Title
Finite Element Modeling of Drilling Using DEFORM

Authors
Gardner, Joel D.
Dornfeld, David

Publication Date
2006-06-01

FINITE ELEMENT MODELING OF DRILLING USING DEFORM

J. Gardner, D. Dornfeld
gard0158@me.berkeley.edu

Sponsored by
NSF Grant DMI-0300549 – "GOALI: Development of Comprehensive Drilling Simulation Tool"

ABSTRACT

DEFORM-3D is a robust simulation tool that uses the finite element method (FEM) to model complex machining processes in three dimensions. One of the most recent processes that has been modeled in DEFORM is drilling. The program has many features that can be daunting and difficult to adjust for a new user. These features combine machining and the FE model to simulate drilling operations. Although this program can generate useful results, it can be very difficult to obtain these results consistently. This is because the code is often subject to crashing when the simulation parameters are not set properly. By recognizing the common errors and idiosyncrasies of DEFORM, a user can generate useful simulations quickly and efficiently. The background, application, troubleshooting, and analysis of DEFORM-3D are presented to showcase its capabilities while discussing its limitations in drilling.

Keywords: Drilling, FEM, DEFORM, Simulation

INTRODUCTION

The DEFORM family of programs are products of Scientific Forming Technologies Corporation (SFTC) that model both metal forming and machining operations using the finite element method. The objective of the first DEFORM program was to model forging operations. Several similar codes have since been developed to model many forming operations as well as machining operations in both two and three dimensions. Drilling is one of the newest and most complex applications that DEFORM has been applied to. This is because drilling is a high speed, three-dimensional operation with complex tool and chip geometries. The mathematical theory and modeling that DEFORM applies is a result of years of academic and industrial development. Although it has achieved reliable results in many applications, there are many areas for improvement in the application of the program, especially in drilling.

Through the proper use of DEFORM, a large amount of information can be obtained that is not easily iterated through experimental work. Closed form analytical solutions are generally not available to model chip formation and burr formation in machining processes where large material deformation is occurring. FEM simulation is a reasonable solution to these problems.

By understanding how to use DEFORM efficiently, one can learn to optimize process parameters, improve tool geometry, optimize tool paths, or minimize burr formation with the minimum amount of simulation time. Although DEFORM is continually being developed and optimized, the current generation gives a substantial amount of information for simulators to analyze.

THEORY OF DEFORM

The first finite element modeling of machining was published in 1973 by Klamecki (1973). The computational resources limited the mesh size and the complexity while requiring substantial simulation time. In addition, the theory was rather rudimentary. Improvements on the theory and application have been published by Strenkowski and Carroll (1985) and a comprehensive review of general FEM code as applied towards machining has been published by Marusich and Ortiz (1995). These papers detail improvements in the mathematical theory and how to apply them toward machining. DEFORM applies this theory in a user-friendly graphical user interface (GUI) that is very robust compared to many custom FEM codes.

Klocke et al. (2001) presented an overview of a two dimensional FEM code used by DEFORM to study orthogonal cutting. They detail many of the topics discussed in this paper but applied in two dimensions. His analysis is also applied toward orthogonal cutting as opposed to three dimensional drilling. Klocke et al. detail much of the basic FEM theory and machining theory embedded in DEFORM's code.

Mesh Types

DEFORM has the ability to use several types of meshes for three dimensional FEM. Historically, the two standard FEM meshes are Eulerian and Lagrangian. There are also combinations such as the Arbitrary Lagrangian Eulerian (ALE) and the Coupled Eulerian Lagrangian (CEL) meshes. Despite the numerous options available, the Lagrangian mesh has been used for all the simulations that have been done for drilling in DEFORM. The iterative solvers that are packaged with DEFORM have been optimized for the Lagrangian mesh and, in turn, simulate much faster than other meshes. Although the Lagrangian mesh is not as comprehensive as the Eulerian mesh, it has much better simulation cycle times as a result.

Liu and Liu present a good overview of the different mesh types in the introduction their book (Liu, 2003). One key advantage of the Lagrangian mesh in simulating drilling processes is the ability to know the entire time history of the key variables at every point during the simulation. That means, if a simulation crashes for any reason, a new simulation can start where the crashed simulation stopped. This is particularly useful because nearly every simulation has some sort of problem during the run. This is possible because the Lagrangian mesh is reformulated at nearly every time step, in order to manage the deformation of the material.

