International Archive of Applied Sciences and Technology
Int. Arch. App. Sci. Technol; Vol 9 [4] December 2018 : 41-47

[ISO9001: 2008 Certified Organization]
www.soeagra.com/iaast.html

IAAST
ONLINE ISSN 2277- 1565
PRINT ISSN 0976 - 4828

CODEN: IAASCA ORIGINAL ARTICLE

DOI: .10.15515/iaast.0976-4828.9.4.4147

Finite Element Analysis of Wire Rope under Axial Tensile Load-I

Krishan Kumar[a*]
[a]Department of Mechanical Engineering, NITTTR, Sector-26, Chandigarh -160019, India
Email: erkrishan.bishnoi@gmail.com

ABSTRACT

Wire ropes have been used in various applications of mechanical, civil and marine engineering since 13th century. Finite element analysis of wire rope is done to get various dpenedent variables at critical points e.g. maximum stress, maximum deflection and minimum factor of safety. Due to helical structure of wire rope value of induced stress under a given loading is not same across a given cross-section. In this analysis bonded contact between wire was used. This paper presents a comprehensive comparative analysis of different stainless steel conventional wire ropes of 16mm, 18mm and 20mm diameters with 1-6-6 configuration. A static tensile load with maximum value of 15.625kN was applied. For finite element element analysis CAE software ANSYS 18.1 was used. Fine meshing having element size of 0.2 mm was used. Results obtained were very close to experimental results.

Key words: Wire rope, Finite Element Analysis, Von-Misses stress,Stainless Steel, ANSYS 18.1

Received 28/07/2018 Revised 25/08/2018 Accepted 24/10/2018

Citation of this article
Krishan Kumar. Finite Element Analysis of Wire Rope under Axial Tensile Load-I.Int. Arch. App. Sci. Technol; Vol 9 [4] December 2018. 41-47.

INTRODUCTION

High strength wire ropes are generally made of steel. Under corrosive conditions stainless steel wire ropes are used due to their higher resistance to corrosion. Wire ropes are very important structural members, used for supporting and transmitting tensile load. Their flexibility under bending and high rigidity under tensile loads makes them suitable for various electrical, mechanical, mining applications [1]. Applications include electrical power transmission, lifts, aircraft arresting cables, suspension bridges, mining equipment [2]. Wire ropes offer a strong, durable and economical option for most of the lifting applications. Various components of wire rope are illustrated in figure 1. There are three basic components i.e. Wire, Strand and core [3].

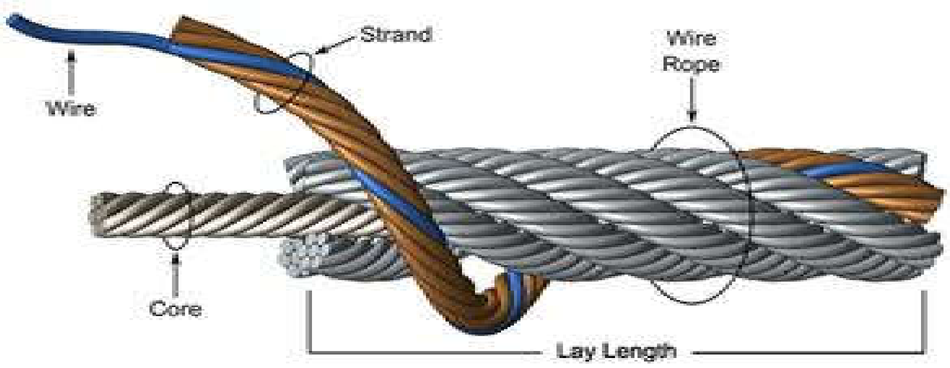

Figure 1 Components of Wire Rope

Technological advancement leads to frequent computer aided design and numerical analyses in predicting the wire rope behavior under various types of loading e.g. tensile loading, torsional loading and a combination of both etc. These analyses are required to conduct various tests to evaluate wire rope structural condition and bearing capacity. Experimental work requires expensive testing devices. Numerical analysis such as finite element analysis was a non-destructive method to study the mechanical behaviour of wire ropes [4]. Mathematical geometric model of wire rope can be generated by many computer aided softwares like Solidworks™**,** Pro-Engineering, CATIA etc [5].

