International Journal of Engineering Research and Technology. ISSN 0974-3154 Volume 10, Number 1 (2017)
© International Research Publication House http://www.irphouse.com

Finite Element Modeling and Simulation of Thin Wall Machining of Al 8011

Sunil S. Karidkar
Assistant Professor, Department of Mechanical Engineering,
K.I.T.'s College of Engineering, Gokul Shirgaon,
Kolhapur, Maharashtra, India.

V. A. Patankar
Research Scholar, Department of Production Engineering,
K.I.T.'s College of Engineering, Gokul Shirgaon,
Kolhapur, Maharashtra, India.

Abstract

Many components used in the aerospace industry are usually thin-wall structures. These thin-walled parts are very easy to deform due to their poor stiffness and low rigidity causing low dimensional precision. These thin complex sectioned components are generally manufactured by the traditional fabrication process in various pieces on various different machines with different set-ups. But this requires a substantial amount of time. After this, additional time required for assembly of these components using riveting or welding process to get the final component. Even though, these traditional processesare less expensive, it requires high part-cost with more time for different set-ups and more laborious work. Thus, machining of these low-rigidity components as a single part is an alternative to the traditional process which eliminates the use of different set-ups and processes, high part-cost, low production and productivity.But machining of these components as a single part is a challenging task. Therefore to study the behavior of these components with different process parameters for effective and efficient operation, process modeling and simulation using appropriate software is highly essential. In this paper, an attempt has been made to model the process using finite element software, ANSYS-15to study componentdeformation. Model deformation is validated with the experimental deformation which shows a very good agreement with about 13% variation.

Keywords: Thin-walled parts, FEM, Simulation, ANSYS, Deformation.

Introduction

Aerospace, marine and power engineering industries often require low rigidity thin sectioned components. These thin complex sectioned components are generally manufactured by the traditional fabrication process in various pieces on various different machines with different set-ups. Then these components are assembled through riveting or welding process to get the final component. Even though traditional process is less expensive, it has higher part-cost with moretime consumption for different set-ups along with more laborious work [1]. Thus, machining of these low-rigidity components as a single part is an alternative to the traditional process which eliminates the use of different set-ups and processes, high-part cost, low production and productivity.The machining of components having many thin sections from a single, monolithic work piece is known as thin-wall machining. The thickness in the range of 1 to 2.5 mm can be considered as a thin-wall [2]. The thin-wall machining reduces part cycle time by machining many thin sectioned parts as a one piece. Advanced machining of thin-wall significantly reduces the need for expensive, time-consuming multiple set-ups and processes. Many manufacturers arefinding it as a way to increase both productivity and profitability compared with traditional fabrication of thin walls.Thin wall machining is performed for varies applications like, die and mould manufacturing, aircraft wings, space industry, heat sink. Such thin-wall products are often large, integral products, with 95% of the blank is machined. But machining of these components as a single part is most difficult task and hence to study in advance the probable deformation of components, most of the authors performed modeling and simulation using different methodologies. Ning et al. [3] applied the finite element method (FEM) to the quantitative analysis and obtained deformation of a typical thin-wall. They observed machining deformation was inverse proportionate to thethickness of the machining walls, but the thickness ofthe remaining three walls has little effect on the deformation. Nahijaet al. [4] determined the optimum operating parameters for the end milling process ofAA6061T6 under wet cooling conditions. They developed a central composite design of response surface methodology to develop an effective analytical model for surface roughness. They analyzed that the surface roughness is more sensitive to feed rate and depth of cut. Tang and Hailong [5] proposed a new analytical deformation model suitable for static machining error prediction of low rigidity components. The part deformation was predicted using a theoretical big deformation equations model, which is established on the basis of the equations of Von Karman when the linear load acts on thin wall plates. The big deformation is simulated using FEA analysis. They analyzed the diverse cutting forces, milling location and thickness of the plate may lead to varies deformation results. Rai and Xirouchakis [6] presented a transient milling simulation model to assist manufacturing engineers in gaining in-depth understanding of the thermo-mechanical aspects of machining and their influence on resulted part quality. Based on the finite-element method approach, the model predicts transient temperature distributions and resulted elastic-plastic deformations induced during the milling of 2.5D prismatic parts comprising features like slots, steps, pockets, etc.. Gang [7] developed a finite element models for helical fluted end mill and the thin-walled workpiece with a cantilever plate structure. The material model of titanium alloy Ti6Al4V was established using the high-temperature SHPB test system. Then author simulated milling process of the thin-walled workpiece of titanium alloy Ti6Al4V using the constitutive equation and the finite element models of the corresponding workpiece and tool. Author adopted a chip separation criterion and the contact friction model in simulation. In conclusion, author performed comparison of simulation and experimental results and observed a reasonable agreement with about 21.56% variation. Thus from above literature, it is well proven that FEM software can be effectively used to predict deformation of a component. In this paper, an attempt has been made to measure and validate the deformation obtained through model with experimental deformation results.

