Journal of Advances in Science and Technology
Vol. 19, Issue No. 1, March-2022, ISSN 2230-9659

Overview on Analysis of Machine Element Failure using Solidworks

Awatade R. S.[1]* Magar R. R.[2] Devkule D. N.[3] Kothawale S. M.[4] Shete Y. S.[5]

[1,2,3,4] Student, Department of Mechanical Engineering, Sahakar Maharshi Shankarrao Mohite Patil Institute of Technology and Research, Akluj, Maharashtra, India

[5] Lecturer, Department of Mechanical Engineering, Sahakar Maharshi Shankarrao Mohite Patil Institute of Technology and Research, Akluj, Maharashtra, India

Abstract – Uses of computer application in engineering design become a very common now a day. It is very obvious to use the different software for performing design operations in various sectors. There are few much popular software are available such as AutoCAD, PTC Creo, Ansys, Solidworks, Solid Modelling and so all. In this paper, author has been focused on the design and analysis of the engine component using Solidworks. Solidworks is one of the most trending software now a day. This software allows design as well as simulation and Modelling by itself. The aim of this paper is to discuss the procedure for modelling different kind of shapes and their analysis.

Keywords – Design, Simulation, Modelling, Solid Works, Failure Analysis

----------X----------

INTRODUCTION

Solidworks becomes one of the most popular software in recent days. Solidworks allows drawing, drafting, design, modeling and analysis using simulation and modeling by itself. These are the main features make it more work friendly and hence it become more popular. In this paper, we are discussing about the procedure used to perform design operations. In addition, we are also having a glance over use of different commands.

CURVED BEAM ANALYSIS

This model, not at all like that of the principal section, will lead you rapidly through those angles of making a limited component Study with which you as of now have insight. Nonetheless, where new data or systems are presented, extra conversation is incorporated. For consistency all through this text, a typical methodology is utilized for the arrangement of all issues.

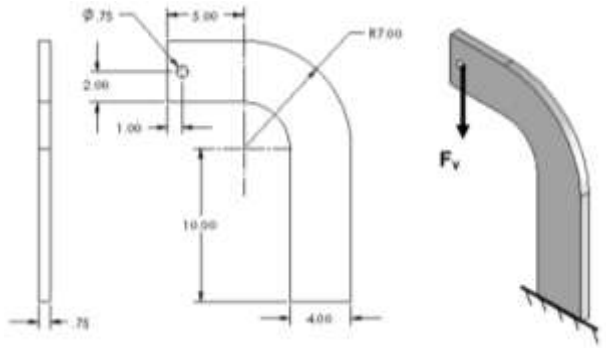

Figure 1: Design of Curved Surface using Solidworks

The above figure shows the modeling of curved surface, i.e. beam. While designing, the beam is supposed to undergone loading conditions. The dimensions and material is decided on the basis of mechanical analysis. However, the analytical calculations need to be performing manually for this study.

This Solidworks study is useful to conducting simulation based analysis of the component. This is important to analyze the actual area of curved surface, which are undergoing maximum stress or load conditions.

PROCEDURE TO CONDUCT THE SIMULATION

Let us consider the loading condition of proposed beam as follows,

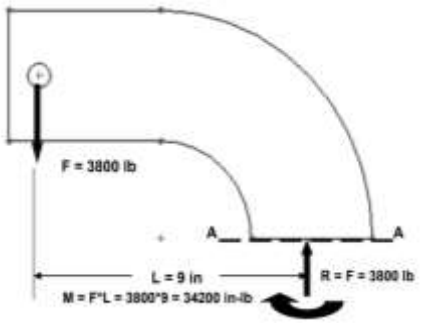

Figure 2: Beam under Loading

The simulation study starts with the generation of element using "SKETCH". This tab is provided to the window.

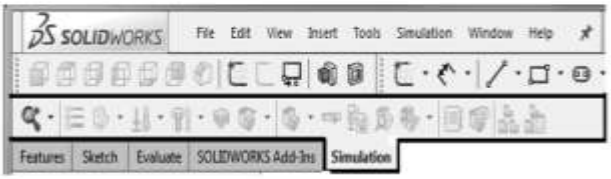

Figure 3: Basic Tabs in Solid works

As per the tabs given in the figure above, it is necessary to ensure is there "Simulation" tab is available or not. If there is "Simulation" tab is not available, then it should have to be import from the "Command Manager". The process is given below,

Figure 4: Command Manger

Figure 5: Addition of Simulation Window to Home Tab

Further, once "Simulation" tab is added to the Home window. Click on it and it will ready to proceed for next practices. To perform the study, user must have to click on "New Study" which is located at the down arrow in "Study Advisor" tab. There are different operations need to carried out to perform the proper tan detailed study of the system.

