IJARIIT INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

ISSN: 2454-132X
Impact Factor: 6.078
(Volume 7, Issue 2 - V7I2-1379)
Available online at: https://www.ijariit.com

Static structural analysis of a water sprinkler using Ansys workbench

Shravan Sanjay Pardeshi
shravan.21810519@viit.ac.in
Vishwakarma Institute of Information Technology, Pune, Maharashtra

ABSTRACT

This paper contains a brief analysis of a mechanical component, which is used in almost all sorts of constructional businesses, viz. "water sprinkler" in Fire Sprinkler System. The paper highlights changes observed in the results of the component, after analyzed with different meshing viz. linear-ordered tetrahedron; Quadratic-order tetrahedron; and Auto meshing provided by ANSYS workbench. The concentration graph is provided for visual analysis of critical portions (suggested to refer the values instead). The result specifies the most appropriate meshing in order to obtain precise static structural analysis in efficient time.

Keywords: *ANSYS Software, Finite element analysis, Static analysis, Water Sprinkler.*

1. INTRODUCTION

The use of fire extinguisher has been practiced by people for a very long time. Since the early days' same materials like sand, water, dirt etc. has been the means of the extinguisher. But the method of the application has been changed drastically. We have also come up with chemical fire extinguishers and mechanical fire extinguisher. Although the complexity has never been increased, rather have decreased for a due course of time.

One of the most famous types is Fire Water Sprinklers. These where only used by industries and large buildings for the cure of unfortunate and unpredictable fire hazard. Due to its design, it is automatic in its task. One does not need to have a command over it. Therefore, now it is seen installed in residents and small-scale projects as well. In the paper, Finite Element Method is used for the structural analysis. The resemblance between the simulated results and the practical ones is so close that, there is no doubt of questioning the accuracy (however, finite element method can never be accurate). ANSYS workbench is most appropriate for finite element testing. The ability of the software to work on various analysis makes it very useful, as all types of simulations can be performed on it. Here in this paper, it is assumed that the pressure applied by the water on the sprinkler (explained in detailed in next section) is a static structural load. The upcoming sections will demonstrate the methodology of the research.

Here, various meshing types are applied to the CAD model of the sprinkler. So as to compare the effect of different meshing on to the outcome of the analysis. The paper will contain three different types of meshing

1. Linear Order Tetrahedron.
2. Quadratic Order Tetrahedron
3. Automatic Meshing generated by ANSYS

2. WORKING OF THE SPRINKLER.

2.1 Installation/Construction

The Sprinkler is installed on the ceiling of a room/area. The empty space inside the nut like structure is attached with a glass bulb containing alcohol, which bursts when heated. The expansion play (can be observed as a air bubble) in the glass bulb is responsible for the different temperature range of the system.

The following figure specifies the colour assigned by IFC® for the different threshold of the alcohol.

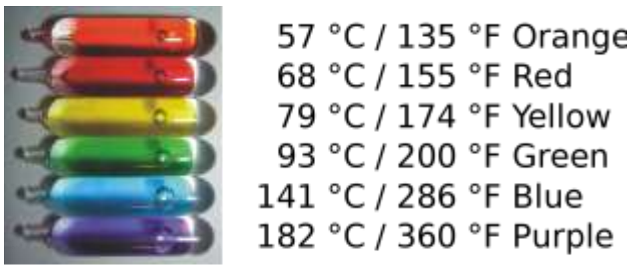

Fig. 1: Types of glass bulb with respective threshold temperature.

2.2 Working of The System

The sprinkler is installed on the ceiling, and is made sure that it is connected to the end of the water supplying line, which leads to the water storage of the building/structure. Since there is no obstacle in between the water storage and the end of the pipe line. All the pressure is experienced by the threaded portion of the sprinkler. Now the glass bulb with alcohol in it, sits at the entrance of the sprinkler, and all that force is acting on the glass bulb.

Now when the temperature of the room exceeds the threshold of the bulb, the bulb breaks within a fraction of the millisecond. Now the scenario is as such: a high pressure around 18000 Pa rushes through the narrow outlet and towards the sprinkling disk. Here, this pressurized water is dispersed with the help of the bullet profile, but still all the pressure is experienced by the sprinkler face.