One of the biggest strengths of DEFORM is its ability to mesh complex geometries. Significant deformation occurs in machining simulations and this has been historically problematic for the Lagrangian mesh. However, if the geometry is remeshed after each time step, the Lagrangian mesh is a reasonable choice to show burr formation. DEFORM is a leader in creating adaptive meshes and remeshing complex geometry and this makes it a desirable code for drilling analysis.

Failure Criterion

During the drilling simulation, the cutting edges of the drill bit are shearing the workpiece material at high speeds which separate the material from the workpiece by chip formation. The material separation criterion for machining has been a topic of interest in the development of the theory of finite element modeling of machining. Initially, a parting line model was assumed to simplify the simulation process. This model assumed a small crack existed in the material and the chip was separated from the workpiece in a predetermined "unzipping" fashion. Eventually, the maximum plastic strain model was proposed and this criterion has been adopted by most FEM models. This maximum plastic strain model assumes that material separation occurs when an element reaches a critical plastic strain for the material model of the workpiece. The element is then split into two elements and a chip is formed. Huang and Black (1996) determined that under smooth separation conditions, the chip separation criteria does not greatly affect chip geometry nor stress and strain distributions. One can argue whether drilling actually produces smooth separation. Regardless, the maximum plastic strain criterion has been implemented and this has been the most accepted method of failure criteria to model burr formation in drilling (Guo, 2000).

DRILLING IN DEFORM

Several different types of machining operations can be accomplished with DEFORM-3D including drilling, turning, milling, and broaching. If a tool geometry can be modeled, the machining operation can be simulated. One of the most difficult problems faced with modeling drilling operations is obtaining an accurate model of a drill bit. Guo (2000) and Vijayaraghavan (2005) both present how this can be done and the latter developed a program to do this quickly and easily. A picture of what a modeled and meshed drill bit, as modeled by Vijayaraghavan and meshed by DEFORM, is shown in Figure 1.

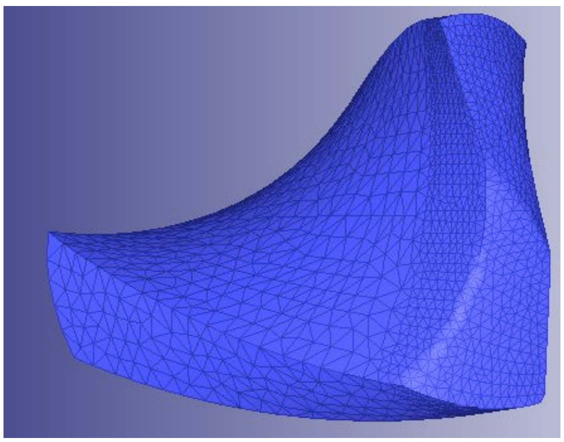

FIGURE 1: ACCURATELY MODELED DRILL FROM PROGRAM WRITTEN BY VIJAYARAGHAVAN (2005).

The key features that need to be modeled on the drill are the diameter, the chisel edge angle, the point angle, the helix angle, the relief angle, the web thickness, and the margin lip. Modeling the chisel edge angle and the relief angle proved to be the most difficult to model and are very important features to model drilling simulations properly.

Although the workpiece geometry is less critical than the tool geometry, the workpiece model affects the simulation time significantly. Generally, simulations are performed on a spotted or center-drilled workpiece. The spot is roughly at the same angle than that of the chisel edge of the drill. Generally, most holes that are drilled are spotted to allow better circularity of the hole. The spot also improves simulation time significantly because this material doesn't need to be removed by the drill bit. The cutting edge of the drill can start chip formation immediately in the simulation rather than having to wait for the chisel edge to remove material.

It is also very efficient to model the workpiece as circular in addition to spotting it. A circular workpiece meshes much more easily than a square or irregularly shaped workpiece does. This is partly due to the rotational nature of drilling operations. Most of the stress, strain, and material flow happen in an axi-symmetrical manner. Furthermore, the workpiece is generally modeled to be somewhat thin. The thickness should be at least the radius of the drill. This improves simulation time significantly because one revolution of a drill can take 200 time steps. At feed rate on the order of 100 microns per revolution, the simulation can take a very long time. A more detailed discussion of simulation time is presented later. A drawing of a typical workpiece for a 118^0 point angle drill is shown in Figure 2.

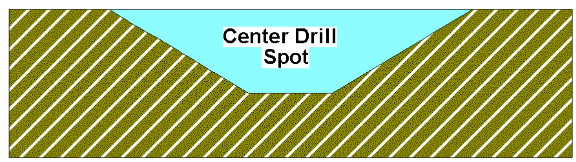

FIGURE 2: CROSS SECTION OF WORKPIECE GEOMETRY TO BE DRILLED BY 118^0 DRILL BIT.

