Static, Modal and Transient Analysis of Radial Arm using Ansys

Rahulji Dala[1], B. Gandhi[1]
[1]Department of ME, Sree Dattha Institute of Engineering and Science, Hyderabad, India.

ABSTRACT

A computational capability is evolved for the gold standard design of radial drilling gadget shape to satisfy static stress and natural frequency necessities the use of finite element idealization. The radial drilling device structure is idealized with body elements and is analyzed by way of the usage of specific combos of move sectional shapes for the radial arm and the column. From the consequences acquired, the best mixture of cross-sectional shapes is recommended for the structure. With this mixture of cross-sectional shapes, mathematical programming strategies are used to find the minimal weight design of the radial drilling device shape. A sensitivity analysis is carried out about the most excellent factor to locate the consequences of adjustments in layout variables on the structural weight and the reaction quantities.3D Modeling in CREO parametric software program and evaluation in ANSYS software.

Keywords: Radial arm, statistical analysis, Ansys, Catia and FEA.

I. INTRODUCTION

A radial arm press is a geared drill head that is set up on an arm meeting that may be moved round to the volume of its arm attain. The maximum crucial components are the arm, column, and the drill head. The drill head of the radial drilling machine can be moved, adjusted in top, and circled. Aside from its compact layout, the radial drill press is capable of positioning its drill head to the paintings piece through this radial arm mechanism.This is probably one of the reasons why extra machinists opt for the usage of this kind of drilling system. In fact, the radial drilling device is taken into consideration the most flexible type of drill press. The tasks that a radial drilling machine can do encompass boring holes, countersinking, and grinding off small debris in masonry works.Although a few drill presses are floor established, the most not unusual set-up of radial arm drill presses are the ones which are hooked up on paintings benches or tables. With this sort of set-up, it's far less difficult to mount the drill and the paintings pieces. There is no want to reposition work portions due to the fact the arm can expand as a ways as its length could allow. Moreover, it is less complicated to move big work pieces with the radial arm drilling gadget. Large paintings portions may be installed on the desk by means of cranes because the arm can be swiveled out of the manner.

II. LITERATURE REVIEW

Optimum Design of Radial Drilling Machine Structure to Satisfy Static Rigidity and Natural Frequency RequirementsA computational capability is developed for the most fulfilling layout of radial drilling gadget structure to fulfill static stress and herbal frequency requirements using finite detail idealization. The radial drilling gadget shape is idealized with body elements and is analyzed by way of the usage of unique combinations of pass sectional shapes for the radial arm and the column. From the outcomes acquired, the high-quality mixture of go sectional shapes is recommended for the shape. With this aggregate of move sectional shapes, mathematical programming techniques are used to discover the minimum weight layout of the radial drilling gadget structure. A sensitivity analysis is performed about the most efficient factor to discover the effects of modifications in design variables on the structural weight and the reaction quantities.

Modal Analysis of Machine Tool Column Using Finite Element Method. The performance of a system tool is sooner or later assessed by its capability to supply a aspect of the desired geometry in minimum time and at small running fee. It is commonplace to base the structural design of any system device ordinarily upon the necessities of static tension and minimum herbal frequency of vibration. The working residences of machines like reducing speed, feed and depth of cut in addition to the scale of the work piece additionally must be saved in thoughts via a gadget device structural designer. This paper offers a singular technique to the layout of device device column for static and dynamic pressure requirement. Model evaluation is finished efficiently via use of General Finite Element Analysis software program ANSYS. Studies on machine device column are used to illustrate finite element based totally idea evaluation technique. This paper also gives effects obtained from the computations of skinny walled field kind columns which might be subjected to torsional and bending loads in case of static evaluation and also results from modal analysis.

III. DESIGN AND ANALYSIS SOFTWARES

CAD

Computer-aided design (CAD) is the use of laptop systems (or workstations) to resource within the advent, modification, evaluation, or optimization of a layout. CAD software is used to growth the productivity of the fashion designer, enhance the exceptional of layout, enhance communications thru documentation, and to create a database for manufacturing. CAD output is frequently within the form of digital documents for print, machining, or exceptional manufacturing operations.