Experimental Setup

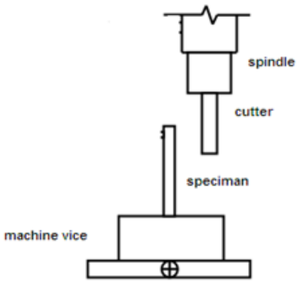


Figure 1: Schematic of the experiment set up

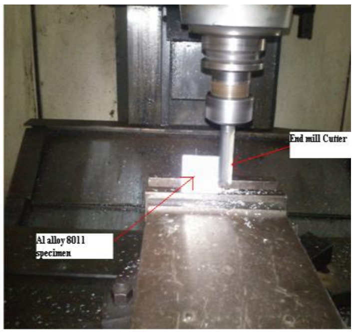


Figure 2: Experimental set-up

The machine used to carry out the experiments was FILLER vertical milling machine with the maximum speed of 4000 rpm and FANUC control. The four fluted flat-end milling cutter of Carbide of diameter 16 mm having helix angle 45^o is used for machining. The schematic of experimental set-up is shown below Figure 1. The prepared specimen is clamped in the machine vice. The end milling cutter is fixed to the spindle.The initial tool set up and programme is fed into the control memory of the VMC for performing the experiments. The experimental set-up is as shown in Figure 2.

Specimens Preparation and Material Properties

The experimental specimen made up of aluminium 8011 alloy is shown in Figure 3 whereas Figure 4 shows its dimensions.Initially the rectangular block of Al alloy 8011 is machined to 6mm wall thickness with 25 mm height of wall.The base length is kept 40 mm with width as 25 mm for clamping purpose.

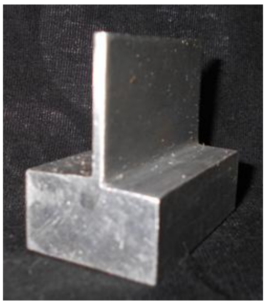


Figure 3: Experiment specimen

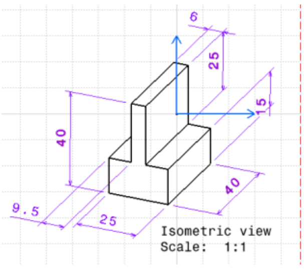


Figure 4: Specimen dimensions

Finite Element Analysis

The part deformation in end milling process is simulated by using FEM ANSYS software. The structural static analysis is used to calculate the deformations for end milling of thin-walled plate. By using FE analysis, the low rigidity part is modeled as a flexible cantilever thin-walled plate. Theinputs to the FE analysis include a set of parameters describing material properties, boundary conditions and other constraints. The radial cutting force is applied with the linear loads, which acts on the contact line between the cutter and the parts. In the process of simulating, the influence of linear loads and deformations of the thin-walled plates are analyzed. The cutting force distributed along the cutter teeth edge is projected to the tool–part intersection line, which distribute among the nodes close to the tool–part intersection line. By using FE analysis, the deformations on every point of the thin-walled plate can be predicted.

