PREDICTION OF VIBRATION INDUCED DISPLACEMENT AND ITS EFFECT ON TOOL WEAR IN TURNING USING 3D FINITE ELEMENT SIMULATION

Abstract: This paper presents the three dimensional (3D) finite element analysis to predict the workpiece displacement amplitude and tool wear in face turning of AISI 1040 steel under dry machining condition. 3D vibration induced turning models have been developed and validated by comparing with experimental results so as to identify the correlation. Extensive experimentation and investigators carried out on CNC lathe machine and the details are reported in [1] have been considered for present work. Commercial FE programme is used for simulating turning process. The effect of workpiece displacement due to vibration on the tool wear is critically evaluated.

Key words: Vibration amplitude, 3D finite element simulation, tool wear, face turning.

1. INTRODUCTION

In metal cutting as a result of the cutting motion, cutting tool wear will be influenced by cutting parameters, cutting force, and vibrations, etc. But the effects of vibrations have been paid less attention. The dynamic phenomena of cutting tool induced by the interaction of elastic system in the cutting process causes the relative displacement between tool and work piece, which generates the vibration during machining [2]. Vibrations in the turning process can be a good way to monitor online growth of the tool wear in turning and, therefore, it can be useful for establishing the end of tool life in these operations [3]. This is very important when the goal is to monitor the cutting process in real time and to establish automatically the end of tool life [4]. Therefore, it is necessary to study on effect of cutting tool vibrations during the machining. Therefore, it is necessary to study on effect of cutting tool vibrations during the machining [5]. In modern machining processes [6] due to continuous demand for higher productivity and product quality asks for better understanding and control of the machining process. A better understanding can be achieved through finite element modelling and simulations of machining process. Thus, in recent years, finite element method has particularly become the main tool for simulating metal cutting processes. Predictions of the physical parameters such as displacement amplitude, tool wear, cutting forces, tool chip interface temperature and stress distributions accurately play a pivotal role for predictive process engineering of machining processes. Simulations of various machining operations using the finite element method have been reported over the last three decades; in Reference [7] a collection of such papers can be found. Vytautas Ostasevicius et al. [8] developed a finite element model of the vibration milling tool and verified experimentally. Rimkevičienė J et al. [9] reported a combined numerical experimental approach for the reduction of surface roughness based on exciting higher-order transverse modes in the vibration turning tool. Tuğrul Özel et al. [10] proposed a FEM model and simulation strategy for orthogonal cutting of AISI 1045 steel is by using dynamics explicit Arbitrary Lagrangian Eulerian method in simulating plastic flow around the round edge of the cutting tool and eliminates the need for chip separation criteria. W. Grzesik [11] created a FEM simulation model in order to obtain numerical solutions of the cutting forces, specific cutting energy and adequate temperatures occurring at different points through the chip/tool contact region and the coating/Substrate boundary for a range of coated tool and uncoated tool materials and defined cutting conditions. A. G. Mamalis et al. [12] results reported simulation of high speed hard turning when using the finite element method for both orthogonal and oblique cutting models. Yung-Chang Yan et al. [13] developed a methodology to predict the tool wear evolution and tool life in orthogonal cutting using FEM simulations. D. Umbrello et al. [14] deals with the 3D FE simulation of cutting processes in order to estimate tool wear development during turning operations and proposed a new subroutine for tool wear
calculation and tool mesh and geometry updating is has been proposed in a commercial 3D code. Maňková et al. [15] identified the influence of cutting parameters in hard turning of hardened steel with hardness of HRC 55 with mixed oxide ceramic inserts. Zhang Wei et al. [16] presented the construction of a 3-D finite element simulation of turning process with an updated Lagrangian method. From literature, it is understood that most of the research work is limited to experimental results. But numerical modelling of turning process can provide a useful data for better understanding of the process. Limited literature available on this topic and this is mainly focused on 2D problems while 3D models are rather rare in the relevant literature. A considerable amount of research has focused on 2D finite element simulations for turning, but studies on 3D finite element simulations (FES) for turning process are not widely available. Major drawback of a cutting simulation is that it does not provide direct information on the increase of tool wear, as opposed to the experimental approach. Nevertheless, it is expected that the tool wear growth (crater and flank wear) is dependent on the relative displacement amplitude due to vibration, cutting temperature, contact stresses, and sliding velocity produced during cutting. In order to overcome the above mentioned limitations and to improve the performance of finite element simulations 3D FE models can be effectively used to simulate actual machining processes. Thus, an extensive investigation on the effect of displacement amplitude of workpiece due to vibration on tool wear in face turning processes is important to realize an accurate finite element simulation (FES) method for the vibration induced machining (VIM).

