International Journal of Advanced Technology in Engineering and Science
Vol. No.4, Issue No. 09, September 2016
www.ijates.com
ijates
ISSN 2348 - 7550

ANALYSIS OF CNC MACHINE: A REVIEW

Supriya A. Bhise[1], Pravin P. Kole[2], Munaf I. Attar[3], Sujit S. Malgave[4]

[1,2,3]Student, Department of Mechanical Engineering, AITRC, Vita, India

[4]Assistant Professor, Department of Mechanical Engineering, AITRC, Vita, India

ABSTRACT

Increase in rapid production significantly increased the usage and implementation of CNC system in industries. From last decade, the industries are focused on the improving superiority of CNC machines for their stability, failure reduction and weight reduction of machine. Accuracy of machine components depends on their strengths and stability. Applications of various forces may cause the deformation, failure of those components. The investigation is carried out to find out the displacement, maximum stresses and deformation and increase the natural frequency. In this study the 3D models of machine tools like machine bed, columns and spindle are created in software like CATIA V5 R18, CREO Parametric 2.0. The 3D model of machine tool is then imported in ANSYS software for various analyses like structural analysis, modal analysis, static and dynamic analysis. The present paper is a review of the analysis performed on various components of CNC machine structure.

Keywords- *CNC machine, deformation, dynamic analysis, static analysis, structural stability*

I. INTRODUCTION

CNC machines are those which are controlled by prepared program containing coded alphanumeric data. From the past year the CNC machines become widely in demand for modern manufacturing industries. CNC machines are used in industries to produce parts from a variety of materials with variety of cutting tool. For machining processes, CNC is widely used for lathe, drill press, milling machine, grinding unit, laser, sheet-metal press working machine, tube bending machine etc. The benefits of CNC machines are i) high accuracy in manufacturing ii) short production time iii) reduces human efforts.

CNC machine tool structure consists of components that are column, table, spindle, base, guide ways, saddle, head etc. The column, including the base is the main casting which supports all other parts of the machine. The column rests on a base that mainly carries the static forces acts on the machine. The table is the rectangular casting located on the base which supports the fixtures & the work piece while machining. It contains several T-slots for fastening the work or work holding devices. The spindle holds and drives the various cutting tools. It is a shaft, mounted on bearings supported by the column. The spindle is driven by an electric motor through a train of gears these all mounted within the column.

ijates
ISSN 2348 - 7550

Increasing competitiveness and for greater productivity are achieved by increasing the performance of machines. The structural components in the machine play a vital role in helping to achieve consistent performance. To have efficient working of machines, its components need to function with accuracy.

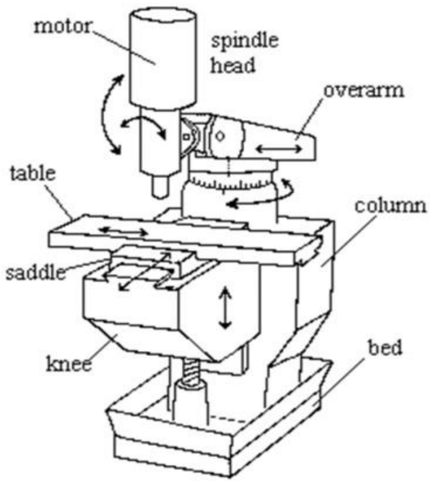

Fig.1. Structure of CNC machine tool

II. REVIEW ON WORK CARRIED OUT

The B. V. Subrahmanyam et al. [1] have made their paper on the static and dynamic analysis of machine tool structure. In this paper they studied about static and dynamic behaviour of three various types of machine tool structure i.e. milling machine, shaping machine, lathe machine. To perform an analysis on machines they used ANSYS which is Finite Element Analysis software. For analysis of machine structure they fixed the base of machine and the load taken as the weight of machine itself. In static analysis they found out equivalent stresses and stresses in X, Y, Z direction, also they found the deformation of machines due to its self-weight. From static analysis they conclude that the equivalent stress generated in the lathe machine structure was higher than other machine structures also they state that the deformation along X-direction was high for all machines. For dynamic analysis they used modal analysis method to determine the vibrational characteristics of machine. They used six natural frequencies modes for each machines for finding the deformations. For milling machine the maximum deformation was 0.079172mm occurred at 17.711Hz, for shaping machine 0.117925mm deformation occurred at 32.247Hz and for lathe machine 0.254419mm deformation takes place at 20.662Hz. From these results they conclude that as the natural frequencies increases the deformation also increases.