<u>Tool and Workpiece Meshing</u>

The meshing of the tool and the workpiece are very critical in modeling the drilling process accurately. A finer mesh generally gives finer accuracy, but the simulation time increases exponentially as the number of elements increases linearly. The resolution of the mesh has a lower limit that isn't a limitation of the commercial software or the computational power. Rather, the mesh resolution is limited by the underlying mathematics of the FEM.

The drill bit mesh is easily compared to the workpiece mesh because the drill only needs to be meshed once. The drill is modeled to be a rigid object that does not deform during the drilling process. Consequently, tool wear and thermal damage to the drill are not considered. It is possible for these to be simulated, but at the expense of increased computation time. The models considered in all simulations were of perfect tools.

To mesh the drill bit, only one turn of the helix is used in the drill model because this is all that contacts the workpiece. Tetrahedral elements are used that have four nodes. The elements' sizes are described by their edge length. A finer mesh is used at the tip of the drill in a 4:1 ratio. That is, the edge length of one element at the drill tip is 25% of the length of the coarser mesh at the helix of the drill. A higher ratio can be used but isn't necessary. In simulations that have been performed, 20,000 elements are sufficient to give an accurate model of the drill bit.

Meshing the workpiece is much more complex and problematic than meshing the drill bit. The workpiece is modeled as a plastic object which means it can be deformed and cut by the drill bit. Consequently, when the mesh is deformed it must be regenerated frequently, often at every time step. As previously mentioned, this frequent mesh regeneration is one of the requirements of a Lagrangian mesh. The workpiece mesh must also be finer than the drill mesh because the chip geometry can sometimes only be described with very fine elements. The stress, strain, and temperature of the elements all have very high gradients across the workpiece as well. These properties generally vary linearly or, at most, quadratically, from node to node, across an element. To approximate these high gradients accurately, a very fine mesh is required.

The mesh is weighted to generate more elements where there are large strains, large temperatures, large deformations, and large strain rates. Areas can also be specified geometrically where to generate finer elements. This option allows the mesh to *adapt* to the workpiece to optimize the simulation time and element allocation. In the simulations that have been performed, the adaptive remeshing is generally weighted toward high strains (~50%), high strain rates (~30%) and the high density mesh window (~20%). The mesh window is placed where the drill bit removes material. An example of an undeformed meshed workpiece is shown in Figure 3, showing high mesh density where major deformation will take place. A deformed workpiece is shown in Figure 4.

The minimum element size that is specified for the workpiece should be, at most, one half of the feed rate of the drill bit. The rigid drill cannot cut *through* an element, it can only separate elements by the nodes. Having a fine mesh allows the mesh to deform according to the failure criterion when the drill advances. The contact and separation relationships between the drill and the workpiece are very complex and subject to many errors.

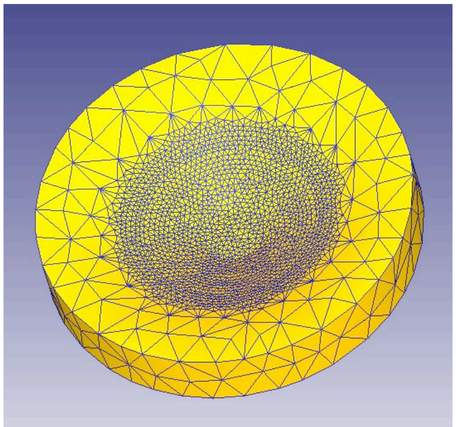

FIGURE 3: TYPICAL MESH FOR AN UNDEFORMED WORKPIECE GENERATED BY DEFORM – NOTE HIGH DENSITY IN CENTER.

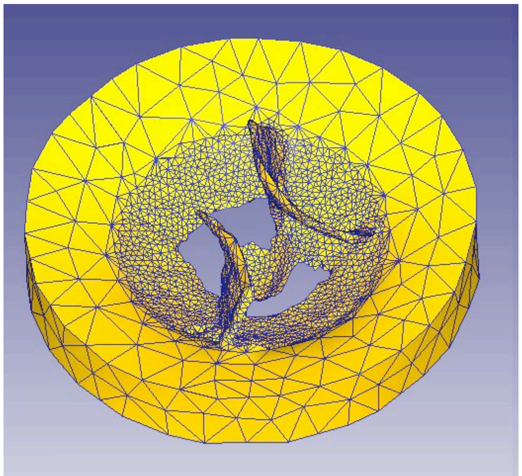

FIGURE 4: TYPICAL MESH FOR A DEFORMED WORKPIECE GENERATED BY DEFORM.

