International Journal of Advance Research in Science and Engineering
Vol. No.4, Issue 11, November 2015
www.ijarse.com
IJARSE
ISSN 2319 - 8354

OPTIMIZATION OF TOOL CHIP CONTACT AREA IN TURNING OPERATION USING FINITE ELEMENT THERMAL ANALYSIS

Kannan Nair M R [1], ShyamSundar V [2]

[1]*Mechanical Department Mar Baselios College of Engineering Nalanchira , Thiruvananthapuram (India*)

[2]*Assistant Professor, Mechanical Department Mohandas College of Engineering Nedumangadu , Trivandrum (India*)

ABSTRACT

Turning is one of the metal cutting operations that are widely used manufacturing technique in the industrial world. In this process, different parameters such as feed, speed and depth of cut will impact the material removal rate, production cost, tool life, cutting process etc. The aim of the present work is to minimize the tool chip contact area temperature in a turning operation. For that general purpose FEA software will be used to set up the finite element model in the turning operation of an EN 31 steel alloy as workpiece and tungsten carbide is selected as the tool material. With the help of Design of Experiment software (DX9) the input parameters are optimized and these optimized variables were used for the finite element analysis. DOE was used to find the factor combination that optimizes the simulation response. For the simulation of the chip morphology Johnson-cook model was also used. It was found that cutting speed is the main variable which affects the chip tool interface temperature and later the thermal and mechanical performances with cutting speed were analyzed.

I. INTRODUCTION

Turning operation is one of the most widely used machining operations in industrial as well as academic world. In this operation there is a rotating work piece. The removal of material from the work piece will be obtained when the tool will be moving parallel to axis of the rotating work piece. The proper selections of machining parameters are required for the performance characteristics like material removal rate, surface roughness, tool life, machining quality. To find out the proper parameters, Experimental as well as modelling techniques are been widely used. Finite Element Analysis are widely used for calculating stress, energy, temperature and other difficult to measure parameters through experiments. Temperature measurement in tool chip interface can't be easily and accurately measured through experiment. For the experimental setup, time consuming and costly thermocouple technique is required. With the help of Finite Element Analysis performance parameters can be easily analysed. With the use of powerful computers commercial software packages, Finite Element Method (FEM) has become one of the most powerful tools for the simulation and analysis of cutting process.

In turning operations, there are different parameters and these parameters can affect the performance characteristics like tool life, cutting temperature, material removal rate, cutting force and surface roughness. In this work three cutting parameters are selected and the selected parameters are feed, speed and depth of cut. In this work simulation of orthogonal turning operation was carried out with using software, ABAQUS. Orthogonal cutting is one of the basic metal cutting using a single point cutting tool and it takes place when the cutting face of tool is 90 degree with line of action of tool. The objective of optimisation is finding the best solution in the decision model. So there is the need of optimising the chip tool interface temperature. In this DX9 software was used for parameter optimisation and later with this parameter FEA was carried out.

II. DESIGN OF EXPERIMENT

Design of experiment is a methodology that defines an optimal set of simulation runs in a design space. DOE is used specially for simulation specified problems whose major goal is to determine which factors have the greatest effect on the response and to achieve that in a minimum simulation runs. In this work here parameters are optimized with DX9 and thus reducing the simulation time for the study of chip tool interface temperature has been studied. This DX9 software can be used to analyze which parameter affects the secondary temperature region in a cutting simulation. In this work the parameter which affects the temperature at the chip tool interface was found and later studied how this parameter affects the mechanical and temperature performance were analyzed. But in this work were able to find the concerned factor level combinations and thus showing the factor combination that optimizes the simulation response. The parameters and their respective levels have been shown in the below table 1.