Due to the outspread of fins of the sprinkler, it is possible to generate enough turbulence so as to convert the water stream into water droplets.

3. MODELING

The CAD modeling of the component was designed using SOLIDWORKS 2018.

Fig. 2: Water sprinkler design with SOLIDWORKS 2018

4. MATERIAL PROPERTIES

The material used for the analysis of sprinkler is Stainless steel. The properties of the stainless steel are tabulated below:

Table 1: Material Properties of Stainless Steel (SS):

Properties	Values	
Young's Modulus	$1.93x10^{+11}$	Pa
Poisson's Ratio	0.31	
Bulk Modulus	$1.693x10^{+11}$	Pa
Shear Modulus	$7.3664x10^{+10}$	Pa

5. MESHING

The meshing of the component was generated in the Ansys workbench. The total nodes and elements involved in this process were meshed using: Auto-meshing by ANSYS, the observed nodes and elements were 25616 and 13165.

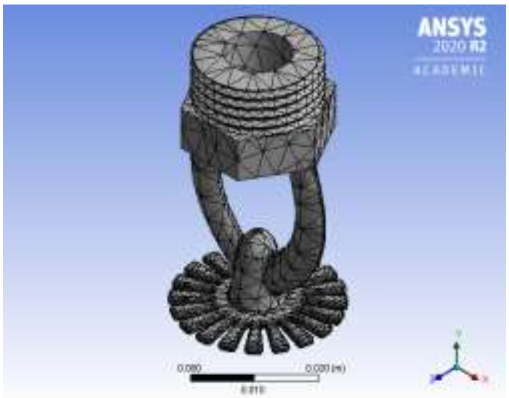

Fig. 3: Auto-meshing by ANSYS

○ Linear Order Tetrahedron, the observed nodes and elements were 3290 and 10146.

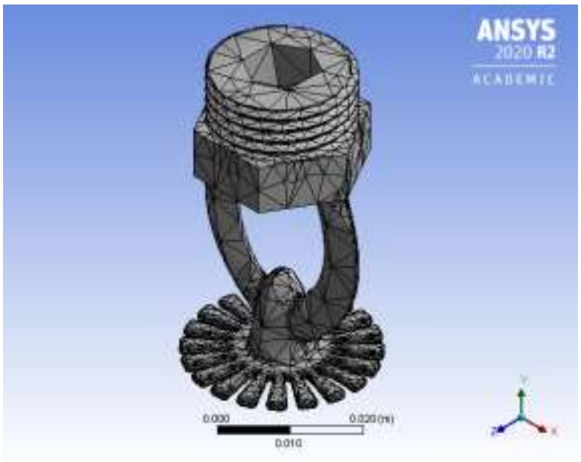

Fig. 4: Linear Ordered Tetrahedron

○ Quadratic Order Tetrahedron, the observed nodes and elements were 19017 and 9882.

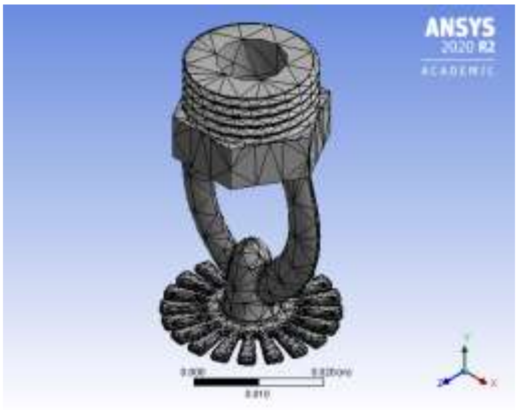

Fig. 5: Auto meshing by ANSYS

6. RESULT

The following results were obtained for the different types of meshing:

6.1 Linear order tetrahedron.

Table 2: Result for linear-Tetrahedron meshing

RESULT	Minimum	Maximum
Equivalent Stress (Pa)	51.555	$7.2409x10^{5}$
Equivalent Elastic Strain (m/m)	$4.0142x10^{-10}$	$3.8522x10^{-7}$
Total Deformation (m)	0	$1.9546x10^{-7}$