There have various methods to generate geometric model of wire rope e.g. by using mathematical equations or by using various commands of Solidwork™. These geometric models are impoted to computer aided engineering softwares like Abaquas, ANSYS and FORTRAN, Matlab etc [6]. These finite elemnt analysis methods give approximate results as compared to analytical methods giving accurate results. But, it is not possible to solve real life complex problems by using analytical methods due to unavailability of static equations [7]. Finite element analysis (FEA) sometimes also known as finite element method (FEM). FEA presents many advantages over analytical methods like visualisation of effect of applied loads, evaluate the product for different loads as well as different materials of construction. No physical model is required for testing the mechanical behaviour of wire rope [8]. However, even after considering all these benefits, it is still very difficult to generate the geometric model and analyze wire ropes by using numerical methods e.g. finite element method (FEM).

This kind of analysis requires substantial computer resources with high capability configurations. Numerical analysis can be used to provide a better understanding and prediction, of the mechanical behavior of the wire rope. Thus, it reduces the need for expensive tests. Finite element analysis of wire rope can be done by considering the interwire friction or by neglecting it [9]. There can also be considered solid bonding between wires. In order to accomplish all of that, the aim of this paper was to explore some aspects of 3D modeling of a wire rope by using the finite element method based ANSYS 18.1 software under different types of axial loading.

FINITE ELEMENT MODEL AND ANALYSIS

For finite element analysis wire rope geometries were constructed by using Solidworks™ 2017 version [10]. Three wire ropes of 16mm, 18mm and 20mm diameters were considered. Wire rope geometries were having 1-6-6 configuration. Connection between wires was considered as solid bonded type. Stainless Steel material for wire ropes with following elstic properties was considered:-

Nominal ultimate strength,σ_{ut}= 586N mm^{-2}

Nominal yield strength, σ_{yt} = 207N mm^{-2}

Modulus of elasticity,E =18 x 10^{4} N mm^{-2} Density, ρ =7750 kg mm^{-3}

Poisson's ratio, ν = 0.3

Material was considered as homogeneous, isotropic and linearly elastic.

Finite Element Analysis of Wire Rope 1

First wire rope of Stainless Steel having diameter, d = 16 mm, Length, L = 100 mm& 1-6-6 strand structure was designed in computer software SolidWork™ and its finite element analysis (FEA) was done in computer software ANSYS 18.1. Geometry of wire rope 1 is shown in figure 2.