CREO

PTC CREO, formerly referred to as Pro/ENGINEER, is three-D modeling software carried out in mechanical engineering, design, manufacturing, and in CAD drafting issuer companies. It became one of the first three-D CAD modeling applications that used a rule-based parametric device. Using parameters, dimensions and abilities to seize the behavior of the product, it is able to optimize the improvement product in addition to the layout itself.The name changed into modified in 2010 from Pro/ENGINEER Wildfire to CREO.

3-D MODEL OF DRILLING MACHINE

Fig.1 3-D model of drilling machine.

FEA

Finite element evaluation is a manner of fixing, typically about, exceptional troubles in engineering and era. It is used mainly for problems for which no actual answer, expressible in some mathematical form, is available. As such, it is a numerical instead of an analytical approach. Methods of this kind are wanted because of the truth analytical techniques can not cope with the real, complicated problems which might be met with in engineering.

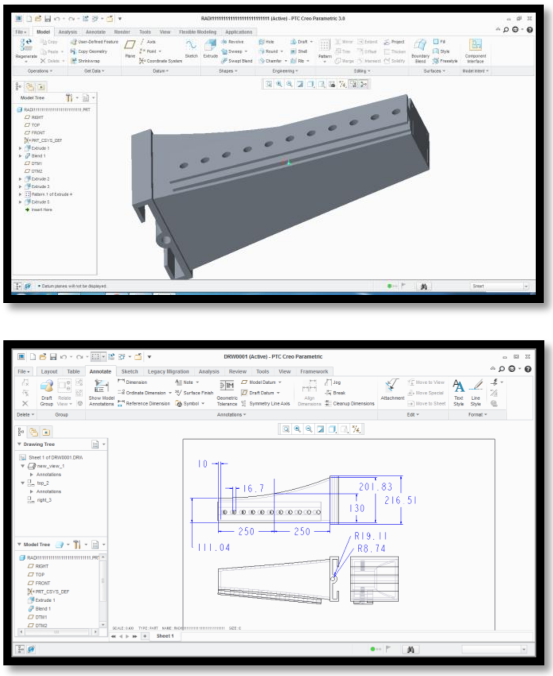

Fig. 2 2D model of radial arm

ANSYS

Structural Analysis

ANSYS Autodyne is pc simulation device for simulating the reaction of materials to quick period immoderate loadings from impact, excessive stress or explosions. ANSYS Mechanical ANSYS Mechanical is a finite detail evaluation tool for structural assessment, together with linear, nonlinear and dynamic studies. This laptop simulation product offers finite factors to version behavior and allows fabric fashions and equation solvers for a big style of mechanical design problems.

IV. STRUCTURAL ANALYSIS OF RADIAL ARM

4.1 STATIC ANALYSIS

Select static structural right click on → insert → pick rotational velocity and stuck assist → Select displacement → select required area → click on practice → positioned X,Y,Z factor 0 →

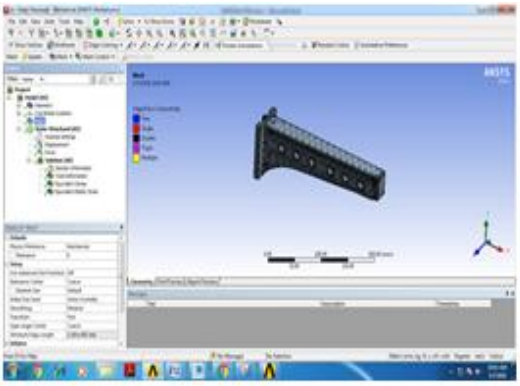

Fig. 3 Meshed model

Select force → pick required place → click on follow → enter rotational speed

Select solution proper click on → remedy →

Solution right click on → insert → deformation → general → Solution proper click on → insert → strain → equivalent (von-mises) →