2. PROPOSED METHODOLOGY IN PRESENT STUDY

In the present work, tool – chip and tool – work piece interfaces is strongly influenced by the displacement amplitude due to vibration, tool wear, cutting temperature and relative sliding velocity at the interface. Therefore, understanding of the tool wear behavior and the capability of predicting it is the key to successful process control and optimization. To achieve this goal, the effect of different cutting variables and process mechanics has been investigated using finite element method (FEM). Overview of proposed methodology in present study is presented in figure 1. Hence ultimate objective of the present study is to develop a 3D finite element simulation based methodology to predict the evolution of displacement amplitude and tool wear in vibration induced face turning. Implementation of machining model(s) in the commercial FEM code (DEFORM®-3DV6.1) that relates the displacement amplitude and tool wear during machining to the predicted process variables. This approach consists of development of tool wear model for the specified tool–work piece pair via a calibration set of tool wear cutting tests in conjunction with FE cutting simulations. Second part includes the validation FS simulation results with experimental results considered in the literature. Turning operation is a steady-state process when continuous chip is formed. The implementation of tool wear estimation is relatively easier and studied first. A tool wear estimation models have been developed for turning operation as part of the work.

Fig. 1. Overview of proposed research work

Present study is planned to carry out in following stages, first stage concentrates on chip formation analysis in which a new chip formation modeling method for continuous steady state chip formation is developed. It can simulate the entire chip formation process from initial chip formation, chip growth to steady state by making use of Arbitrary Lagrangian Eulerian technique. Second stage concentrates on tool chip interface temperature analysis. In order to save the calculation time and the temperature distribution in the cutting tool at thermal steady state is studied by performing pure heat transfer analysis. Present analysis is confined to only to study the temperature distributes in the cutting tool. Last stage involves in modeling the displacement amplitude and its corresponding predicted load. Through previous stages, the present commercial FE code also predicts the tool wear along with normal stress, sliding velocity and tool temperature at steady state according to the test conditions.

3. THREE DIMENSIONAL (3D) FINITE ELEMENT METHOD SIMULATION OF TURNING PROCESS

The 3D model, implemented in DEFORM 3D v6.1, is reported in figure 2 where it is possible to see the work
piece with the growing chip and the tool. The tool, a rigid object meshed with more than 1,00,000 elements, is oriented according to the cutting angles set in experimental test and reported in table 2 and it moves along the feed direction. Work piece considered as a rigid-plastic object meshed with more than 30,000 elements, is fully constrained on the lower and lateral sides so it cannot move. On the same faces thermal boundary conditions are set so to simulate the heat diffusion. The thermal exchange between tool and chip is regulated by heat diffusion relationship whose characteristic parameters are reported in table 1. The friction is modeled considering a shear factor equal to 0.82. An adaptive re-meshing scheme is implemented to optimize between the computational time and accurate prediction. The top and back surfaces of the tool are fixed in all directions. The workpiece is constrained in vertical (Z) and lateral (Y) directions on the bottom surface and moves at the cutting speed in the horizontal direction (X) toward the stationary tool. Tool wear estimation models for turning mainly composed of chip formation analysis, heat transfer analysis, wear calculation procedure and tool geometry updating, as shown in figure 2.

In this work, a commercial software code, DEFORM 3D and explicit dynamic ALE modeling approach is used to conduct the FEM simulation of oblique cutting process. The chip formation is simulated via adaptive meshing and plastic flow of work material. Therefore, there is no chip separation criterion is needed. In this approach, the elements are attached to the material and the un-deformed tool is advanced towards the work piece. The programme menus are designed in such a way that they allow the user to minimize the model preparation time.

The thermo-mechanical FEM simulation schemes (models) are created by including tool and work piece thermal and mechanical properties, boundary conditions, contact conditions between tool and work piece as shown in figure 2 and as per the conditions given in table 1. As discussed in literature, this research is aimed to the 3D numerical prediction of tool wear using the knowledge acquired in 2D studies. Thus, in order to validate 3D predictions for both flank wear and displacement amplitude due to vibration, experimental details are reported in [1] have been considered for the present research work. Tool rejection criterion developed based on both ISO 3685 and ISO 10816 standards for evaluate the cutting tool condition in turning.