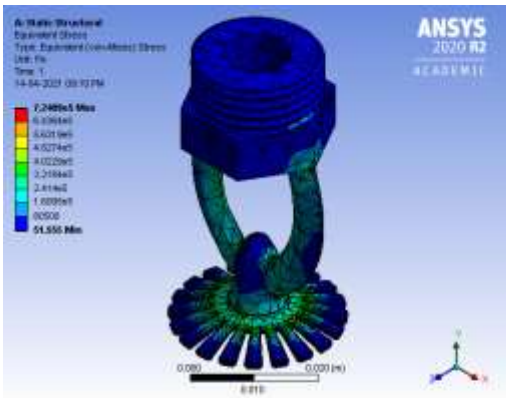

Figure 6: Equivalent Stress (linear-tetrahedron)

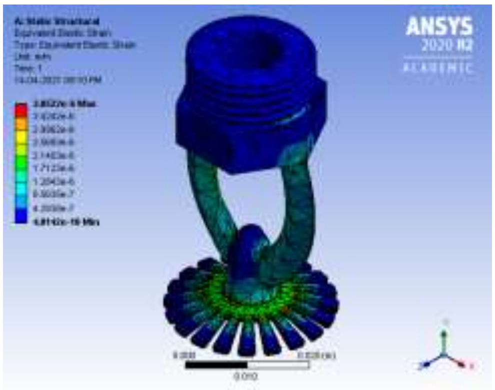

Figure 7: Equivalent Strain (linear-tetrahedron)

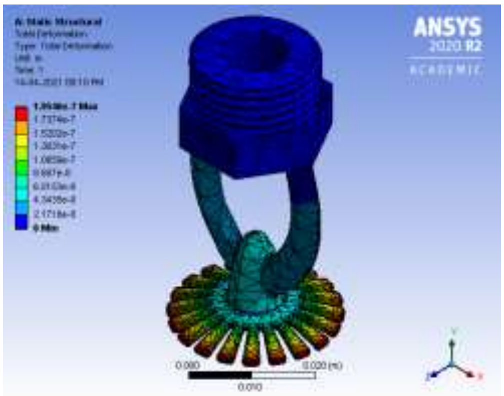

Figure 8: Total Deformation (linear-tetrahedron)

6.2 Quadratic Ordered Tetrahedron

Table 3: Result for Quad-Tetrahedron meshing

RESULT	Minimum	Maximum
Equivalent Stress (Pa)	91.639	$1.3708e^{5}$
Equivalent Elastic Strain (m/m)	$5.9085x10^{-10}$	$7.1534x10^{-7}$
Total Deformation (m)	0	$2.9312x10^{-7}$

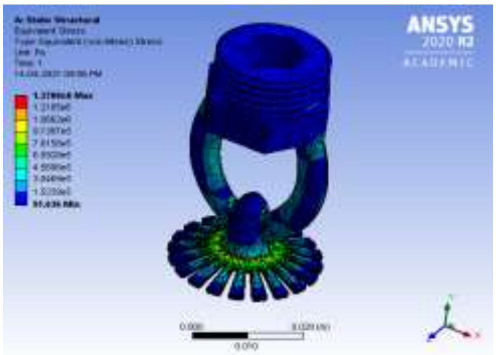

Fig. 9: Equivalent Stress (Quad-tetrahedron)

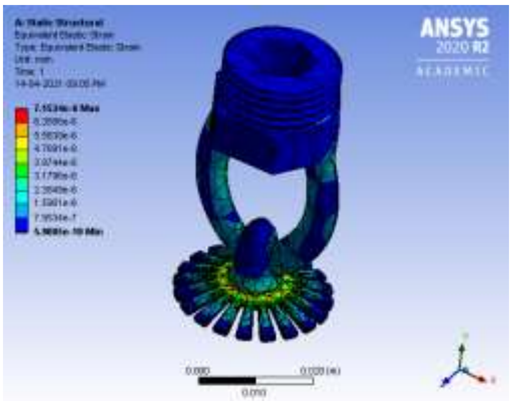

Fig. 10: Equivalent Strain (Quad-tetrahedron)