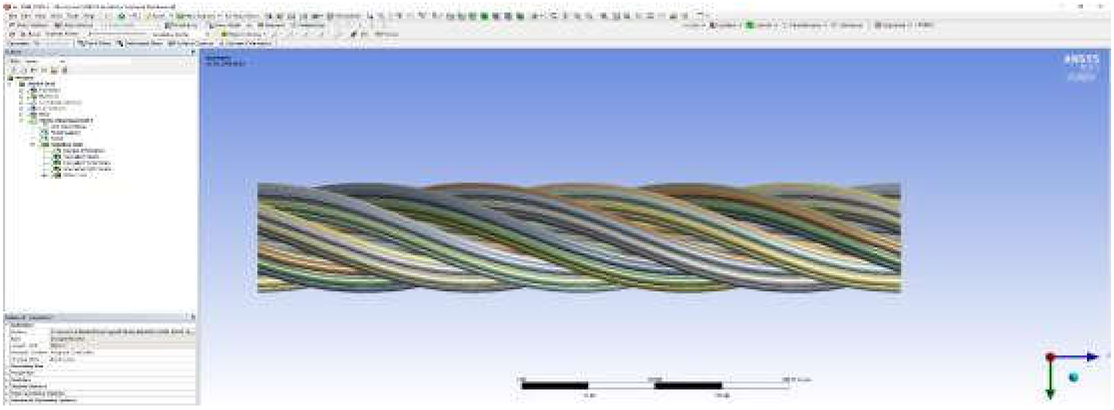

Figure 1 Geometry of Wire Rope 2

Imoprtation of wire rope was followed by mesh generation to convert it into a finite degree of freedom system. Numbers of nodes were 895642 and numbers of elements were 5961994. Meshed wire rope is shown in figure 3.

Figure 3 Meshed Wire rope 1

Boundary condition of one end fixed and other end loaded with tensile force was applied. Details of applied load and induced maximum stress is given in table1. Under various load wire rope was analysed for induced von-misses stress.

Table1 Applied Tensile load & Induced maximum Stress in wire rope 1

Sr. No.	Tensile Load (kN)	Maximum Stress (MPa)
1	5	34.938
2	10	69.876
3	15	104.81
4	15.625	109.18

Distribution of induced stress in illustrated in figure 4. From figure it can be observed that stress indued was not uniform in wire rope. Maximum stress induced was into the core and few of the outer most wire because of the helical nature of wire rope.

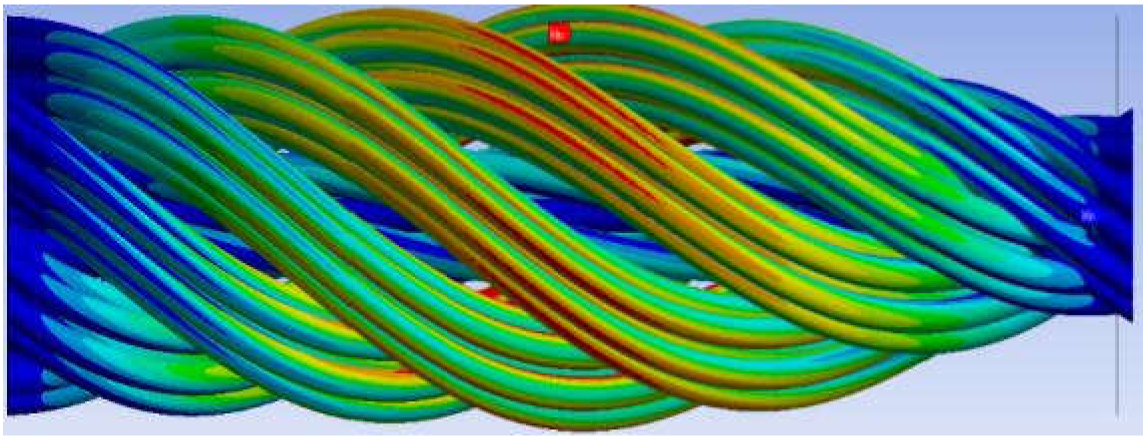

Figure 4 Stress distribution in wire rope 1

Finite Element Analysis of Wire Rope 2

Second wire rope of Stainless Steel having diameter, d = 18 mm, Length, L = 100 mm & 1-6-6 strand structure was designed in computer software SolidWork™ and its finite element analysis (FEA) was done in computer software ANSYS 18.1. Geometry of wire rope 2 is illustrated in figure 5.