3.1. Material flow properties
According to a comparative analysis described by [16], Johnson-Cook model is one of the most convenient material models which also produce excellent results describing the material behaviour and chip formation. In this work, the Johnson-Cook constitutive model was used to predict the post-yield behaviour of AISI 1040 steel is given by equation 1 is used.

$$\sigma = \left( A + B \varepsilon^\theta \right) \left[ 1 + C \ln \left( \frac{\dot{\varepsilon}}{\dot{\varepsilon}_0} \right) \right] \left[ 1 - \left( \frac{\theta - \theta_{room}}{\theta_{melt} - \theta_{room}} \right)^m \right]$$  \hspace{1cm} (1)

where $\dot{\varepsilon}$ is plastic strain rate, $\varepsilon$ is equivalent plastic strain, $\dot{\varepsilon}_0$ is reference strain rate, $A$ is initial yield stress, $B$ is hardening modulus, $C$ is strain rate dependency coefficient, $n$ is work hardening exponent, $m$ is thermal softening coefficient, $\theta$ is the process temperature, $\theta_{melt}$ is the melting temperature of the workpiece and $\theta_{room}$ is the ambient temperature ($35^\circ$C).

![Fig. 2. FE modelling of displacement amplitude due to vibration in turning](image)

In this work,  a commercial software code, DEFORM 3D and explicit dynamic ALE modeling approach is used to conduct the FEM simulation of oblique cutting process. The chip formation is simulated via adaptive meshing and plastic flow of work material. Therefore, there is no chip separation criterion is needed. In this approach, the elements are attached to the material and the un-deformed tool is advanced towards the work piece. The programme menus are designed in such a way that they allow the user to minimize the model preparation time.

The thermo-mechanical FEM simulation schemes (models) are created by including tool and work piece thermal and mechanical properties, boundary conditions, contact conditions between tool and work piece as shown in figure 2 and as per the conditions given in table 1. As discussed in literature, this research is aimed to the 3D numerical prediction of tool wear using the knowledge acquired in 2D studies. Thus, in order to validate 3D predictions for both flank wear and displacement amplitude due to vibration, experimental details are reported in [1] have been considered for the present research work. Tool rejection criterion developed based on both ISO 3685 and ISO 10816 standards for evaluate the cutting tool condition in turning.

3.1. Material flow properties
According to a comparative analysis described by [16], Johnson-Cook model is one of the most convenient material models which also produce excellent results describing the material behaviour and chip formation. In this work, the Johnson-Cook constitutive model was used to predict the post-yield behaviour of AISI 1040 steel is given by equation 1 is used.

$$\sigma = \left( A + B \varepsilon^\theta \right) \left[ 1 + C \ln \left( \frac{\dot{\varepsilon}}{\dot{\varepsilon}_0} \right) \right] \left[ 1 - \left( \frac{\theta - \theta_{room}}{\theta_{melt} - \theta_{room}} \right)^m \right]$$  \hspace{1cm} (1)

where $\dot{\varepsilon}$ is plastic strain rate, $\varepsilon$ is equivalent plastic strain, $\dot{\varepsilon}_0$ is reference strain rate, $A$ is initial yield stress, $B$ is hardening modulus, $C$ is strain rate dependency coefficient, $n$ is work hardening exponent, $m$ is thermal softening coefficient, $\theta$ is the process temperature, $\theta_{melt}$ is the melting temperature of the workpiece and $\theta_{room}$ is the ambient temperature ($35^\circ$C).