Solution proper click → insert → stress → equivalent (von-mises) →

Right click on on deformation → examine all result

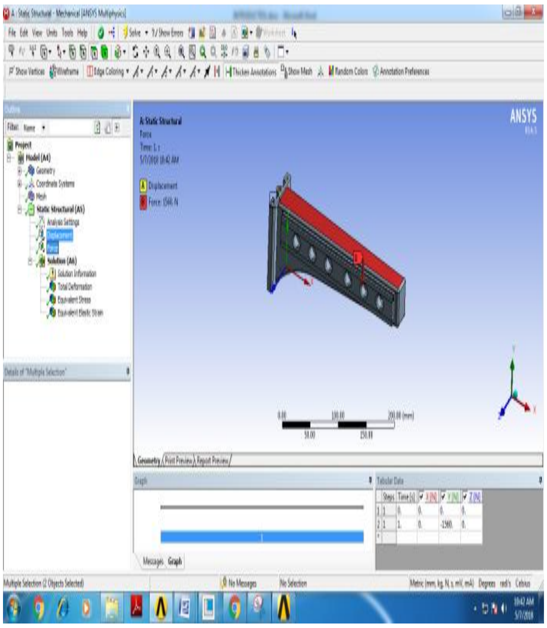

Fig. 5Boundary conditions

4.1.1 MATERIAL-MILD STEEL

- **TOTAL DEFORMATION**

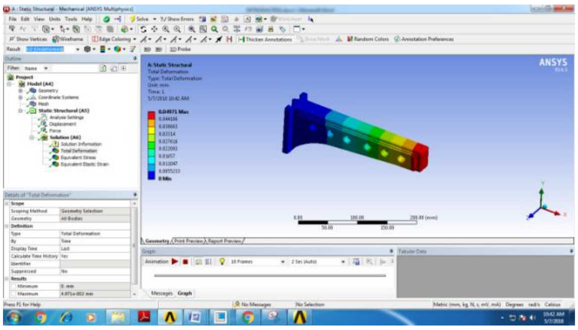

Fig. 6Total deformation of mild steel

- **VON-MISES STRESS**

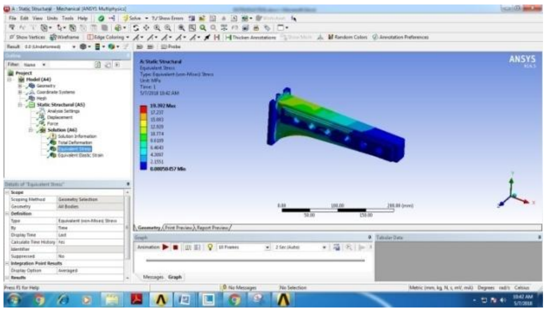

Fig. 7Von-mises stress of mild steel

- **VON-MISES STRAIN**

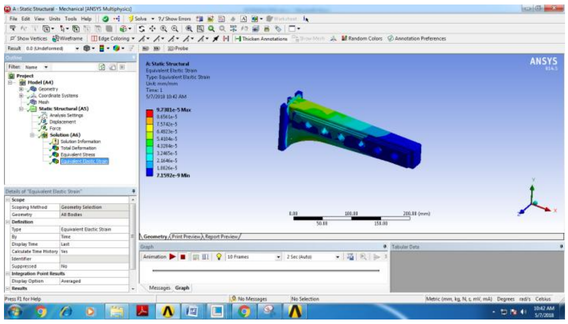

Fig. 8Von-mises strain of mild steel

FATIUGE ANALYSIS OF RADIAL ARM

- **SAFTEY FACTOR**

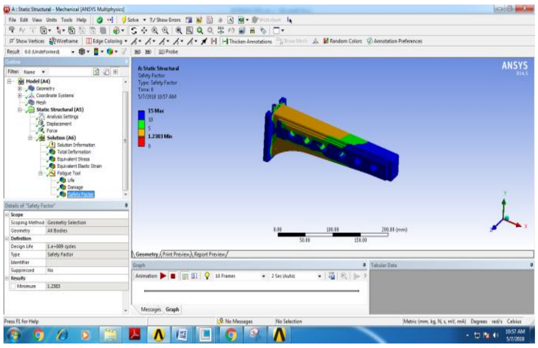

Fig. 9 Safety factor of EN 8 steel

4.2 MODAL ANALYSIS OF RADIAL ARM

4.2.1 MATERIAL-MILD STEEL

- **TOTAL DEFORMATION 1**

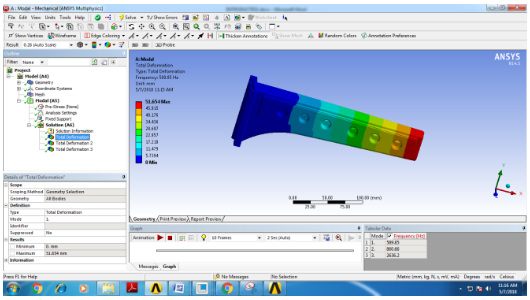

Fig.10Total deformation 1 of mild steel

- **TOTAL DEFORMATION 2**

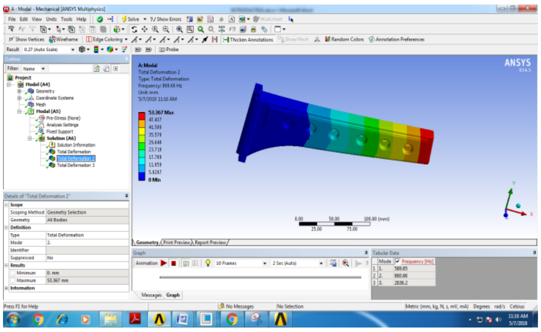

Fig. 11Total deformation 2 of mild steel

- **TOTAL DEFORMATION 3**

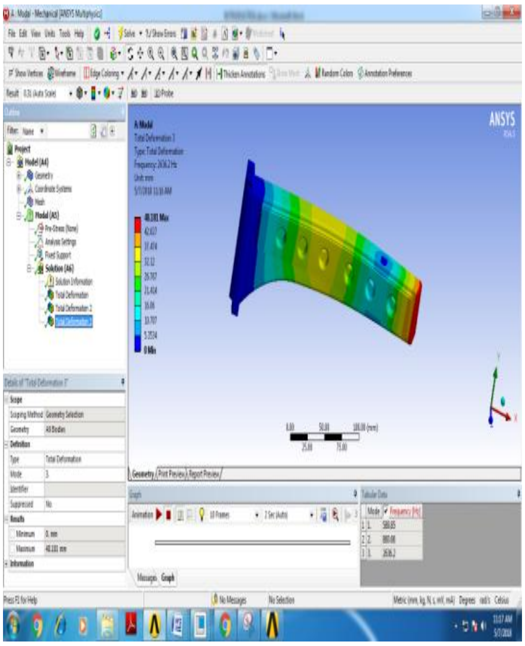

Fig. 12 Total deformation 3 of mild steel

MATERIAL –EN 8 STEEL

AT TIME -10 SEC

- **DEFORMATION**

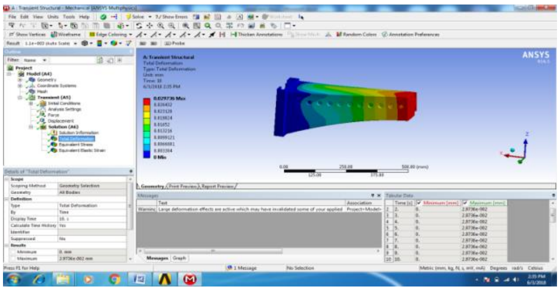

Fig. 13Transient analysis deformation EN 8 steel at 10sec

- **STRESS**

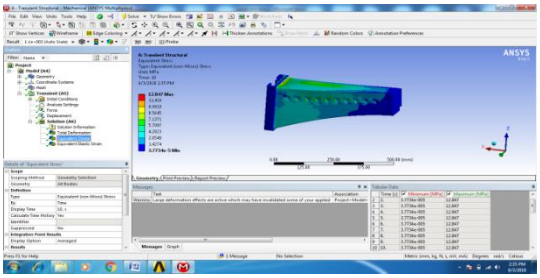