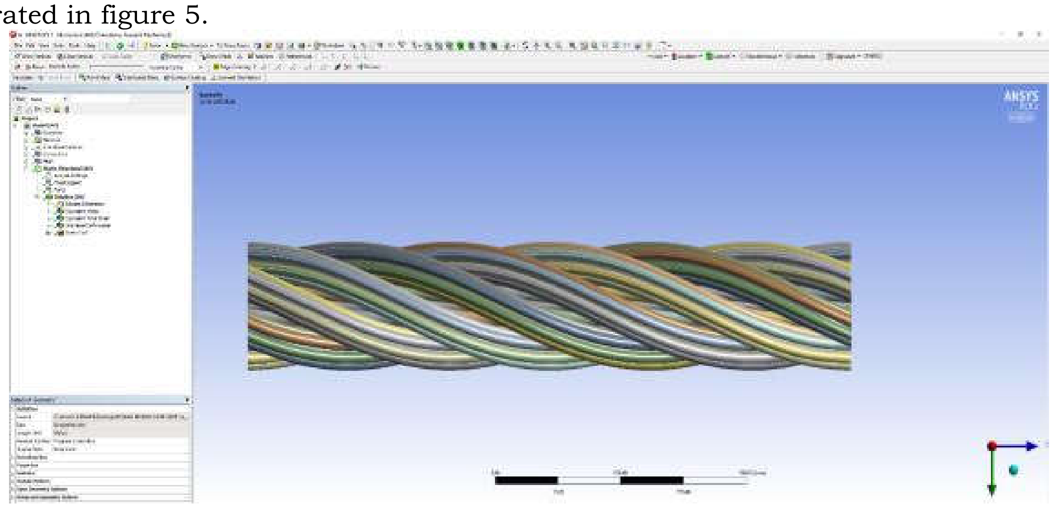

Figure 5 Geometry of Wire rope 2

After meshing numbers of nodes were 8951905 and numbers of elements were 5962004. Meshed wire rope 2 is shown in figure 6.

Figure 6 Mehsed Wire Rope 2

Details of applied load and induced maximum von-misses stress is given in table 2.

Table 2 Tensile load and Maximum stress induced in wire rope 2

Sr. No.	Tensile Load (kN)	Maximum Stress (MPa)
1	5	24.01
2	10	48.02
3	15	72.04
4	15.625	75.04

Stress in wire rope follows non-uniform pattern as shown in figure 7.

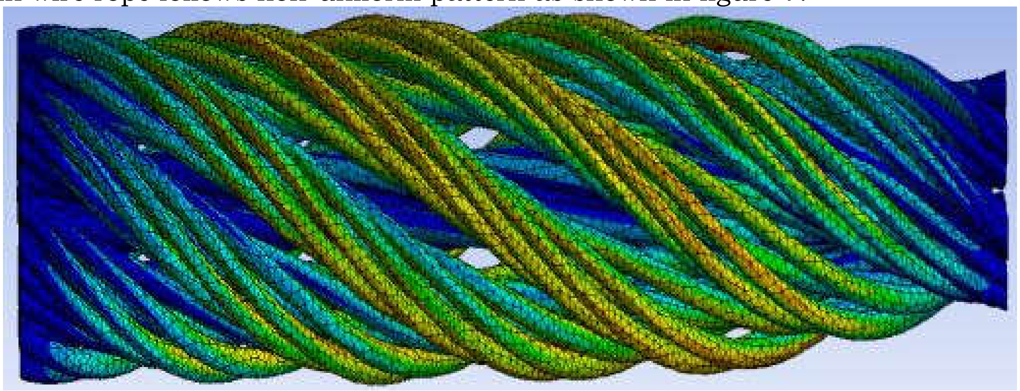

Figure 7 Stress distribution in wire rope 2

Finite Element Analysis of Wire Rope 3

Third wire rope of Stainless Steel having diameter, d = 20 mm, Length, L = 100 mm was designed in computer software SolidWork™ and its finite element analysis (FEA) was done in computer software ANSYS 18.1. Geometry of wire rope 3 is shown in figure 8.