<table>
<thead>
<tr>
<th>Oblique cutting parameters: rake angle (°) -5, clearance angle (°): 5.</th>
<th>Feed rate (mm rev⁻¹)</th>
<th>Depth of cut (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cutting speed (N rpm)</td>
<td>0.08 - 0.48</td>
<td>0.5 - 0.15</td>
</tr>
<tr>
<td>538</td>
<td>0.08 - 0.48</td>
<td>0.5 - 0.15</td>
</tr>
<tr>
<td>838</td>
<td>0.08 - 0.48</td>
<td>0.5 - 0.15</td>
</tr>
<tr>
<td>1135</td>
<td>0.08 - 0.48</td>
<td>0.5 - 0.15</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Work piece material: AISI 1040 steel of size (150 x 150 mm)</th>
<th>CNC tool properties: DSKLA 402, uncoated, VMC as base material. Tool holder: EDJNR.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coefficient of thermal expansion (μm/m°C)</td>
<td>Coefficient of thermal expansion (μm/m°C)</td>
</tr>
<tr>
<td>11 (at 20°C)</td>
<td>49 (at 1000°C)</td>
</tr>
<tr>
<td>Density (g/cm³)</td>
<td>Density (g/cm³)</td>
</tr>
<tr>
<td>7.8</td>
<td>15</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>Poisson’s Ratio</td>
</tr>
<tr>
<td>0.3</td>
<td>0.2</td>
</tr>
<tr>
<td>Specific heat (kJ/kg°C)</td>
<td>Specific heat (kJ/kg°C)</td>
</tr>
<tr>
<td>442.6</td>
<td>384</td>
</tr>
<tr>
<td>Thermal conductivity (W/m°C)</td>
<td>Thermal conductivity (W/m°C)</td>
</tr>
<tr>
<td>4.7</td>
<td>4.6</td>
</tr>
<tr>
<td>Young’s modulus (GPa)</td>
<td>Young’s modulus (GPa)</td>
</tr>
<tr>
<td>200</td>
<td>900</td>
</tr>
</tbody>
</table>

Table 1 Test conditions for experimental investigation
4. RESULTS AND DISCUSSIONS

As part of the present research work, finite element simulations have been carried out for 27 cases of experiments consisting of 3 different feed rates, cutting speeds and depth of cuts. Table 2 presents results of both experimental and predicted FE simulation values at different test cutting conditions under which simulations are carried out. FE predicted results are verified with experiment results. In FE simulations, the resultant field variables have been extracted after 5 milliseconds (ms) of cutting time. This short cutting time is assumed sufficient for the field variables to reach a steady-state. Effect of displacement amplitude on tool wear and analysis is done for all test conditions. Effect displacement on tool wear is studied and correlation is established. Results of test conditions TC 1-1, TC 2-5 and TC 3-9 are discussed in detail whereas table 2 gives the details of entire experimentation.

4.1. Case 1 (TC1-1: Rotational speed 538 rpm, feed at 0.08mm/rev, depth of cut 0.5 mm)

The 3D models, implemented in Deform 3D v6.1, are reported in figure 3, figure 4 and figure 5 where it is observed that workpiece with the growing chip and the tool. Figure 3 presents the 3D FE modeling of oblique metal cutting process for TC 1-1 which shows the predicted displacement amplitude due to vibration during cutting in the work piece and the chip separation.

In agreement with experimental results [1] the chip is a continuous one due to material model and cutting condition chosen. Displacement predicted from FEM simulation are also maintains good agreement with the results of experiment. In addition, chip separation achieved by continuous re-meshing which is triggered by sharp tool penetration is presented in figure 3(a). It is a known fact that the friction between the tool and the workpiece is little when the tool is new (sharp) and accordingly the amplitude of vibration will also be low. Due to the presence of dominant signals vibration is measured in the feed direction (Z direction). Predicted load in Z direction each test condition are presented in figure 3(b), 4(b) and 5(b) are showing the good correlation with dominant signals vibration is measured in the feed direction (Z direction) as part of the results discussed in [1]. Figure 3(b) shows a typical cutting force versus time increment, where it can be seen that after some point steady-state is reached. Therefore all the results presented in this work are gathered under steady state conditions. Pure heat transfer analysis is performed after chip formation analysis and it is shown in figure 3(c). At the end of the chip formation analysis in turning operation, temperatures at nodes inside the cutting tool are climbing while those at tool-chip interface nodes approach steady state as shown in figure 3(c). As mentioned earlier, this analysis is presents the temperature distributes in the cutting tool. Figure 3(d) presents the profile of 3D tool flank wear after simulation.

4.2. Case 2 (TC 2-5 Rotational speed 836 rpm, feed at 0.4 mm/rev, depth of cut 0.8 mm)

Results of test condition 2-5 are given in figure 4. In figure 4(a) an increase in displacement amplitude due to vibration is clearly found. As the length of the workpiece decreases, the stiffness of the workpiece increases and a corresponding increase in vibration will occur is [17] clearly observed in figure 4(b).