Fig. 14Transient analysis stress EN 8 steel at 10sec

- **STRAIN**

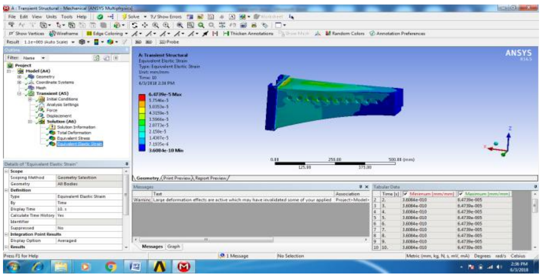

Fig. 15 Transient analysis strain EN 8 steel at 10sec

AT TIME -20 SEC

- **DEFORMATION**

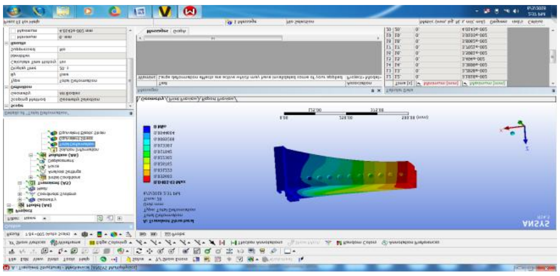

Fig. 16Transient analysis deformation EN 8 steel at 20sec

STRESS

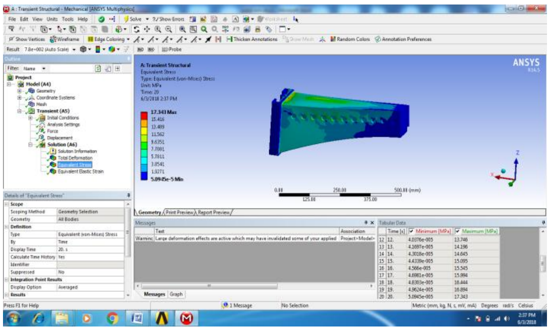

Fig. 17Transient analysis stress EN 8 steel at 20sec

- **STRAIN**

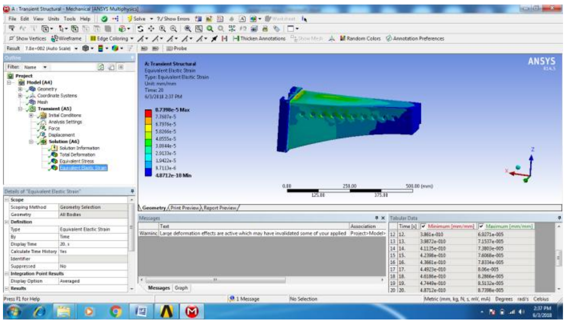

Fig. 18Transient analysis strain EN 8 steel at 20sec

AT TIME -30 SEC

- **DEFORMATION**

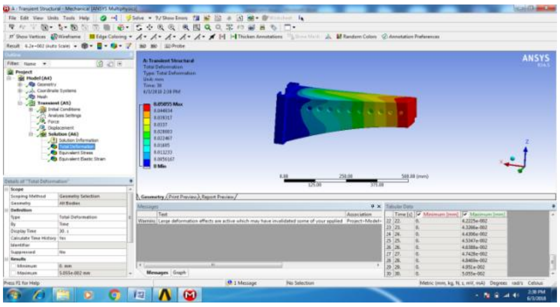

Fig. 19Transient analysis deformation EN 8 steel at 30sec

- **STRESS**

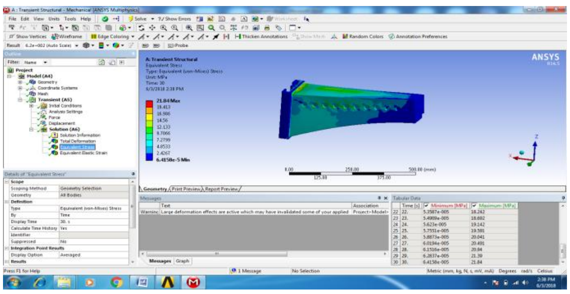

Fig. 20Transient analysis stress EN 8 steel at 30sec

- **STRAIN**

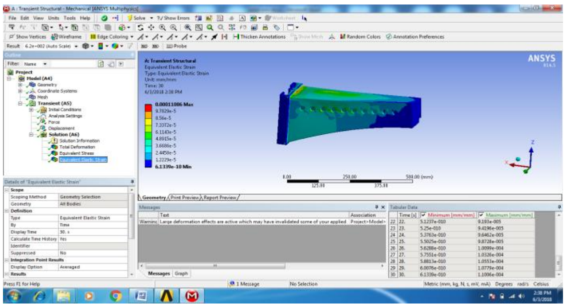

Fig. 20Transient analysis strain EN 8 steel at 30sec

STATIC RESULTS

Material	Deformation (mm)	Stress (N/mm^2)	Strain	Safety factor	
				Min	Max
Mild steel	0.04971	19.392	9.7138e-5	1.1113	15
EN 31 steel	0.046204	18.025	9.0514e-5	1.1956	15
EN 8 steel	0.044611	17.403	8.7393e-5	1.2383	15
Carbon steel	0.043815	17.092	8.532e-5	1.2608	15

MODAL ANALYSIS RSULTS

Material	Deformation 1 (mm)	Frequency (Hz)	Deformation 2 (mm)	Frequency (Hz)	Deformation 3(mm)	Frequency (Hz)