Modeling

FEA model is prepared by usingANSYS software. The thickness of wall is 3mm and height of wall is 25 mm. The base is having 25 mm width and 40 mm length. This FEA model is prepared as per specimen for analysis.

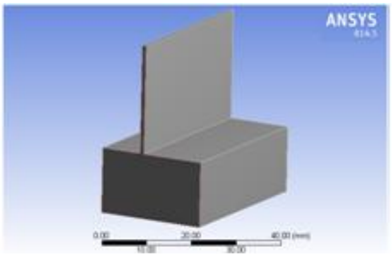


Figure 5:FEA Model

Table 1: Mechanical properties of AA-8011

Density	Tensile Strenght	Young's Modulus	Poisson's ratio
2710 Kg/mm^3	105 MPa	85 Gpa	0.33

Table 2: Process Parameters with levels

Machining Parameters	**Level 1**	**Level 2**	**Level 3**
Cutting speed (rpm)	500	1500	2500
Axial depth of cut (mm)	8.33	12.5	25
Radial depth of cut (mm)	0.25	0.5	1.0
Feed rate (mm/min)	50	60	70

To perform static structural analysis, the important stepsinvolved were,

- Construct a FEM model by 3D-modeling of the required work piece and sequentially using eight nodes structural SOLID-45 brick element, providing the required material properties as listed in Table 1 and mapped meshing is used.
- Apply boundary condition so that the both the ends and the bottom of the thin-wall is constrained. Perform the static analysis by using the proper load and time steps. For applying the load, the function equation is created using function creator of empirical formula equation and apply it using function loader by inputting process parameters listed in Table 2 to the selected nodes which are in contact at the interface of cutting tool-work piece. Apply the average cutting force on the node of that particular thin wall.
- Create required load steps and solve it sequentially to get the nodal displacement in the required direction.

Meshing and Boundary Condition

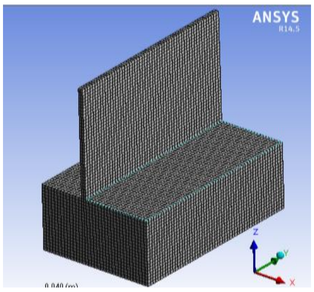


Figure 6 : Component Meshing

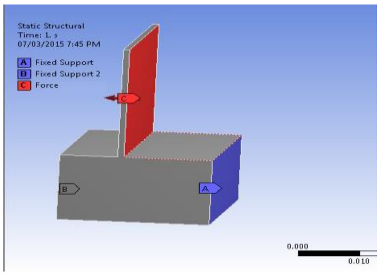


Figure 7 : Boundary condition and applied load

Meshing of model is done by using Hexahedron elements (Ref. Figure 6). It is also called as a square meshing. This meshing gives more numbers of nodes, hence accuracy of results increases. After meshing 194823 numbers of nodes and 44275 numbers of elements were obtained.Apply boundary condition so that the bottom of the thin-wall is constrained. Perform the static analysis by applying load (Ref. Figure 7). For applying the load, the function equation is created using function creator of empirical formula equation (*I* and *II*) and apply it using function loader by inputting process parameters to the selected nodes which are in contact at the interface of cutting tool-work piece.

$$F_x = 1275.3 a_p^{1.1053} f_z^{0.4365} v_c^{-0.0314} a_e^{0.2761} \qquad (I)$$

$$F_y = 35.1075 a_p^{1.0229} a_e^{0.3850} f_z^{0.5490} N^{0.0494} \qquad (II)$$

From these two forces the force acting perpendicular to the axis of milling cutter (F_x) is predominant for deformation. Table 3 shows the values of force in x direction.

