Journal of Manufacturing Engineering, March 2015, Vol. 10, Issue. 1, pp 27-34

EXPERIMENTAL ANALYSIS ON DEFORMATION OF 6061-T6 ALUMINIUM BRACKET USING FINITE ELEMENT METHOD

*Hussain Lal A[1], Kannan TTM[2], Giridharan R[3] and Suresh Kumar M[4]

[1]Asst.Professor,Department of mechanical Engineering,MIET Engineering college,Trichy,India.
[2,3&4] Asst.Professor,Department of Mechanical Engineering,PRIST UNIVERSITY,Thanjavur,India

ABSTRACT

Machining fixtures are used to locate and constrain a work piece during a machining operation. To ensure that the work piece is manufactured according to specified dimensions and tolerances, it must be appropriately located and clamped.Minimising work piece and fixture tooling deflections due to clamping and cutting forces in machining is critical to machining accuracy. An ideal fixture design maximizes locating accuracy and workpiece stability, while minimizing displacements. The purpose of this research is to develop a method for modeling workpiece boundary conditions and applied loads during a machining process, analyze modular fixture tool contact area deformation and optimize support locations, using finite element analysis (FEA). ANSYS parametric design language code is used to develop an algorithm to automatically optimize fixture support and clamp locations, and clamping forces, to minimize work piece deformation, subsequently increasing machining accuracy. By implementing FEA in a computer-aided-fixture-design environment, unnecessary and uneconomical “trial and error” experimentation on the shop floor is eliminated.

Keywords *Aluminum Alloy, Fixture, Ansys, Finite element analysis and Optimization.*

1. Introduction

Machining fixtures are used to locate and constrain a work piece during a machining operation. To ensure that the work piece is manufactured according to specified dimensions and tolerances, it must be appropriately located and clamped. Production quality depends considerably on the relative position of the work piece and machine tools. Minimizing work piece and fixture tooling deflections due to clamping and cutting forces in machining is critical to machining accuracy. The work piece deformation during machining is directly related to the work piece- fixture system stiffness. An ideal fixture design maximizes locating accuracy, work piece stability, and stiffness, while minimizing displacements. Traditionally, fixtures were designed by trial and error, which is expensive and time consuming. Research in flexible fixturing and computer-aided-fixture-design (CAFD) has significantly reduced manufacturing lead-time and cost. The purpose of this research is to develop a computer-aided tool to model workpiece boundary conditions and applied loads in machining. This study acknowledges that work piece boundary conditions are deformable and influence the global stiffness of the work piece-fixture system. The boundary conditions of the work piece, the locators, are modeled as multiple springs in parallel attached to the actual work piece-fixture contact area on the surface of the work piece. Also, tangential and normal stiffness components of the boundary conditions are not assumed to be equal as in rigid Coulomb friction, but are assigned independently. In applying loads representative of the machining operation, torque, axial and transverse loads due to feeding are considered. Material properties, element type, and real constants are defined. The work piece is meshed and boundary conditions and loads are applied. The model is then solved and results are retrieved parametrically, and support locations, clamp locations, and clamping forces are optimized to minimize work piece deflection [1].

2. Literature Review

Principles of fixture design and preceding FEA research in fixture design are discussed. Although some research has been conducted in fixture design, a comprehensive finite element model that accurately represents applied boundary conditions and loads has not been developed. Tables 1 and 2 summarize the precedent research conducted on FEA and fixture design. Lee and Haynes [2] used FEA to minimize work piece deflection. Their work piece was modeled as linear elastic; however fixture tooling was modelled as rigid. Their objective function included the maximum work done by clamping and machining forces, the deformation index, and the maximum stress on the

**Corresponding Author - E- mail: ttmk_8@rediffmail.com, er_lal@yahoo.co.in*

workpiece. Their study considers the importance of part deformation with respect to the necessary number of featuring elements and the magnitude of clamping forces [3]. Coulomb’s law of friction was used to calculate the frictional forces the workpiece- fixture contact points. The machining forces were applied at nodal points. Manassa and DeVries [4] conducted similar research to that of Lee and Haynes [2], but modelled fixturing elements as linear elastic springs. Pong et al. [3] used spring-gap elements with stiffness, separation, and friction capabilities to model elastic work piece boundary conditions. Three-dimensional tetrahedral elements were used to mesh the finite element model of the solid work piece. All contacts between the work piece and the fixture were considered to be point contacts and machining forces were applied sequentially as point loads. The positions of locators and clamps, and clamping forces were considered design variables for optimization. Trappey et al. [5] developed a procedure for the verification of fixtures. FEA was used to analyze the stress strain behavior of the work piece when machining and clamping forces were applied.