The number of elements in the workpiece should not exceed 50,000 elements for an efficient simulation. The normal number of elements is around 10,000 to 20,000. This runs at a steady rate, but there are several other factors that control computation time. As a rule of thumb, using less than 20,000 elements works well depending on the patience of the simulator. Future work to determine the optimal number of elements and the effects on simulation time and accuracy would be desirable.

Material Properties

In order to use the equations of the finite element method, extensive material data is needed for the workpiece. The model of the workpiece is subject to severe deformation at very high strain rates and temperatures. In these conditions, there are many variables that control the deformation. DEFORM-3D is pre-packaged with a database that includes several metals and alloys. In addition, if the material data are known for a new material, these can be added to the database. The information that is needed to model the material is very difficult to obtain experimentally and many of the material models used are the result of comprehensive testing and modeling. The key relationships to define a new model are the

- Stress vs. strain vs. strain rate vs. temperature model
- Elastic modulus vs. temperature model
- Poisson's ratio
- Thermal expansion, conductivity, heat capacity vs. temperature.

Other information can be added to investigate other phenomenon. For example, to study a TiN coated tool on a titanium workpiece a diffusion coefficient would be needed. Thermal emissivity, hardness, and grain size can also be added to the material model. The material database that DEFORM-3D v5.1 is equipped with includes (but is not limited to):

- Aluminum alloys
- Tool and die materials (e.g. HSS, WC)
- Stainless steels
- General iron-carbon steels
- Titanium alloys
- Brass

The database is reasonably equipped with common engineering materials. This is another strength that DEFORM possesses relative to competing commercial codes like ABAQUS or AdvantEdge (Gardner, 2005).

Boundary Conditions

The boundary conditions for drilling are straightforward. The edges of the workpiece are fixed in all directions. The drill bit will have a rotational velocity and a feed rate (process parameters). Other parameters can be specified such as downward force and spring loaded die force, but these are usually unnecessary. The heat exchange with the environment is defined. This heat exchange is usually small because the drilling process happens very quickly (<1second).

Contact Conditions

The contact conditions control the friction, heat transfer, and master-slave relationships between the tool and the workpiece. The tool is set to be the master object and the workpiece is the slave object, meaning that the workpiece will deform according to the tool movement. That is, if the workpiece mesh tries to move into the tool, this is not allowable. An additional master-slave relationship is set up between the workpiece and itself. This is set so that chips that are generated cannot flow back into the material.

According to a developer at DEFORM, friction modeling is still a matter of discussion amongst researchers, but in their experience a value of friction between 0.5 and 0.6 has given reasonable results. This is in the range of sticking friction which is 0.577.

The interface heat transfer coefficient for convection is set to be between 20 and 40 W/m^2K. This models flooded coolant lubrication for the drilling operation. Smaller values on the order of 5 W/m^2K can be used for dry drilling.

The friction and heat transfer coefficient models for most of the simulations that have been run have used these values. Alternative models can be used to model tool wear and minimum quantity lubrication (MQL) machining.

Simulation Parameters

DEFORM has several simulation parameters that can be changed to achieve different objectives. Although there are several options, the same parameters are used for most simulations. The number and size of time steps can be changed. However, the number of time steps needs to be only arbitrarily large enough so the tool goes through the workpiece. Although the time step can be specified, it is usually determined by the mesh size. Smaller element sizes require smaller time steps. Remeshing criteria and alternative stopping conditions can also be applied, but the default values are usually sufficient.

Database Generation

Once the tool and workpiece models are imported and meshed, the boundary/movement conditions and the materials are specified, and the contact conditions are set, the first step of the simulation database can be generated. DEFORM checks to make sure that all the key parameters were input and then generates a .DB file to be run in the simulator.

COMMON ERRORS

The next logical step is to run the simulation. This can take several days and even weeks depending on the parameters and hardware resources. During this time, several errors can occur that will prohibit the simulation from proceeding. Since DEFORM uses a Lagrangian mesh that saves information at every time step, crashes are *not* catastrophic. Most of the errors that are encountered can be dealt with in the pre-processor by changing some of the parameters previously discussed.