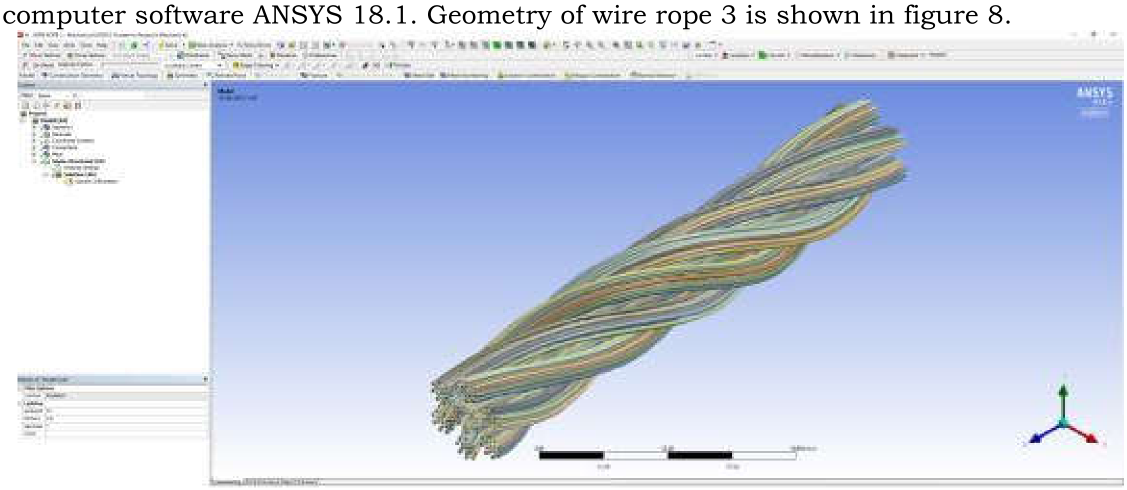

Figure 8 Geometry of Wire Rope 3

Meshing was done by considering the element size of 0.2 mm and Numbers of nodes were 8634039 and numbers of elements were 4573409. Figure 9 shows meshed wire rope 3.

Figure 9 Meshed Wire Rope 3

Details of applied load and induced maximum von-misses stress is given in table 3.

Table 3 Tensile load and Maximum stress induced in wire rope 3

Sr. No.	Tensile Load (kN)	Maximum Stress (MPa)
1	5	16.919
2	10	33.838
3	15	50.757
4	15.625	52.872

Von Misses stress in wire rope 3 follows a non-uniform pattern similar to that wire rope 1 and wire rope 2. Distribution of induced stress in wire rope 3 under applied tensile load is illustrated in figure 10.

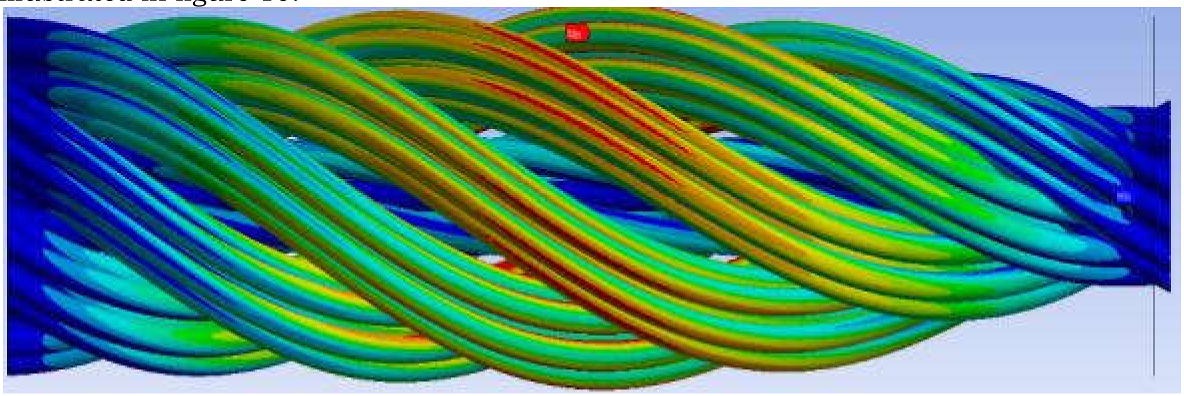

Figure 10 Stress distribution in wire rope 3

RESULTS AND DISCUSSION

From the table 1, 2 and 3, it can be concluded that as the diameter of wire rope increases, value of maximum stress induced in wire rope deceases. Stress induced in wire rope is not uniform across a given cross-section. Moreover, magnitude of von-misses stress (Von-Misses stress is considered here because wire rope material that is Stainless Steel is ductile) depends on multiple factors. Few of crucial factors on which induced stress depends are listed below:

- Combination of wires with varying diameters in core as well in strands.
- Helix angle/ angle of twist per unit length of wires in strands.
- Helix angle/ angle of twist per unit length of strands in wire riope.