After clicking on "New Study", at the left bottom, list of possible studies will available. There are different types of study were enlisted such as Static Study, Thermal Study, Fixture, Factor of Safety etc., however, to perform this study, we need to select Static Study.

ASSIGNING THE MATERIAL TO THE MODEL

Figure 6: Material Assigning to the Model

Assigning material plays very important role in the determination of study results. Above figure shows the table for material assignment. In this table, if there is requirement, then user can easily change or specify the properties given in the same.

ASSIGNING THE FIXTURE TO THE MODEL

Figure 7: Fixture Advisor

Assignment of Fixture decides the types of loading and its direction. It includes points such as Fixed Geometry, Roller/Slider, Fixed Hinges, Elastic Support and so all.

APPLYING LOAD CONDITINOS

Application of Load Conditions takes place after the Fixture Advisor. The load can be applied in terms of SI Unit system or any other British Unit System.

MESHING

Meshing is the process of determining the boundary conditions and division of the component in all possible slices. This will be the guide for load distribution over the entire model.

Below fig. shows the view of model after applying meshing.

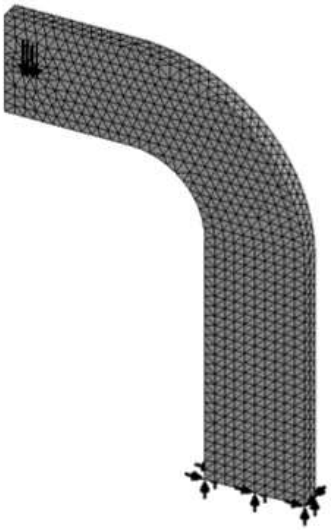

Figure 8: Meshing of Component

Meshing can be done using by applying conditions manually or by applying conditions automatically.

ANALTYSING STUDY RESULTS

Study results will be getting in the form of von misses Stress. It also provides information about displacement and Strain too.

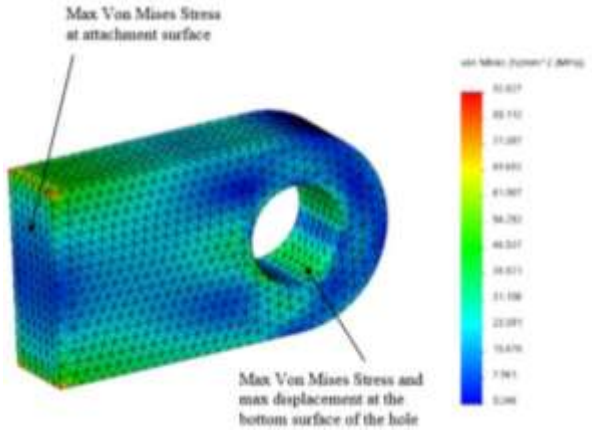

Figure 9: Sample result of Simulation Study

Above figure shows the formation of stress at different points on designed model. The area shown by Dark blue color having least amount of induced stress, however the area having Dark Red color undergoes highest amount of Stress. The stress progress is given from Dark Blue to Dark Red.

CONCLUSION

As a part of Mechanical Engineering, it is necessary to understand the concept and application of various software based studies. This paper provides useful information and slight overview on the “Simulation” i.e. Analysis of the mechanical element using Solidworks. The variety of studies and their effectiveness is also been described in this paper.

REFERENCES

1. F. Ashida, T. R. (1998). Tauchert An inverse problem for determination of transient surface temperature from piezoelectric sensor measurement Journal of Applied Mechanics, 65 (2), pp. 367-373 [doi:10.1115/1. 2789064].

2. P.R. Barrett, H. Foadian, R.J. James, Y.R. Rashid (1992). Thermal-structural analysis methods for RCC dams Proceedings of the Conference of Roller Concrete III, ASCE, San Diego, pp. 407-422.

3. L.J. Liu, J.P. Ren (2005). “Application of the secondary development in SolidWorks”, Mechanical Management and Development (1), pp. 74-75.

4. J. Zhang (2005). “Interface design between AutoCAD and ANSYS”, Chinese Quarterly of Mechanics, 26 (2), pp. 257-262.

5. B.F. Zhu, P. Xu (2001). ‘Methods for stress analysis simulating the construction process of high concrete dams”, Dam Engineering, 6 (4), pp. 243-260.

Corresponding Author

Awatade R. S.*

Student, Department of Mechanical Engineering, Sahakar Maharshi Shankarrao Mohite Patil Institute of Technology and Research, Akluj, Maharashtra, India