Mild steel	51.654	589.85	53.367	860.667	48.181	2636.2
EN 31 steel	49.64	566.85	51.286	827.1	46.302	2533.4
EN 8 steel	48.241	550.88	49.841	803.79	44.997	2462.0
Carbon steel	46.955	536.18	48.512	782.36	43.797	2396.3

TRANSIENT ANALYSIS RESULTS

MATERIAL	TIME (SEC)	DEFORMATION(mm)	STRESS(N/mm^2)	STRAIN
MILD STEEL	10	0.0037172	16.058	8.0922e-66
	20	0.0047538	20.554	1.0358e-5
	30	0.0057989	25.051	1.2624e-5
EN 31 STEEL	10	0.0033453	14.453	7.832e-5
	20	0.043856	18.949	9.5491e-5
	30	0.054267	23.446	0.00011815
EN 8 STEEL	10	0.0029736	12.847	6.4739e-5
	20	0.040143	17.343	8.7398e-5
	30	0.05055	21.84	0.0011006
CARBON STEEL	10	0.026019	11.241	5.6647e-5
	20	0.036427	15.737	7.9306e-5
	30	0.046834	20.234	0.00010196

V. CONCLUSION

3D modeling in CREO parametric software program and evaluation in ANSYS software program. In this thesis, static, fatigue and modal evaluation finished with extraordinary substances including moderate metal, EN 31 metal, EN eight metallic and carbon metal.

- Static evaluation is to determine strain, deformation and pressure.
- Modal analysis is to decide the deformation mode shapes with appreciate to frequencies.
- Fatigue evaluation is to decide the lifestyles of the component.
