International Journal of Engineering and Advanced Technology (IJEAT)
ISSN: 2249 – 8958, Volume-3, Issue-3, February 2014

Analysis of Hooks Joint Using Ansys by Von-Mises Method

Kamal Kashyap, D.G.Mahto

Abstract- Hooks joint (universal Coupling) is a very important for transmitting power from one shaft to another. Sometimes the failure in this assembly takes place due to improper design and analyzing .The aim of this paper is to present a finite element analysis predicting the behavior of hooks joint under different loads on different parts. The software package ANSYS is used to model the joint. This paper may help to improve the quality of hooks joint.

Keywords:- hooks joint, ANSYS ,Finite element analysis ,Von Mises Equation.

I. INTRODUCTION

A universal joint is a positive, mechanical connection between rotating shafts, which are usually not parallel, but intersecting. They are used to transmit motion, power, or both.[1].its very important to analyze all the aspects regarding universal coupling, Von Mises stress is widely used by designers, to check whether their design will withstand given load condition. In this paper we will try to measure total deformation by Von Mises stress using ANSYS software.

Condition of failure under Von mises equation in Y axis

$$\left[\frac{(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2}{2}\right]^{1/2} \geq \sigma_y$$

Left hand side of above equation is denoted as Von Mises stress[2].

$$\left[\frac{(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2}{2}\right]^{1/2} = \sigma_v$$

Joint under load

In order to analyze a hooks joint a finite element model is made with the help of ANSYS software as shown in fig 1.1[4].The purpose of making this model is to make the calculation done by ANSYS as accurate as possible.

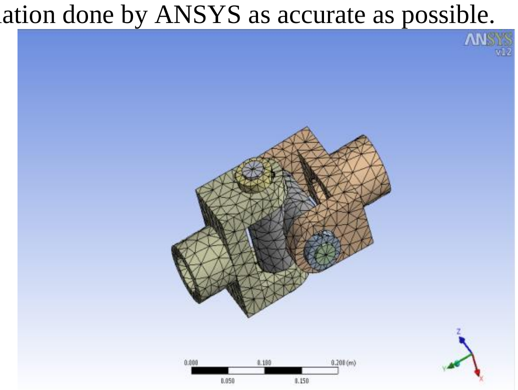

Fig1.1 FIFNITE ELEMENT MODEL OF HOOKS JOINT

Manuscript received February 2014.

Kamal Kashyap, Department Of Mechanical Engineering , LLR Institute, H.P,173223, India
Dr.D.G.Mahto, Green Hills Engineering College, H.P,173229, India

Joint configuration

Fig1.2 shows the typical configuration of the joint which consist of two fork which are connected with the help of centre block and pins.

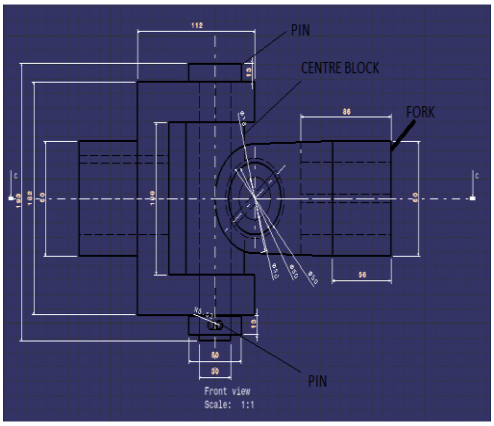

Fig1.2 JIONT CONFIGURATION OF HOOKS JOINT

II. PARAMETRES

There are some parameters which are to be given to the ANSYS software in case to analyze the Hooks Joint these parameters are the units that we consider in solving the particular problem, The constraints of the material itself from which the product is made and lastly the parameters that we decide to analyze the problem.

2.1 Unit system used

The unit system that we use during this project is (Metric (m, kg, N, s, V, A) Degrees rad/s Celsius).

Angle	Degrees
Rotational Velocity	rad/sec
Temperature	Celsius

Table1.1 UNIT SYSTEM USED

2.2 Material -structural steel constants

The material that's used in this project for testing is structural steel which have the properties as shown in TABLE1.2

Density	7850 kg m^-3
Coefficient of Thermal Expansion	1.2e-005c^-1
Specific heat	434 J kg^1c^-1
Thermal Conductivity	60.5 W m^-1 C^-1
Resistivity	1.7e-007 ohm m
Compressive ultimate strength in Pa	0
Compressive yield strength in Pa	2.5e+008
Tensile yield strength in pa	2.5e+008
Tensile ultimate strength in pa	4.6e+008
Environmental temperature	22°c

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication Pvt. Ltd.