TABLE 1: Parameters and their corresponding levels

Serial number	Parameter	Symbol	Level 1	Level 2
1	Cutting speed (m/min)	V	39	112
2	Feed (mm/rev)	F	0.6	0.10
3	Depth of cut (mm)	D	0.4	0.6

In this it has got two levels and three parameters so 2^3 simulation runs has to be performedin the normal case. Knowing the number of parameters and the number of levels, the proper orthogonal array can be selected.In this present considered the L4 array. L4 means the array requires 4 runs. 2^3 indicates that the design estimates up to three main effects at 2 levels each.In this work Taguchi method based on orthogonal array which gives much reduced variance for the simulation with optimum settings of control parameters. This allows for the collection of the necessary data to determine which factors most affect the interface temperature with a minimum amount of simulation.

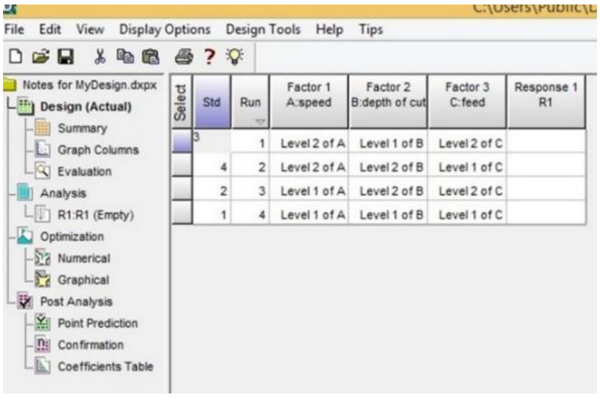


Fig. 1: Optimized Simulation run designed with the help of DX9 software

Here A is considered as parameter cutting speed, B as parameter depth of cut and C is taken as feed. With this pairs of combinations finite element analysis will be carried out. Later with from the FEA results the chip tool interface temperature can be found out. The values that are obtained are added in the response columns and later the significant parameter that affects the response are analyzed.

III. NUMERICAL METHOD

The ABAQUS unified FEA product suites offer powerful and complete solutions for both routine and sophisticated engineering problems covering a vast of the industrial and design applications. In this section, ABAQUS 6.13 was used for simulating metal machining operation. ABAQUS was used in years for analysing the machining approaches and was used to solve this class of numerical model with very good accuracy. ABAQUS includes both mechanical and thermo mechanical simulation. Metal cutting is highly nonlinear and coupled thermo mechanical process where the coupling is introduced through localised heating and temperature rise in work piece, which is caused by plastic flow in work piece by friction along the tool chip interface. In this Johnson cook material model was used to describe the material model.

3.1 Modelling of tool and Workpiece

From past experience and reviews, it is clear that the tool geometry has great importance for machining the work piece. Cutting tool was modelled in Abaqus6.13 software. A simple 3D tool with a rake angle of 10 degree is used. 3D tool was created as a deformable body. The geometric variables and thermal and mechanical properties of tungstencarbide tool are shown respectively in table 1 and table 2.

TABLE 2: Geometic variables of tool

Rake angle α (º)	**Clearance angle c (º)**	**Tool tip radius (mm)**
10	5	0.005

IJARSE
ISSN 2319 - 8354

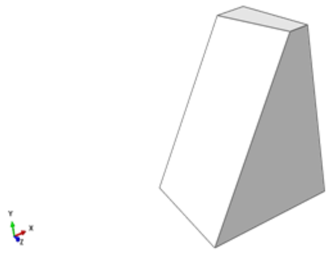


Fig. 2: Tool Model

TABLE3: Mechanical Properties of Tungsten Carbide tool

Density, ρ (kg/m^3)	Young's Modulus, E (Pa)	Poisson's Ratio, ϑ	Thermal Conductivity, K (Wm^{-1} k^{-1})	Specific Heat, C (Jkg^{-1}k^{-1})	Expansion, α (k-^{1})
15500	6.5e11	0.21	50	203	5e-6 at 293K

In the current work EN 31 steel alloy was taken as a work piece material. This type of steel alloy has got lot of application in industrial sectors. This steel is widely used in Ball and Roller Bearing,spinning tools, Beading Rolls, Punches and Dies. In this 3D analysis of orthogonal turning operation was considered. Here work piece was of C.S (4x2 mm^2) with a length of (20 mm) was modelled. Mechanical and thermal properties of work piece is shown in the below table 4

In the work piece model the model is partitioned to two different zone that is deformable zone and non deformablezone. Partition is a necessary step in machining stimulation because this deformation region is required for the formation of chip during machining. As from the DX9 software for different runs are needed to be analysed. So work is of required depth of cut has to be modelled. That work piece of 0.6 mm depth of cut and work piece of 0.4 mm depth of cut has been modelled. The partitioning of both work piece had been carried out.