Inconsistent step number

This can be a common error for beginners. If the database is not generated in the pre-processor directly before the simulation is started, this error will occur. To solve this, the database must be opened in the pre-processor, the database needs to be checked and generated, then the simulation should be started again.

Identical remesh step

This is one of the most common and difficult errors to deal with. There are a few reasons this error is given. In order to manage the solution effectively, the message file must be examined.

Occasionally, the message file will indicate that a negative value was extrapolated from the material data for a certain variable. This means that the material properties file has a negative slope at a certain point that extends across the abscissa at high temperatures. The processes simulated frequently encounter very high temperatures that are beyond the range of the material data. In order to solve this, the material data must be altered to set property values at high temperatures to be some positive value. The slope can also be set to be zero at high temperatures. A representation of SS-304, where this error occurred, is shown in Figure 5.

Another possible cause of the identical remesh step error is that the meshing algorithm cannot generate a new mesh around the deformed workpiece. As mentioned previously, when the drill advances, it can separate elements, but it cannot split them. Occasionally, the resolution of the mesh is not fine enough to be separated by the drill and the drill mesh interferes with the workpiece mesh. This causes the remesh to be severely deformed, which, in turn, leads to negative Jacobian matrices for some of the elements. The mapping functions cannot be operated if there is a negative Jacobian matrix. Consequently, the FEM cannot work. To solve this problem, the file should be opened in the pre-processor and the mesh should be refined with a smaller minimum element size. This doesn't always solve the problem, but is the best solution for this problem.

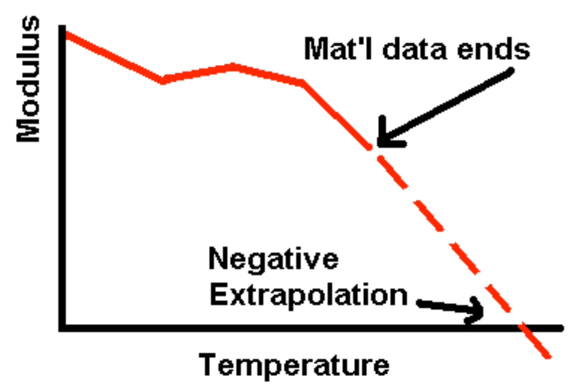

FIGURE 5: PLAUSIBLE CAUSE OF A IDENTICAL REMESH STEP: MATERIAL PROPERTY DATA IS EXTRAPOLATED TO A NEGATIVE VALUE.

The final common cause of the identical remesh step error is when the drill mesh breaks through the workpiece mesh. Breakthrough occurs when the workpiece is too thin to support the force of the drill and separation of the nodes occurs in a non-chip forming process. When this error occurs, the drill advances and sneaks through the nodes of the workpiece to contact the backside of the workpiece. The code is not pre-programmed to recognize when this happens. When it does happen, a special series of steps should be performed.

Under the advanced options tab in the pre-processor, a Boolean operation should be performed on the workpiece with the drill as the master object. The boundary conditions and contact conditions should be re-initialized and the simulation can be resumed. This usually happens once during the simulation right before breakthrough is about to occur.

Stagnant Mesh

Occasionally, a simulation will run indefinitely without advancing any time steps. This happens when the remeshing algorithm process, DEF_AMG.exe, gets caught in an infinite loop while creating a new mesh. The simulation should be manually aborted, the mesh parameters should be slightly changes, and a new mesh created.

ANALYSIS

The ultimate goal of running drilling simulations is to generate usable and practical information that can be analyzed. The post-processor in DEFORM gives a plethora of information of which the more usable are material flow characteristics, workpiece temperature, stress, strain, and strain rate. This information is available at every node in the workpiece. Cross sections can be generated and all data can be exported for further analysis. Force and torque values of the drill bit are also available to determine what types of loads are encountered during a machining operation.

Like many FEM post-processors, DEFORM generates short films to show the movement that occurred during the simulation. These films show how the temperature and stress change while showing how the material flows. This paper is limited in its ability to show films so a sample screenshot showing the temperature and deformation of the workpiece is shown in Figure 6.

The results from DEFORM are subject to occasional inaccuracies and continuous improvements are being made in the theory and application of the FEM to minimize this. Due to these occasional inaccuracies, the results need to be interpreted rather than simply taken as fact. The inaccuracy occurs in the material flow patterns more frequently than in the temperature and stress evaluation. This is most likely due to the failure criteria used during the simulation as previously discussed. Since machining is not completely understood, the models are subject to unpredictable error. However, by comparing experimental results to FEM results and recognizing logical patterns, these inaccuracies can be corrected.