TABLE 1.2 STRUCTURAL STEEL CONSTANTS

2.3 Other parameters

VOLUME	MASS
1.5833e-006m^3	1.2429e-002kg
1.577e-005m^3	0.12379kg
1.5191e-004m^3	1.1925kg
7.727e-004m^3	6.0657kg
1.5833e-006m^3	1.2429e-002kg

TABLE 1.3 VOLUME MASS TAKEN DURING ANALYSIS

CENTROID X	CENTROID Y	CENTROID Z
-1.3981e-018m	-2.0683E-019m	-8.8e-002m
-1.4861e-009m	-2.9202e-011m	-8.7998e-002m
-1.4593e-012m	1.181e-012m	7.6249e-003m
-2.7206e-008m	-5.0103e-002m	-9.0313e-005m
-1.3981e-018m	-2.0683E-019m	-8.8e-002m

TABLE 1.4 CENTROID IN X,Y AND Z AXIS

M.O.I Ip 1(Kg.m^2)	M.O.I Ip 2(Kg.m^2)	M.O.I Ip 3(Kg.m^2)
3.257e-006	5.5202e-008	3.257e-006
2.7111e-005	2.887e-005	5.2395e-005
4.2504e-003	1.7187e-004	4.251e-003
2.0486e-002	2.6527e-002	1.2694e-002
3.257e-006	5.5202e-008	3.257e-006

TABLE1.5 MOI IN IP1,2 AND3

III RESULT

According to the above parameters when we apply the load on the assembly and the deformation takes place differently at different parts as shown in the FIG 1.4 and from that we are able to get the resultant of total deformation on ANSYS from which we finally able to plot a graph between the stress and total deformation and we can also find out that at which particular part there is maximum deformation and at which part we get the minimum deformation after applying the load.

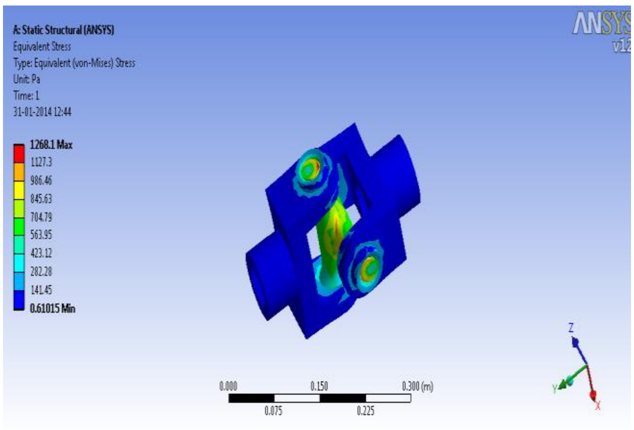

FIG 1.3 EQUIVALENT STRESS DISTRIBUTION

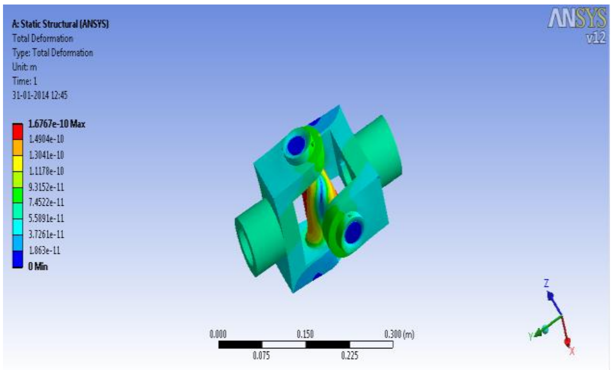

FIG 1.4 TOTAL DEFORMATION

EquivalentStress(Pa)	Total Deformation
0.6101	0
141.45	1.863 e-11
282.28	3.7261 e-11
423.12	5.5891 e-11
563.95	7.4522 e-11
704.79	9.3152 e-11
845.63	1.1178 e-10
986.46	1.3041 e-10
1127.3	1.4904 e-10
1268.1	1.6767 e-10

TABLE1.6 RESULT STRESS APPLIED /TOTAL DEFORMATION

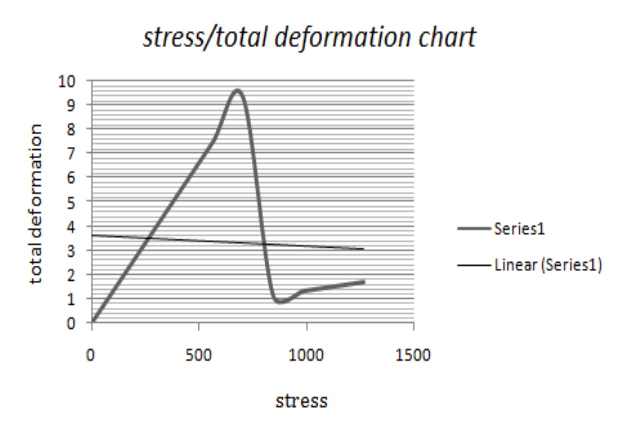

CHART1.1 STRESS APPLIED/TOTAL DEFORMATION

IV. CONCLUSION

The conclusion that came out from this project is that the total deformation that came out is max (9.3152 e-11) which takes place at the centre block of the entire assembly. and the minimum deformation takes place at the hub.

REFRENCES

1. www.sdp-si.com
2. www.learnengineering.org
3. Vedam, K., Naganathan, N., Szadkowski, A., and Prange, E., "Analysis of an Automotive Driveline with Cardan Universal Joints," SAE Technical Paper 950895, 1995, doi:10.4271/950895.
4. A.N. Sherbourne, M.R. Bahaari, 3D simulation of end-plate bolted connections, Journal of Structural Engineering, ASCE 120 (11) (1994) 3122–3136
5. Elsayed Mashaly"finite element alalysis of beam to column joints in steel frames under cyclic loading",Alexandria EngineeringJournal (2011) 50,91–104.

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication Pvt. Ltd.