TABLE 4: Mechanical and Thermal Properties of EN 31 Steel Alloy

Density, ρ (kg/m^3)	Young's Modulus, E (Pa)	Poisson's Ratio, ϑ	Thermal Conductivity, K (Wm^{-1} k^{-1})	Specific Heat, C (Jkg^{-1}k^{-1})	Expansion, α (k-^{1})
78100	215000	0.285	46.6	420	11e-6 at 293K

Section view of the work piece material is shown in the figure below.

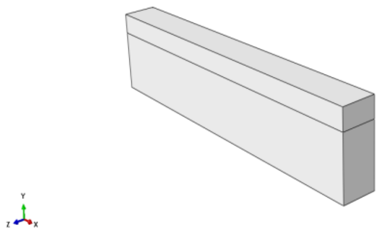


Fig. 3: Workpiece Model

3.2 Material Modelling

In finite element analysis flow stress models are necessary while the analysis consists of high strain rate temperature etc. So this flow stress models are required to maintain work material behaviours at high strain rate deformation condition. The material is modelled as elastic plastic and its flow stress is taken as a function of stress strain and temperature as to represent real behaviour in cutting process. The constitutive model proposed by Johnson Cook is been widely used to describe the flow of stress at high temperature and deformation condition. Johnson Cook has been chosen to implement the mechanical behaviour of the work piece material. This Johnson Cook model was used to describe material behaviour in a plastic region over a large strain at high strain rate and temperature. Equation for Johnson Cook model is shown below equation (1)

$$\bar{\sigma} = [A + B(\bar{\varepsilon})^n]\left[1 + C\ln\left(\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}\right)\right]\left[1 - \left(\frac{T-T_0}{T_{melt}-T_0}\right)\right]^m \quad (1)$$

The constants for Johnson Cook constitutive model for the work material EN 31 steel alloy is given in the table 5. The mechanical and thermal material properties were applied to both the tool and work piece models.

TABLE5: Johnson Cook Parameters for EN 31 Steel Alloy

A(M Pa)	B(M Pa)	N	C	M	**$\gamma 0(s^{-1})$**	T_{melt} (k)	T_0 (k)
553.1	600.8	0.254	0.013	1	0.001	1733	300

3.3 Initial Damage Criterion

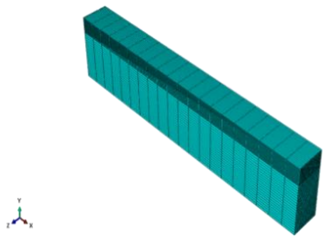
For the understanding of chip formulation this Johnson Cook damage model also places a significant role. So this Johnson Cook damage model is used in conjunction with Johnson Cook flow stress model. Johnson Cook damage model is based on values of equivalent plastic strain at element integration point. The Johnson Cook damage model is used when there is high strain rate deformation of the material which occurs mainly in machining operation. The damage constants for EN31 steel alloy is shown in the table below. The critical plastic strain rate value used to govern damage initial criteria is 1.5.

TABLE 6: Johnson Cook Damage Parameters for EN 31 Steel alloy

d1	d2	d3	d4	d5
0.06	3.31	-1.96	0.018	0.58

3.4 Meshing of the Models

A continuous region is divided into discrete region called element in finite element analysis. This procedure is called discretization or meshing. After applying material properties to the models proper meshing is done. The meshing of both the tool and work piece has been performed separately. The work piece consists of 4400 number of hexahedral elements with 9246 total number of nodes. The element is of type C3D8RT with linear hexahedral elements. The mesh density at the cutting plane is made very fine in work piece model for to increase the accuracy of the result in this zone. The upper part of the material which comes in contact with the tool and at this region chip formation may occur. The region which was partitioned earlier and in these region element deletion criteria was taken from the removal of material from the work piece.When the equivalent plastic strain at integration point reaches a defined failure strain, material failure takes place. So the element is removed from the mesh at the respective integration points where the material failure occurs. In this C3D8RT element type where taken, and with the help of this element it will be able to get nodal temperature at the result.