Comparative analysis of maximum stress induced in concerned wire ropes of 16mm, 18mm and 20mm diameters under the maximum load of 15.625kN is illustrated in figure 11.

Wire Rope 1	Wire Rope 2	Wire Rope 3	
	-31.27%	-51.57%	Wire Rope 1
		-29.54%	Wire Rope 2
			Wire Rope 3

Figure 11 Comparative analysis of Stress induced in Wire Ropes

From figure 6, it can be observed tha when diameter of wire rope increased from 16 mm to 18mm there will 31.27% less stress induced under the load of 15.625kN. Similarily, for 20mm diameter wire rope there will be 51.57% less stress induced as compared to 16mm diameter wire rope. Again, when diameter is increased from 18mm to 20mm, maximum stress induced in wire rope will be 29.54% less as compared to that is in 18mm diameter wire rope.

CONCLUSION

Conventional wire ropes of diameters 16mm, 18mm & 20mm were analysed here by using numerical method i.e. finite element analysis by using ANSYS 18.1. Wire rope geometry can be generated by using various computer aided softwares like Solidworks, CATIA, Pro-E etc. The results for maximum stress induced in wire ropes decreses with increase in diameter for a given applied load. Stress distribution in wire rope is not uniform over ther cross-section of wire rope. Stress induced in wire rope also depends on several other factors e.g. helix angle, combination of wires with different diameter etc.

REFERENCES

1. Kumar, K., Goyal, D., & Banwait, (2018). S. S. KEY VARIABLES IN FRETTING BEHAVIOUR OF WIRE ROPE: A REVIEW, International Journal of Technical Innovation in Modern Engineering & Science (IJTIMES), vol. 04, 20-24.
2. http://www.ushamartin.com/wp-content/uploads/2014/04/Wire-Rope-Handbook.pdfDated: 27/09/2018
3. http://www.ushamartin.com/wp-content/uploads/2015/09/Crane-Ropes_Usha-Martin.pdf Dated: 27/09/2018.
4. Kastratović, G., Vidanović, N., Bakić, V., & Rašuo, B. (2014). On finite element analysis of sling wire rope subjected to axial loading. Ocean Engineering, 88, 480-487.
5. Fedorko, G., Stanova, E., Molnar, V., Husakova, N., & Kmet, S. (2014). Computer modelling and finite element analysis of spiral triangular strands. Advances in Engineering Software, 73, 11-21.
6. Du, W., Ma, B., Cao, D., & Xie, Z. (2017, February). Finite element analysis on the stress state of rope hoisting equipment based on the ABAQUS. In Mechanical and Intelligent Manufacturing Technologies (ICMIMT), International Conference on (pp. 84-88). IEEE.
7. Gokhale, N. S. (2008). Practical finite element analysis. Finite to infinite.
8. Zienkiewicz, O. C., Taylor, R. L., Zienkiewicz, O. C., & Taylor, R. L. (1977). The finite element method (Vol. 36). London: McGraw-hill.
9. Strang, G., & Fix, G. J. (1973). An analysis of the finite element method (Vol. 212). Englewood Cliffs, NJ: Prentice-hall.
10. Kumar, K., Goyal, D., & Banwait, S. S. (2018). Geometric Modeling Of Wire Rope, International Journal of Technical Innovation in Modern Engineering & Science (IJTIMES) vol 04, 12-19.