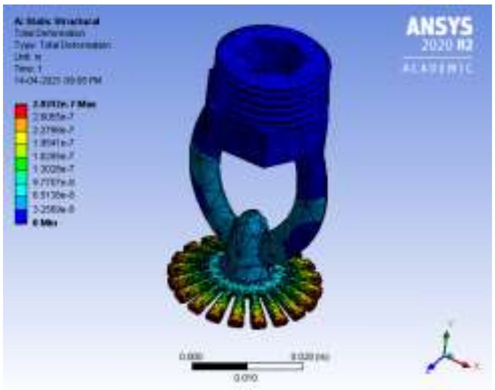

Fig. 11: Total Deformation (Quad-tetrahedron)

6.3 Automatic meshing Generated by ANSYS

Table 4: Result for Auto-meshing by ANSYS

RESULT	Minimum	Maximum
Equivalent Stress (Pa)	119.9	$1.625x10^{5}$
Equivalent Elastic Strain (m/m)	$1.5765x10^{-10}$	$8.4759x10^{-7}$
Total Deformation (m)	0	$2.9791x10^{-7}$

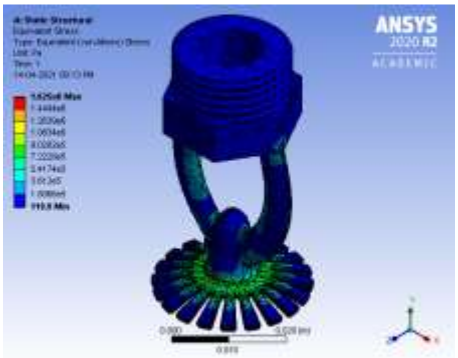

Fig. 12: Equivalent Stress (Auto meshing by ANSYS)

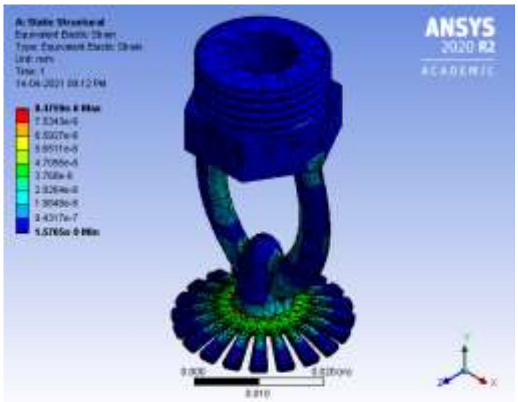

Fig. 13: Equivalent Strain (Auto meshing by ANSYS)

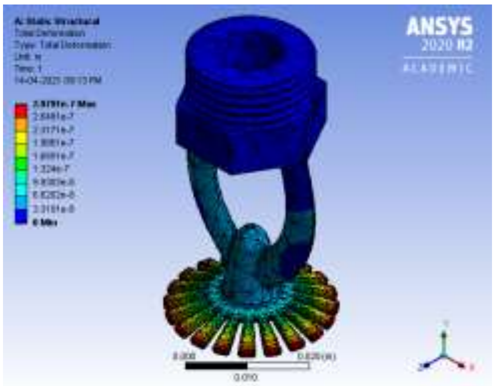

Fig. 14: Total Deformation (Auto meshing by ANSYS)

7. CONCLUSION

In this paper, from the above analysis, we have done a comparison between linear-tetrahedron; quad-tetrahedron; and auto-meshing by ANSYS. And came up with a conclusion as follows:

Overall, maximum elements were obtained in Auto-meshing, which will definitely give better results than others. The next most appropriate meshing would be Quad-tetrahedron meshing even if linear-tetrahedron meshing gives higher number of elements. The reason behind this conclusion is the structure of the component. Since the structure contains curvilinear portions, it is recommended by the software to use the quadratic order. The above statement can be supported by an argument i.e. by observing the meshing obtained by quad-tetrahedron and auto-meshing by ANSYS, the nodal placement is very similar to each other unlike linear-tetrahedron.

8. REFERENCES

[1] Finite Element modeling and Simulation with ANSYS Workbench by Xialin Chen, Yijiun Liu.
[2] Ansys Workbench tutorial by Lawrence Kent.
[3] Pavol Lengvarsky*, Miroslav Pástor, Jozef Bocko "Static Structural Analysis of Water Tank"

BIBLIOGRAPHY

Shravan Sanjay Pardeshi
Vishwakarma Institute of Information Technology, Pune, Maharashtra, India