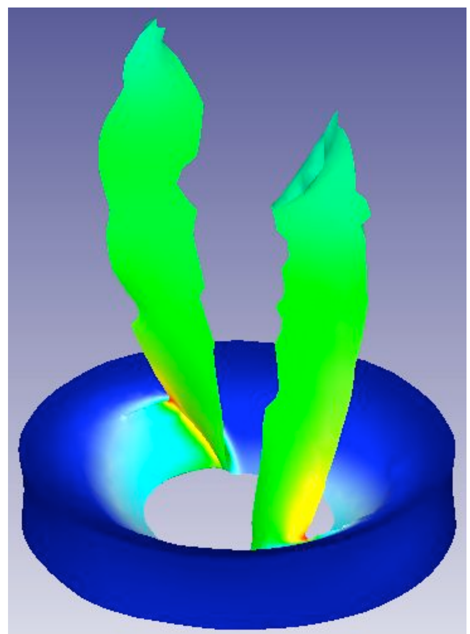

FIGURE 6: SCREENSHOT FROM THE POST-PROCESSOR OF A DEFORM DRILLING SIMULATION SHOWING TEMPERATURE.

The DEFORM drilling simulations, when analyzed, will show many important features. By simulating these drilling operations, many different tool geometries can be investigated without actually making the drill bits. Different workpieces can be investigated as well. By knowing where the highest stresses and highest temperatures are occurring and how the material flows in the chip, the tool geometry and process parameters can be modified to optimize the entire operation and to minimize capital expense for manufacturing.

SUMMARY

DEFORM-3D is a very robust computational tool to simulate drilling processes. The process of using DEFORM takes CAD models of a drill bit and a workpiece, creates a mesh, generates boundary conditions for both, sets contact conditions, and finally simulates the drilling operation. The underlying theory of the code cannot be modified, but many other variables can be changed. Several guidelines have been discussed and troubleshooting advice has been offered for some errors that are commonly encountered. Once a simulation is complete, the stress, strain, temperature, etc. are available for every element at every time step. This information can be used for a multitude of objectives. Although DEFORM is under continual development, it has many features that aid in the selection of optimal process parameters and tool geometries.

ACKNOWLEDGMENTS

The authors thank Chris Fischer of SFTC for providing the LMA with software and assistance. The authors also thank the members of the Consortium on Deburring and Edge Finishing (CODEF) for their discussions, feedback and financial support. Athulan Vijayaraghavan is also appreciated for his provision of CAD models of drills using his software module. The National Science Foundation is gratefully acknowledged for funding this research under grant DMI-0300549 – "GOALI: Development of Comprehensive Drilling Simulation Tool".

REFERENCES

Gardner, J., Vijayaraghavan, A. (2005), "Comparative Study of Finite Element Simulation Software", *2004/05 LMA Reports.*

Guo, Y.B., Dornfeld, D.A, (Nov 2000), "Finite Element Modeling of Burr Formation in Drilling 304 Stainless Steel", *ASME Transactions, Journal of Manufacturing Science and Engineering*, Vol 122, pp. 612-619.

Huang, J., Black, J. (1996), "Evaluation of Chip Separation Criteria for the FEM Simulation of Machining", *ASME Transactions, Journal of Manufacturing Science and Engineering*, Vol 118, pp. 545-554.

Klamecki, B (1973), "Incipient Chip Formation in Metal Cutting – A Three Dimensional Element Analysis", Ph.D. Thesis, *University of Illinois, Urbana-Champaign.*

Klocke, F., Raedt, H., Hoppe, S. (2001), "2D-FEM Simulation of the Orthogonal High Speed Cutting Process", *ASME Transactions, Machining Science and Technology*, Vol. 5, No. 3, pp. 323-340.

Liu, G., Liu, M. (2003), <u>Smoothed Particle Dynamics</u>, *World Scientifi.c.*

Marusich, T., Ortiz, M. (1995), "Finite Element Study of Chip Formation in High-Speed Machining", *International Journal for Numerical Methods in Engineering*, Vol. 38, pp. 3675-3694.

Strenkowski, J., Carroll, J. (1985), "A Finite Element Model of Machining", *Journal of Engineering of Industry*, Vol 107, pp. 349-354.

Vijayaraghavan, A. (2005), "Drilling of Fiber-Reinforced Plastics - Tool Deflection and Defect Prediction", Master's Report, University of California at Berkeley.