Table 3: Force values for each experiments

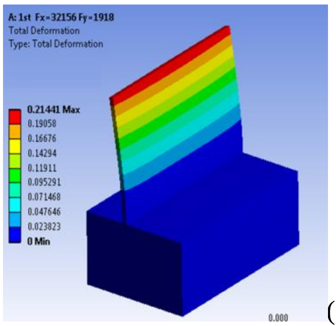
EXPERIMENT NO	FORCE (N)
1	32156
2	742363
3	61491
4	63491
5	42370
6	187163
7	37280
8	162668
9	79074

For FE analysis base section of geometry is constrained in all direction and force is applied on the thin wall in x direction as computed from equation no I.

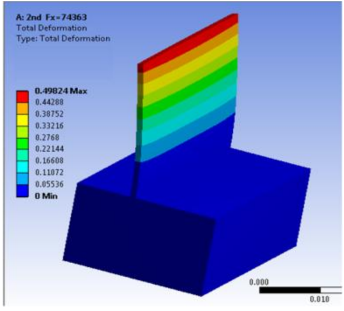
Post Processing

After applying the boundary conditions and load as stated earlier, the values of deformation for wall are obtained. The deformation is of two types one is elastic deformation and second is plastic deformation. Out of which plastic deformation is a permanent deformation and elastic deformation is regained according to the mechanical properties of material. In present FEA maximum deformation of wall is taking into consideration. The maximum deformation of wall isat top as it has less support. The wall deformation is less at the bottom as it is having more support at bottom.

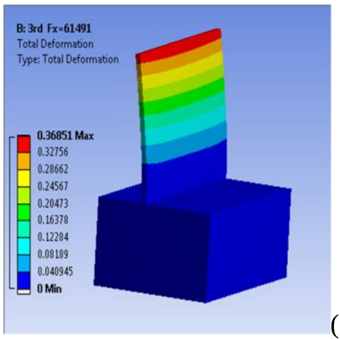
Results



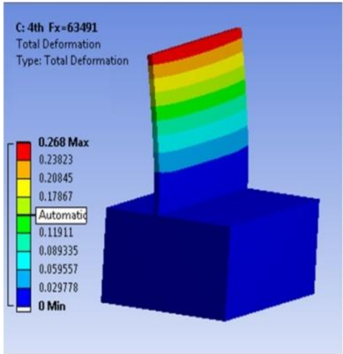
(Expt. No. 1)



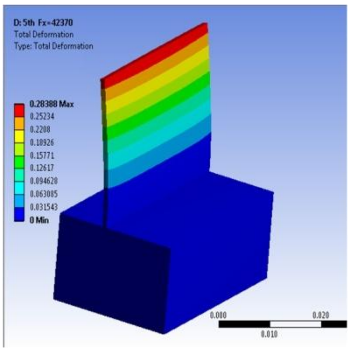
(Expt. No. 2)



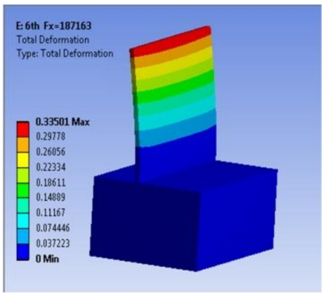
(Expt. No. 3)



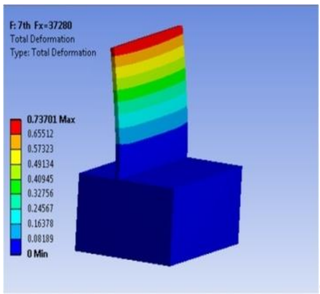
(Expt. No. 4)



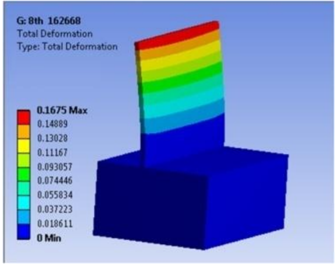
(Expt. No. 5)