A mathematical optimization model was formulated to minimize work piece deformation with a feasible fixture configuration. Cai et al. [6] used FEA to analyze sheet metal deformation and optimized support locations to minimize resultant displacements. Kashyap and DeVries [7] used FEA to model work piece and fixture tool deformation, and developed an optimization algorithm to minimize deflections at selected nodal points by considering the support and tool locations as design variables. A summary of research on FEA and fixture design optimization is shown in Table 3. The majority of research conducted in finite element analysis and fixture design optimization, resulted in the development of a mathematical algorithm. Pong et al. [3] used the ellipsoid method to optimize support locations and minimize nodal deflection. Trappey et al. [5] used an external software package, GINO [8], to optimise support locations and clamping forces. Cai et al. [6] used a sequential quadratic programming algorithm in an external FORTRAN based software package, VMCON, to perform a quasi-Newton non-linear constrained optimization of N-2-1 support locations to minimize sheet metal deflection. Kashyap and DeVries [7] developed a discrete mathematical algorithm for optimization.

3. Fixture Design Analysis Methodology

The flowchart in Fig. 1 is a summary of the fixture design analysis methodology developed and used in this work. In summary, workpiece IGES geometry is imported from the solid modeling package, the work piece model is meshed, boundary conditions are applied, the model is loaded, representative of a machining operation, the model is solved, and then boundary conditions are optimized to minimize workpiece deflections.

The work piece model is the starting point of the analysis. This research currently limits the work piece geometry to solids with planar locating surfaces. Some work piece geometry may contain thin-walls and non-planar locating surfaces, which are not considered in this study.

3.1 Geometry

The work piece model, created in Pro/ENGINEER or other solid modeling software is exported to ANSYS in IGES format with all wireframes and surfaces. IGES is a neutral standard format used to exchange models between CAD/CAM/CAE systems. ANSYS provides two options for importing IGES Files, DEFAULT and ALTERNATE. The DEFAULT option allows file conversion without user intervention. The conversion includes automatic merging and creation of volumes to prepare the model for meshing. The ALTERNATE option uses the standard ANSYS geometry database, and is provided for backward compatibility with the previous ANSYS import option. The ALTERNATE option has no capabilities for automatically creating volumes and modes imported through this translator require manual repair through the PREP7 geometry tools. To select the options for importing an IGES file, the IOPTN is used. See Appendix A in [1] for a detailed description of implementation.

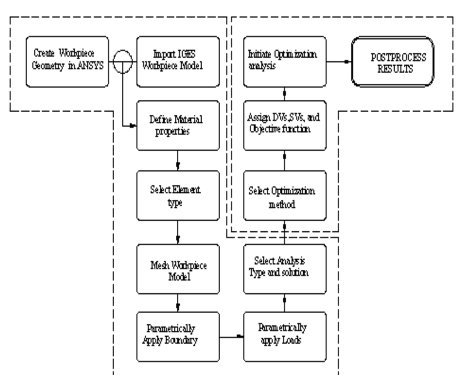

Figure.1

3.2 Material properties

The work piece material in this study is homogenous, isotropic, linear elastic and ductile; this is consistent with the material properties of most metal work pieces. The material selected is SAE/AISI 1212 free-machining grade(a) carbon steel with Young's modulus, E = 30×106 psi Poisson's ratio, í = 0.295, and density, ñ = 0.283 lb/in3, and hardness of 175 HB. Although SAE1212 steel was selected for use in this study because it is commonly used and is a benchmark material for mach inability, any material could be used for the work piece by simply changing the isotropic material properties in ANSYS. Table 4 lists the material properties selected in this study for the work piece and locators.

Table 1 Locator Mechanical properties

Part	Material	E (Pa)	kg/m^3	v	$^s y$ (Pa)
Locators	AISI 1144	2.0×10^{11}	7861	0.295	6.7×10^8

An 8-node hexahedral element (SOLID45), with three degrees of freedom at each node, and linear displacement behaviour is selected to mesh the workpiece. SOLID45 is used for the three-dimensional modeling of solid structures. The element is defined by eight

Nodes having three degrees of freedom at each node: translations in the nodal X, Y, and Z directions. The SOLID45 element degenerates to a 4-node tetrahedral configuration with three degrees of freedom per node. The tetrahedral configuration is more suitable for meshing non-prismatic geometry, but is less accurate than the hex configuration. ANSYS recommends that no more than 10% of the mesh be comprised of SOLID45 elements in the tetrahedral configuration. For a detailed description of the element type selection process, refer to [1].