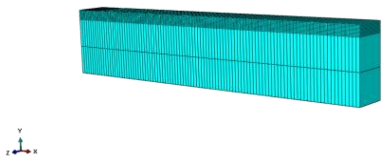


Fig.4: Meshed Work Piece Model of Depth 0.6 mm and 0.4 mm

Tool which is a deformable body has got 550 total numbers of elements with 792 numbers of nodes. A plane strain coupled thermo mechanical analysis was carried out.

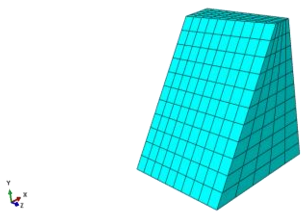


Fig.5: Meshed Tool Model

3.5 Friction Model

As both the tool and work piece comes in contact, there occurs the importance of friction. To model the effect of contact friction, in metal cutting simulations, often Coulomb's friction law is used with a constant coefficient of friction. In this work constant coulomb friction is adopted. The friction model is defined as

$\tau = \mu\ \sigma_n$. (2)

Where, τ is the frictional stress, σ_n is the normal stress and μ is the coefficient of friction.In this coulombs friction coefficient was taken as 0.5 due to friction heat will be generated and this generated heat will be evenly distributed between the interface surfaces. Later general contact was defined between the tool and the work piece.In the simulation model surface-to-surface contact conditions is used to define the tool-chip as well as tool-work piece contact pair.

3.6 Assembling and Boundary Conditions

In this machining simulation after completing the modelling of parts it is necessary to assemble them together. Then only the cutting can be performed. So here it is necessary to position the tool relative to the work piece. It is necessary step to identify the location of thecutting tool where the contact will first occur between the parts, in this case here it is the work piece.The cutting tool has been positioned in such a way that the cutting edge or the tip should come in contact with the workpiece thus the tool can easily pierce through the work material. Cutting tool had been placed as it comes at the deformation zone of the work material. The boundary condition of assembled models is specified .In this simulation the lower part or half of the work piece undergoes less or negligible deformation, so at that region zero displacement is required. Initially the movement of the work piece and tool was arrested in all direction and during machining step tool moves along the X direction with velocity 39m/min and 112m/min. Coupled temperature analysis was conducted by giving an initial temperature of 30^0C.

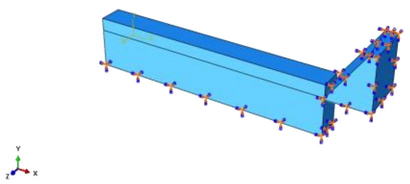


Fig 6: Initial Boundary Condition

IV. RESULTS AND DISCUSSION

4.1 Mesh Convergence Study

In finite element analysis just because a solution was obtained this doesn't mean that solution may be correct. In order to select a suitable mesh for further study, optimal distribution of elements were to be found out. To analyse the results there comes the importance of mesh convergence study. A mesh study was conducted using three different mesh densities namely coarse, medium and fine. In the table 8.1, fine mesh was taken as reference mesh. The difference from the reference mesh is shown in the difference column in percentage. The difference can be calculated by using below formula

$$\textbf{Difference (\%)} = \frac{\text{Reference mesh} - \text{'desired mesh'}}{\text{Reference mesh}}\ X\ 100 \qquad (3)$$

From the mesh comparison it is clear that fine mesh shows high temperature at the chip tool interface than the medium and coarse mesh. The results obtained from the mesh study is shown with the help of graph as shown below. So fine mesh wasapplied to all other models for obtaining accurate results.