The C. C. Hong et al. [2] worked on five axis turning-milling complex CNC machine. This paper provides details about computer aided engineering (CAE) for heavy industries. They have done research analysis with CAE software on secondary and primary shaft system, machinery bed with respect to static construction, stresses & deformation. They were used commercial computer software SOLIDWORKS^{R} 2014 simulation software. In this study, the analysis done without considering inertia force, damping force and impact force also nonlinear state. They completed work with mathematical model in matrix form as

[M] {u} = {f}

Where; M = material stiffness constant,

u = displacement vector

f = external load factor

They worked on machinery bed with five types of clamp with 36 boundary conditions. They provide different meshing size for analysis and find maximum displacement values for machinery bed. From this paper it is clear that external load value below 10Mpa is safe. They conducted two types of analysis, 1) with gravity force, 2) with gravity force & external load. The study concluded with four types of boundary conditions if machinery bed conditions. They succeed to reduce weight of CNC machine and maintain good stress to resist external load. The maximum values of stress obtained due to gravity force effect, so CNC was in safely condition.

The B. Malleswara et.al [3] worked on 3D modelling software CATIA and Computer Aided Engineering [CAE] software ANSYS. To reduce the weight of machine bed by its dynamic and static analysis, during which they don't change its structural rigidity and accuracy of machine tool. In this paper, they given information about vertical mill whose spindle is vertically mounted and table is lowered and raised. They further define two categories of vertical mill: i) the bed mill and ii) the turret mill. They mentioned in turret mill, spindle is stationary and table is moved perpendicular and parallel to spindle axis for successful cut. While in bed mill spindle moves to its own axis and table moves only in perpendicular direction. They carried modelling in CATIA and further they worked on ANSYS with boundary conditions i.e. fixed constraint and material properties like G15, G40 and G70 were applied. In this paper, they compared initial modelling with optimized one. Due to which their research showed that grade G15 was better than other two grades i.e. G40 and G70. In this study they compared 1g static and modal analysis with original and optimized modal. They conclude that the displacement, vonmesis stresses and strains and principle stresses are minimized for G15. The natural frequency and weight of machine bed was reduced. The results made by them is showed by following table,

Table No. 1. Displacement, Vonmesis stresses, weight of milling machine bed

	Initial Model			Optimized Model		
Material Properties	Displacement	Vonmesis Stress	Weight of Machine	Displacement	Vonmesis Stress	Weight of Machine
G15	0.024155	26.1305	1056.3	0.03532	33.2082	1040.2
G40	0.03547	28.9518	1106.5	0.05919	68.2932	1094.7
G70	0.0458	31.492	1147.1	0.07554	64.134	1138.9

The Shihao Liu et al. [4] have made their paper which deals with the study and analysis of column of gantry machining centre. For obtaining light weight structure of column with increasing natural frequencies, there was bionic design for column structure established. In which the bionic design for stiffener plate was made first, which was arranged in the column structure. They used the arrangement of stiffener plate in the shapes of '#' or 'X' which is traditional method of arrangement. The arrangement is made by using the principle of gingko root system. The gingko root system consists of tap root and branched root structure similarly in the column main stiffener plate and subprime stiffener plate was arranged. Also the reinforcement plates were added on load bearing guide and load area of top part. They made three types of bionic structure using cad software i.e. A, B and C. these three models are prepared in CAD software and then imported into ANSYS software. They applied load on the top of the column was 69217N, on the middle of the column was 118216N and during machining maximum load was 40000N applied in X and Y directions. From these conditions they obtained the results as follows,

Table No. 2. Results for three bionic structures A, B and C

	A	B	C
Deformation reduced	15.05%	11.48%	16.38%
Mass reduced	3.96%	2.74%	1.32%
Maximum stress reduced	5.29%	10.22%	8.24%
Increased natural frequencies	5.78%	6.22%	6.32%
Manufacturability	Best	Good	Complicated

From above results the authors concluded that the type B is suitable bionic design for column structure because type B achieve light weight structure with reducing maximum stresses and increasing natural frequencies.

Rishikesh B. Kamthe et al. [5] has made their paper on design analysis and testing of spindle of CNC machine. In this paper they carried out three methods for testing the spindle of CNC machine are as follow i) mathematical ii) Analysis in CAE software iii) Experimental testing. In mathematical testing they find out maximum cutting force by using formula;

$$Fc = Ks.f.d$$

Where; Ks is specific cutting resistance,

f is feed rate in mm/rev and d is depth of cut (mm).