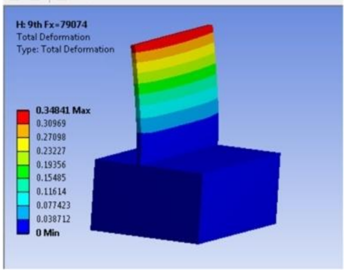
(Expt. No. 6)



(Expt. No. 7)



(Expt. No. 8)



(Expt. No. 9)

Figure 8: Experiment 1 to 9 FEA results for deformation

The part deformation in end milling process is simulated by using ANSYS software. The structural static analysis is thus used to calculate the deformations for end milling of thin-walled plate. The inputs to the FE analysis include a set of parameters describing material properties, boundary conditions and other constraints. Figure 8 shows the FEA results for all experiments.

Table 4: Difference between experimental and model deformation

Expt. No.	**Deform.(mm)**		**Error (mm)**	**% Error**
	Expt.	**FEA**		
1.	0.203	0.211	0.008	3.94
2.	0.489	0.498	0.009	1.84
3.	0.408	0.369	0.039	9.56
4.	0.26	0.268	0.008	3.08
5.	0.263	0.283	0.02	7.60
6.	0.373	0.335	0.038	10.18
7.	0.71	0.737	0.027	3.80
8.	0.149	0.168	0.019	12.75
9.	0.34	0.348	0.008	2.35

The experimental results are compared with FEA results as shown on Table 4.The deformation of thin wall measured in experiments is validated with FEA results. Table 4 clearly shows that the percentage difference of actual deformation measured with FEA results are closely matching with variations of about 13%.

Discussion

The modeldeformation shows a very good agreement with the experimental deformation with some deviation. This deviation might be due to variations in plastic deformation stage. In actual practice, the material is not fully deformed but some amount of material regain its original shape during plastic deformation. Also some problems like improper flushing of debris cannot be idealized into model which surely reduces deformation.

Conclusion

Milling of thin wall components is a common application in aerospace, automotive, power generation, bio-engineering industries. Thus successful thin-wall milling does, require careful consideration of multiple machining factors.In the present investigation, initially the basics of thin wall machining were studied through literature survey withdifferent experimental studies. Various cutting forces and the deflection were measured during as well as after machining.Several modeling techniques were studied to generate the output prediction with the study of fundamentals of milling process by empirical modeling of static cutting forces during milling operation. Finally finite element modeling and simulation of thin wall machining were carried out using ANSYS software and after comparing model and experimental deformation, it was observed that the developed model is capable enough topredict the actual deformation with about 13% variation.

References

[1] Tang Aijun, Liu Zhanqiang, Ma Hailong, "Modeling and Simulation of Big Deformations of Thin-walled Plate in End Milling Process", Proceedings of the IEEE International Conference on Automation and Logistics, Jinan, China, August 18 - 21, 2007.

[2] Bill Kennedy, "Wall SmartCutting Tool Engineering Magazine", volume 59, February 2007.

[3] He Ning, Wang Zhigang, Jiang Chengyu, Zhang Bing, "Finite element method analysis and control stratagem for machining deformation of thin-wall components", Journal of Materials Processing Technology 139, 2003, 332–336.

[4] Najiha M.S. , M.M. Rahman, A.R. Yusoff, "Modling of the end milling process for aluminimun alloy AA6061T6 using HSS tool", Internatinal journal of Automotive and mechanical engineering(IJAME), Dec 2013, 1140-1150.

[5] Tang Aijun, Ma Hailong, "Modelling and simulation of Big deformation of thin walled plate in End milling process", International conference on Automation and logisticts August 18- 21, 2007.

[6] JitenderK.Rai, Paul Xirouchakis, "FEM based prediction of workpiece transient temparature distribution and deformation during milling", International Journal on adv.Manuf. Technol., 42, 2009, 429-449.

[7] Liu Gang, "Study on deformation of titanium thin-wall part in milling process", Journal of materials processing technology 209, 2009, 2788–2793.