- By looking on the static evaluation the strain values are a lot much less for carbon metal examine with different substances.
- By looking at the modal evaluation the deformation values are plenty less for carbon steel examine with other materials.
- So it may be finish the carbon metal is higher material for radial arm.

REFERENCES

[1] Yi Xin Yang et al, (2010) Advanced Materials Research 126-128, 779.

[2] Tapas Kumar (2013) Experimental investigation of micro drilling operation of printed circuit board. B.Tech Thesis.
[3] Osamu Horiuchi et al, (2013) Bending of Drill and Radial Forces in Micro Drilling, Advanced Materials Research, 797, 642.
[4] Dilip Kumar Bagal, Experimental investigation and modelling micro drilling operation of aerospace cloth.
[5] I. Garitaonandia, M.H. Fernandes, J.M. Hernandez-Vazquez, J.A. Ealo, Prediction of dynamic behaviour for specific configurations in a drilling-milling machine based totally on substructing evaluation.
[6] Suman Chatterjee, Siba Sankar Mahapatra, Kumar Abhishek, (2016) Simulation and optimization of machining parameters in drilling of titanium alloys.
[7] E. Uhlmann, I. Dethlefs, F. Faltin, L. Schweitzer, (2015) Cutting and Drilling of Metals and Other Materials: A Comparison