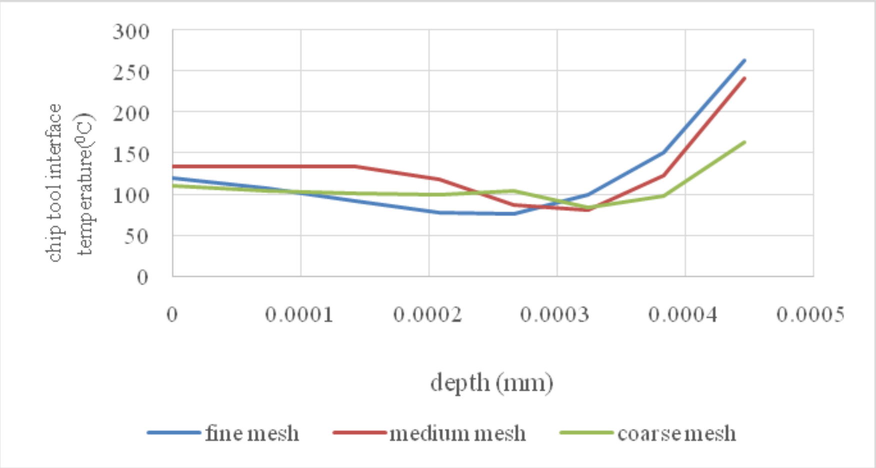


Fig. 7: Mesh Comparison

4.2 Numerical Simulation Results

Chip formation model was created using FEA software ABAQUS 6.13. During machining simulation the chip tool interface temperature, effective stress and effective strain were noted for both depth of cuts. Chip formation was obtained for all the simulation runs. The chip tool interface temperature obtained from FEA analysis has been entered into DX9 software where the response has to be entered. From that the parameters which affects thetemperature at that region can be easily analyzed through the simulation runs.From the below tabular column it is clear that irrespective of the feed and depth of cut the higher the temperature is obtained in the run where cutting speed has got higher value. So to minimize the tool wear and to avoid corrosion and burning at the tool surface and also for obtaining better surface finish low range of cutting speed of range 27-40 m/min has to be applied.

TABLE 7: Results obtained from Finite Element Analysis

Serial no:	Cutting speed (m/min)	Depth of cut (mm)	Feed (mm/rev)	Response (chip tool interface temperature) (^{0}C)
1	112	0.4	0.10	263.5
2	112	0.6	0.06	304.6
3	39	0.6	0.10	246.4
4	39	0.4	0.06	239.05

4.3 Effect of Mechanical and Thermal Performance Measure with Varying Cutting Speeds

As mentioned earlier cutting speed is the main factor which affects the temperature zones during machining. For studying the effect of effective stress or Von Mises stress, effective strain and temperature at the chip tool interface different range of cutting speed was selected for 0.6mm and 0.4mm depth of cut for the work piece material with constant feed. Cutting speed values selected for the carrying out simulations are 100m/min, 125m/min, 150m/min, 175m/min and 200m/min. Corresponding simulations were carried out and the following results have been taken to consideration.

4.3.1 Variation of temperature at chip tool interface with varying cutting speeds

When the cutting speed increases the temperature at the chip tool interface increases as it is clear from the graph. The maximum temperature at the chip and tool interface region was noticed at a cutting speed of 200m/min for both depth of cut and lower temperature was noticed at lower cutting speed. So it is clear that as the cutting speed increases, the generated temperature on the chip also increases. From the graph it is also clear that as the depth of cut increases there will be an increase in chip tool interface temperature.

