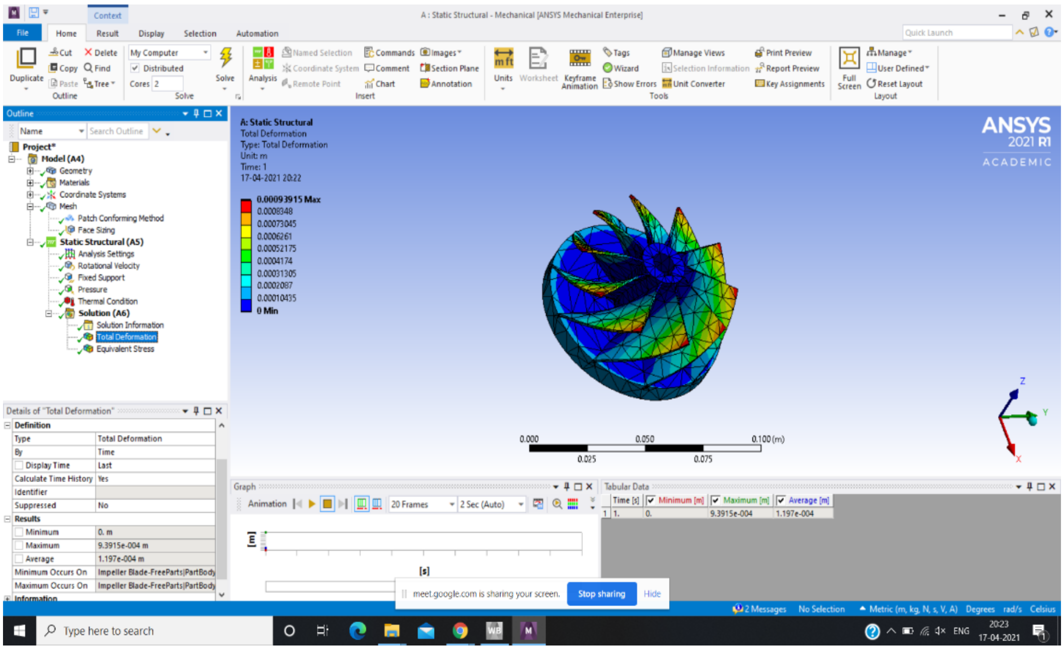

Fig 18: Total deformation

2) Equivalent stress

Minimum	Maximum	Average
1.1215e+005 Pa	2.808e+009 Pa	4.5947e+008 Pa

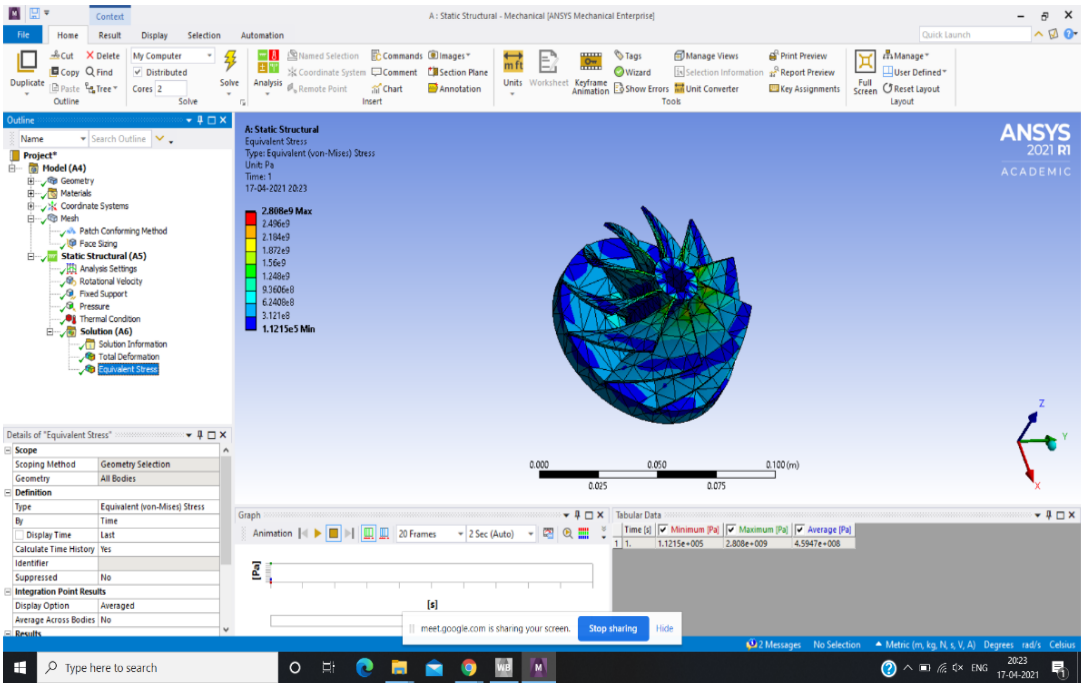

Fig 19: Equivalent stress

5. DISCUSSION

Analysis of an impeller blade is done. Linear, quadratic and program controlled meshing is done. Four boundary conditions are added, rotational velocity, fixed support, pressure and thermal condition. Total deformation and equivalent stress is found.

6. CONCLUSION

After meshing and boundary conditions, we get the total deformation and equivalent stress.

6.1 Total deformation

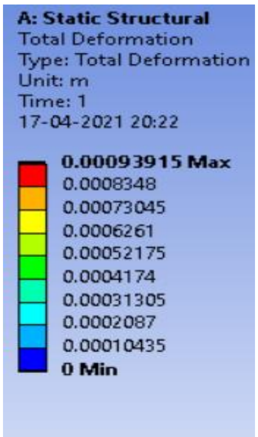

Fig 20: Total deformation

6.2 Equivalent stress

Fig 21: Equivalent stress

7. REFERENCES

[1] https://www.researchgate.net/publication/325284112_Analysis_of_Centrifugal_Pump_Impeller_Using_ANSYS
[2] https://en.wikipedia.org/wiki/Impeller
[3] https://www.hidrostal.co.uk
[4] https://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT1998/78668/V005T16A001/4219162/v005t16a001-98-gt-022.pdf
[5] https://www.youtube.com/watch?v=5rH5HmYsfnE&list=LL&index=4&t=1s
[6] https://www.youtube.com/watch?v=ekua7Uf3Ltg