They found various parameters for turning operation i.e. feed=0.15mm/rev, radial force=2207.25N, axial force=1274.34N, torque=77.254Nm, power=6.62Kw. They used Macaulay's method for finding out maximum deflection i.e. 0.003mm. For CAE analysis they used ANSYS and Creo parametric 2.0 software for modelling purpose. They create complete assembly of spindle and evaluate result by giving various boundary conditions that are, forces in Y direction 137.24N, 2124.7N, 2207.25N and 4469.2N. From this analysis they found values of maximum deformation = 3.5711x10^-3mm, equivalent elastic strain = 3.5711x10^-5 and equivalent stress =6.3875MPa. From the experimental study on spindle, they found the safe condition for the operator. They state that noise level of 87.4dB, temp at 44.4c and the speed 6174 rpm is good for the operator.

Hareesha [6] have made his paper on static and dynamic analysis of CNC milling spindle. In this paper he carried out two methods for testing of CNC milling spindle are as follows i.e mathematical and analysis in CAE software. He used two materials for spindle is alloy steel and manganese steel. The spindle was supported by two sets of angular contact ball bearing which was installed back to back on the front end and rear end with respective to diameter for bearing bore was 45mm and 40mm. A static analysis calculate the effect of steady loading on a structure while ignoring inertia and damping effect was assumed for determine the stress strains and forces in structure. From the mathematical testing they found deflection by using formula.

$$\delta = P_z * \{1/S_A * (a+L/L)^2 + 1/S_B (a/L)^2 + a^2/3e (a/I_a + L/I_L)\}$$

Where; P_z=load (N),

δ=Deflection at point application of P_z (mm),

$\delta 1$=deflection due to radial yielding of the bearing (mm),

$\delta 2$=deflection due to elastic bending of the spindle (mm),

a =length of the overhanging portion of the spindle form the effective support of bearings,

S =P/δ overall stiffness of the spindle unit (kgf/mm),

S_A=Radial stiffness of the load point,

S_B=radial stiffness of the bearing away from load point,

I_a=moment of inertia of overhang portion of spindle,

I_L=moment of inertia of spindle section between bearings,

L=Bearing span (mm).

Axial stiffness of front and rear bearing S_A=260000N/mm and S_B =230000N/mm.

They found deflection by using above formula i.e. δ=0.01284mm. Also they found bearing span length by using formula

$$L_o = [6\,EIL\,(\frac{1}{SA} + \frac{1}{SB}) + (\frac{6EIL}{axSA})\,Q]^{1/3}$$

Where; Lo=static optimum bearing span,

Q=trial value for iterative determination of Lo=4a.

They found value i.e. Lo=114.23mm. Also they found spindle stiffness (k) = $\frac{Pz}{\delta}$, they found value i.e. k=87x10^3N/mm. For analysis they created model as solid 10 nodes, 92 elements, whereas the bearing front and rear side using COMBIN 14 element. They create complete design and evaluate the result by giving various boundary conditions that was applied on spindle nose end at cutting force 1120N. The nodes at both bearing sets were constrained fully to have no degree of freedom. They found value of deflection I.e. δ=12.66μ.

Migbar Assefa [7] has made her paper deals with study and analysis of static and dynamic rigidity requirement of the column of machine. In this paper the analysis done on 70 column structure. The purpose of paper was to evaluate static and dynamic rigidity for improving machining accuracy and better surface finish. The parameters used for the modelling of column structure are tapers, apertures, aspect ratio and apertures size. They used the shell type column for analysis purpose. Shell63 element was used for analysis. 3D model is firstly made in CAD software and then imported into the ANSYS 10.0 software and then static and modal analyses were done. The material used for the column was cast iron. In static analysis they performed torsional analysis in which they give forces P1=P2=1560N subjected to column to twisting moments and also performed bending analyses in which they provide P1=P2=1560N acts at corners of column. The modal analysis is performed to calculate the natural frequencies and mode shape for the column. From this study they conclude that orientation of apertures and aspect ratio does not affects to the static and dynamic rigidity. But they observed that size of apertures affects to the rigidity of column. The size of apertures also affect to the deformation near aperture and natural frequencies.

The Swapnil Killedar et al. [8] worked on design evaluation using FEA for machine frame. In this paper, they decided for fabrication of machine frames. Hence the machine frame is modified in software CATIA V5 R18. This model was meshed by using hypermesh software. For special purpose machines (SPM) they selected fabricated square tube. The material selected by them was gray cast iron (CI). The square tube was mesh with 2D-quad element and solid parties with second order 10 noded tetra element. They introduce why CI was chosen for fabrication purpose. Their study showed that for existing machine frame, machine frame model in CATIA V5 R18 software. To create a mesh model they used 2D quad and second order 10 noded tetra element. The material used by them was CRCA steel. After that they applied boundary condition on entire machine bed was static load of 50000N. Subsequently they applied on diagonal machine frame. The material used by them was cast iron SG300 and obtained the following results as follows.