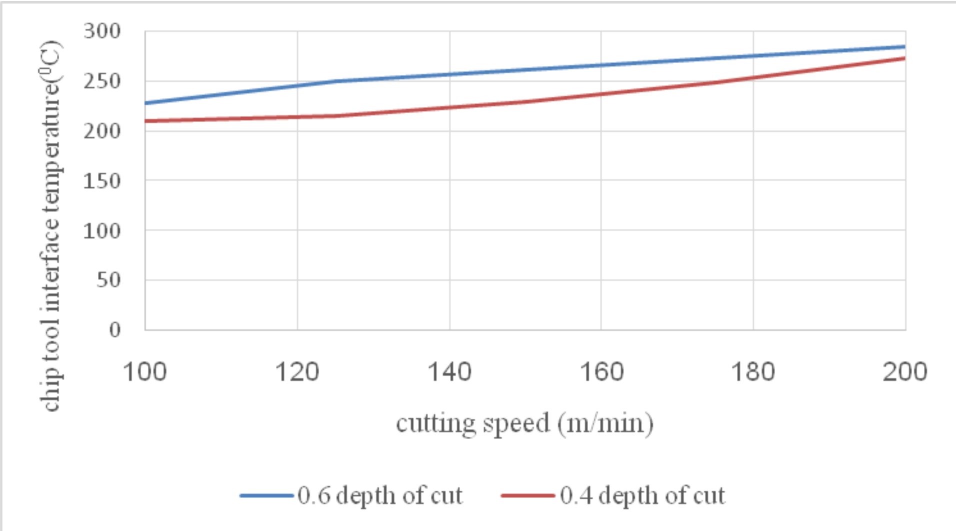


Fig.8: Chip tool Interface Temperature Variation vs Cutting Speed

4.3.2 Variation of Von Mises stress with varying cutting speeds

As the cutting speed is increased from 100 m/min to 200m/min there is a drastic decrease in effective stress at the chip tool interface zone. The higher value of effective stress was obtained at lower cutting speed.

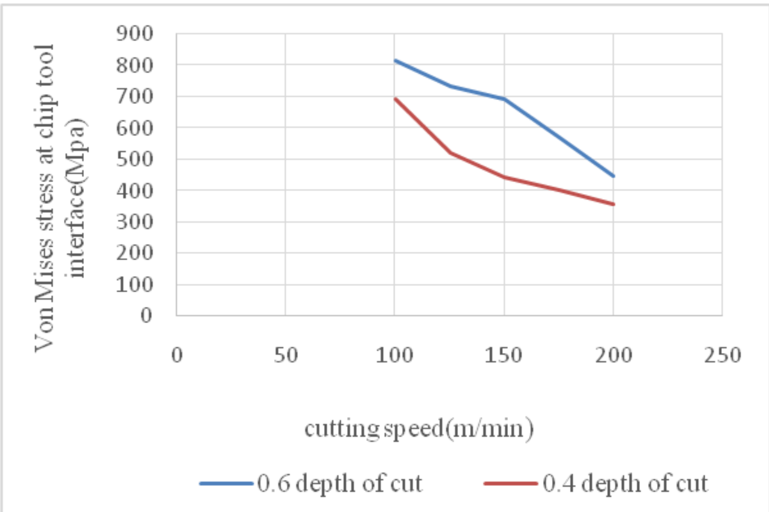


Fig.9: Von Mises Stress Versus Cutting Speed

4.3.3 Effective strain distribution with varying cutting speeds

As the value of cutting speed increases the strain at the chip tool interface is found to increase. In the secondary region there exists a plastic strain region due to the contact with the tool and chip formed. So as the cutting speed is increased the strain values will also increase accordingly.