This increased level of vibration due to increase in tool wear. At the end of TC 2-5 condition, temperatures at nodes inside the cutting tool are still climbing whereas at tool-chip interface nodes approach steady state as shown in figure 4(c). The highest temperature is at the rake - chip interface and most part of the tool is still at room temperature. Figure 4(d) shows the simulated tool flank wear corresponding to middle point of the test for each of the cutting conditions for AISI 1040. It is observed that as long as values of VB≤0.14mm the tool flank wear developed gradually without any BUE formation.
near VB=0.2 mm formation of workpiece built-up on the edge and material smearing on the flank wear face can be observed. It is observed that as long as values of VB<0.25 mm the tool flank wear developed gradually with significant BUE formation.

4.3. Case 3 (TC 3-9 Rotational speed 1135 rpm, feed at 0.8 mm/rev, depth of cut 1.5 mm)

FEM simulation results corresponding to test condition TC 3-9 is presented in figure 5. In figure 5(a) a further increase in vibration amplitude is found when compare to previous cases and this is because of the increasing friction between the workpiece and cutting tool which is due to increase in tool wear.

Any displacement amplitude beyond this value is showing excessive vibration which increases tool flank wear and reducing the tool life. Figure 5(b) shows a highest predicted load in all test cases presented in results and discussions. Result of pure heat transfer analysis for TC 3-9 is shown in figure 5(c).

Fig. 5. FEM simulation results corresponding to test condition TC 3-9

Table 2. Experimental and FE simulated values

In figure 5(c) it is observed that, temperatures at nodes inside the cutting tool are climbing while those at tool-chip interface nodes approach steady state condition. Figure 5(d) gives the profile of 3D tool flank wear after simulation. In figure 5(d) flank wear (VB) values are found to be more than 0.3 mm. Flank wear value more than 0.3 mm is clear indication of the end of the tool life as per ISO 3685 and rejection criterion adopted in present study. From table 2, it is found that according to ISO 10816 displacement amplitude values up to 20 μm do not have any effect on tool flank wear. Tool flank wear is found to be affected by the measured displacements in the range between 20 and 60 μm. A displacement value beyond 60 μm is not a acceptable as per ISO 10816. From table 2 it is clear that in all conditions i.e, TC 1-1 to TC 1-9 displacement values is found to be less than 60 microns and VB<0.3 mm.

With increase of flank wear, displacement amplitude due to vibration is found to be increase in all test conditions. As it can be observed from table 2, the tool life is reduced significantly with the increasing vibration amplitude at all test conditions.

5. CONCLUSIONS

In this work, a finite element model of chip formation process in face turning is combined with a dynamic model of machine tool system in order to obtain a comprehensive model which realistically predicts the effects of various cutting parameters on displacement amplitude of the workpiece in face turning. A Lagrangian FE approach is used in combination with mesh adaptation techniques to deal with severe mesh distortion in machining process simulations. Using the combined model, the displacement amplitude of the workpiece is determined at different cutting conditions. In particular, the effects of depth of cut at constant cutting speed is studied which defines the onset of instability. Also, the effect of cutting speed on the vibration amplitude is investigated. The simulation results show a good agreement with experimental results in the range of cutting speed and feed rate considered. It is shown that the stability borders obtained at mid-to high speeds are in agreement with the results considered in the present work for investigation. Furthermore, 3D FE models are able to predict the tool wear and corresponding load during the cutting process. It is demonstrated that the FE simulations can be used successfully to distinguish between new and worn tools. This is very meaningful for the scientific research and education. A high degree of identified between the experimental and FE simulated results in identifying the effect of vibration amplitude on tool wear state.

6. REFERENCES


INFORMATION

Dept.of Mechanical Engineering, GITAM Institute of Technology, GITAM University, Visakhapatnam, India-530045
Tel: +91-891-2840250

Acknowledgements

The authors would like to acknowledge Dr. R V S Subrahmanyam Scientist ‘G’ & Director, CNC Center NSTL, Visakhapatnam, India for providing both computational and experimental support.

Authors:
M. Prakash Babu, Corresponding author, Ph.D. scholar, Dept.of Mechanical Engineering, GITAM Institute of Technology, GITAM University, Visakhapatnam, India-530045, Tel: +91-0-9573219821
e-mail: m.parakshbabu1535@mail.com

Dr Balla Srinivasa Prasad, Research guide, Assistant Professor, Dept.of Mechanical Engineering, GITAM, Institute of Technology, GITAM University, Visakhapatnam, India-530045, Tel.: +91-0-98483210710 Fax: (+381 21) 454-495
e-mail: bsp.prasad@gmail.com