Table No. 3. Maximum Displacement and stress of variant

Variant	Maximum Displacement	Maximum Stresses
Existing	0.8	242.43
Diagonal	0.13	208.13
Parabola	0.13	377.03
Trapezoidal	0.12	248.43
Casting	0.14	62.46

From above result table they conclude that variant diagonal has low maximum stress of 208.13MPa which is under safe stress calculated as 244.8MPa as compared to existing variant and hence the study shows that manufacturing cost also decreased for machine frame.

The Sujeet Ganesh Kore et al. [9] worked on design and analysis of machine tool based on bionics. In the nature there are large number of structure available which was used as prototype and creative approach for solving engineering problem. Structure bionic is a new method for mechanical design which is very effective and progressive. This method can help in nature. In this study they mentioned structural bionics was made a great change or process in aircraft, watercraft, micro-air vehicles (MAV) and other design field. It is also helpful for light weight design or structure. This structure increases stiffness and toughness. The structure or machine investigation is carried out to reduce the weight, stresses and displacement. In this paper 3D model was used in existing bed model and optimized bed model. They states, bamboo mesostructure was best example in nature which we can use as structural bionics using the biological structure. They can achieve optimum design with good stiffness and strength. The bionics system is based on structure, process, function, organization etc. In this study the three stages were discussed for design process.

i. Requirement identification: It gives the information of general design procedure for bionic requirement process.
ii. FEM Analysis: In this analysis they learn about design and analysis in FEM simulation.
iii. Experiment verification: After the FEA analysis experiment verification were formed.

In this paper modelling of lathe bed was mainly two types: i) Existing bed model, ii) Optimized bed model. This two modelling method results was calculated in weight, stresses, displacement and model analysis for CI material and structural steel material. While comparing the above results they conclude that optimized model selected as the best model because of low stresses, low weight, less displacement, high natural frequencies.

CONCLUSION

This literature study is mainly focused on the machine stability, structural analysis in static and dynamic loading. In the literature, the analysis performed on the machine tool has been reported. The suggested work on the analysis of CNC machine tool has done successful in weight reduction, increase in natural frequencies, reduction in deformation etc. This study will help for enhancement and development of machine tool structure. The machine structures can be improved by using bionic structure design, modal analysis, static and dynamic analysis and by validation of results.

REFERENCES

[1] B.V. Subrahmanyam, A. Shrinivasa Rao, S.V. Gopala Krishna, CH. Rama Krishna, Static and Dynamic analysis of machine Tool structures, IJRMET, Vol. 4, Issue Spl-1, 2013- 2014, 14-18

[2] C.C. Hong, Cheng-Long Chang, Chien-Yu Lin, Static structural analysis of great five-axis turning–milling complex CNC machine, ELSEVIER, 2016, 1-14

[3] B. Malleswara Swami, K.Sunil Ratna Kumar, Design and Structural Analysis of CNC Vertical Milling Machine Bed, IJAET, Vol. 3, Issue IV, 2012, 97-100

[4] Shihao Liu, Wenhua Ye, Peihuang Loua, Weifang Chen, Jungui Huangb , Lili Xiao, Bionic design for column of gantry machining centre to improve the static and dynamic performance, Shock and Vibration 19, 2012, 493-504

[5] Rishikesh B. Kamthe, Chetan S. Magdum, Aditya S. Gandhi, Pranav A. Naik, P. D. Gharge, Design, Analysis and Testing of Spindle for high speed CNC Lathe Machine, IJIRSET, Vol. 5, Issue 5, 2016, 7282-7289

[6] Hareesha, Static and Dynamic Analysis of a CNC Milling Spindle, IRJET, Vol. 2, Issue 7, 2015, 222-228

[7] Migbar Assefa, Modal Analysis of Machine tool column using Finite Element Method, IJMAIMME, Vol. 7, 2013, 699-708

[8] Swapnil Killedar, Sanjay Deodas, Swapnil Kulkarni, Design Evaluation Using Finite Element Analysis for Machine frame to meet Compliance for Structural Integrity, IJAERS, Vol. 3, 2014, 72-76

[9] Sujeet Ganesh Kore, M I Sakri, L N Karadi, Design and Analysis of a Machine tool Structure Based on Structure Based on Structural Bionics, IJMERR, Vol. 3, 2014, 731-737