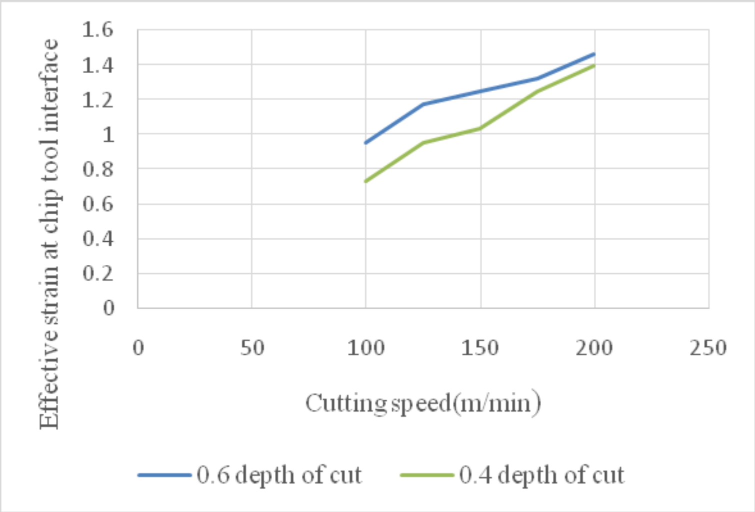


Fig.10: Effective Strain Versus Cutting Speed

V.CONCLUSION

In this work, with the help of FEA the machining of steel alloy was performed. This work is concerned on the secondary region of heat generation during a metal cutting process. Here first the parameters were arranged in such a manner that they give maximum and agreeable output with minimum simulation runs thus saving cost and time. Later with this parameters simulation was carried out with the help of FEA software ABAQUS 6.13.With the help of DOE software parameter combinations were obtained with these combination simulation was performed. By varying depth of cut, cutting speed chip tool interface temperature was found and later it was analysed and interpreted that cutting speed is the main influencing factor which affects the temperature as well as mechanical performance variables like stress and strain. So by applying proper cutting speed the minimization of chip tool interface temperature can be bought down to great extent. The objective of this work focuses on the variables that affect the temperature at the chip tool interface zone and by using suitable cutting parameter values the temperature at this region can be bought to minimum.

REFERENCES

[1] "Machining Process Simulation" Claudia H. Nascimento& Alessandro R. Rodrigues &Reginaldo T. Coelho Univestity of Sao Paulo, Brazil.

[2] "Optimization of Machining Parameters for Turning using Taguchi Approach"AnandS.Shivade, ShivrajBhagat, SurajJagdale, Amit Nikam, Pramodlondhe March 2014

[3] "Chip-Tool Interface Temperature Prediction Model for Turning Process"L.B.Abhang and M. Hameedullah International Journal of Computer Communication and Information System (IJCCIS) Vol2. No1. ISSN: 0976–1349 July – Dec 2010

[4] "Optimization of cutting parameters based on tool chip interface temperature in turning process using Taguchi's method"MilenkoSekulić, Marin Gostimirović, PavelKovač, BorislavSavković, TMT 2011, Prague, Czech Republic, 12-18 September 2011

[5] "Finite Element Modeling of chip formation process with abaqus/explicit TM 6.3"P.J.Arrazola, D.Ugarte, J.Montoya, A.Villar, S. Marya VIII International Conference on Computational Plasticity, Barcelona, (2005)

[6] "Determining the effect of interface friction on tool-chip contact length in orthogonal cutting using the finite element method" S. A. Iqbal, P. T. Mativenga, M. A. Sheikh,The University of Manchester, Manchester, M60 1QD, UK

[7] "Prediction of tool and chip temperature in continuous and interrupted machining" Ismail Lazoglu, Yusuf Altintas International Journal of Machine Tools & Manufacture 42 March (2002) 1011–1022

[8] "Finite Element Simulation and Experiment of Chip Formation Process during High Speed Machining of AISI 1045 Hardened Steel"C.Z.Duan , T.Dou, Y.J.Cai , Y.Y.Li International Journal of Recent Trends in Engineering, Vol 1, No. 5, May 2009

[9] "3D FEM Analysis of Cutting Processes presents a modelling and simulation analysis with FEM" Corina Constantin ,SorinMihaiCroitoru

[10] "Modelling and simulation of metal cutting by finite element method"Cenk KILIÇASLAN, et.al 2009

[11] Wikipedia

[12] "Modeling and Simulation of Turning Operation"M.KumaraSwamy IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) ISSN: 2278-1684 Volume 3, Issue 6 (Nov-Dec. 2012).

[13] Johnson, W.K., Cook, R., 1983, "A Constitutive Model and Data for Metals Subjected to Large Strains, High Strain Rates and High Temperatures". Proc. 7th Inter. Symp. on Ballistics, pp.541-547.