3.3 Boundary conditions

Locators and clamps define the boundary conditions of the work piece model. The locators can be modelled as point or area contact and clamps are modelled as point forces.

Locators

Point contact. The simplest boundary condition is a point constraint on a single node. A local coordinate system (LCS), referenced from the global coordinate system origin, is created at the centre of each locator contact area, such that the z-axis normal to the work piece locating surface. The node closest to the centre of the local coordinate system origin is selected and all three translational degrees of freedom (u*x*, u*y*, and u*z*) are constrained. The point constraint models a rigid locator with an infinitesimally small contact area. To model locator stiffness and friction at the contact point, a 3- D interface spring-gap element is placed at the centre of the LCS. The element is connected to existing nodes on the surface of the work piece and to a fully constrained copied node offset from the work piece surface in the z-direction of the local coordinate system, i.e., perpendicular to the surface. Figure 2 is a model of the CONTAC52 element used to represent a linear elastic locator.

Area contact. To model a rigid locator with a contact *area*, multiple nodes are fixed within the contact area. An LCS is created on the work piece surface at the centre of the locator contact area. For a circular contact area, a cylindrical LCS is created and nodes are selected at 0 <*r* < *r*L. For a rectangular contact area, a Cartesian LCS is created and nodes are selected at 0 < *x* < *s*ly and 0 < *y* < *y*L. All three translational degrees of freedom (u*x*, u*y*, and u*z)* of each of the nodes are constrained. This model assumes rigid constraints; however in reality locators are elastic. A more accurate representation of the elastic locators consists of multiple ANSYS CONTAC52 elements in parallel. Nodes are selected within the locator contact area and are copied offset perpendicular to the locating surface. Each selected node is connected to the copied node sequentially with the CONTAC52 element. Figure 3 shows the contact area model with multiple spring-gap elements in parallel used to represent a linear elastic locator. It is important to note, that the user is constrained to the number of nodes within the specified contact area, when attaching the CONTAC52 elements. It is possible that there could be a different number of elements modeling each locator, because of the number of associated nodes within the contact area. Thus, the element normal and tangential stiffness, which is specified in the real constant set, would vary. For this reason, multiple real

Constant sets must be created for the CONTAC52 element, and then assigned accordingly when creating elements in a specified local coordinate system.

In Fig. 03, the method for obtaining the normal and tangential stiffness for a locator is shown. The stiffness divided by the total number of springs is assigned accordingly to each spring-gap element, in the real constant set. A point load is applied to the three-dimensional finite element model of the real locator, normal to the contact area to determine the normal stiffness. A point load is applied tangent to the contact area of the real locator to determine the tangential or "sticking" stiffness of the locator. The stiffness values are then assigned to the CONTAC52 elements.

Clamps – The clamps are used to fully constrain the work piece once it is located. It is common to use multiple clamps and clamping forces that are generally constant for each clamp. The clamping force, F*cl* is applied through either a toggle mechanism or a bolt mechanism, which lowers a strap that comes into contact with the work piece. Although friction is just as important in clamping as it is in locating, it is not modelled at the clamp contact area due to limitations in ANSYS. In order to model friction, a comprehensive three-dimensional model of the entire workpiece-fixture system is required, with contact and target surfaces defined at the fixture-work piece contact areas. The

Clamping forces are modelled in ANSYS as point loads on nodes selected either within a rectangular area for a clamp strap or a circular area on the work piece surface for a toggle clamp. Both clamps may also be modeled with a single point load at the center of the clamp contact area.

3.4 Loading

The two machining operations, milling and drilling, are discussed. The purpose of this research is not to accurately model the machining process, but to apply the torque and forces that are transferred through the work piece in machining, to determine the reactions at the boundary conditions of the work piece. The desired

Result of the load model is the trend of rotation from the applied torque of the cutting tool, and translation, due to axial feeding of the work piece and transverse motion of the table in milling.

Milling – The loading in a milling operation involves an axial load, a transverse load due to the linear feeding of the work piece, a torque to generate tool rotation, which is transmitted through the work piece, and shear force in the cutting area. Figure 6 is the loading model for end milling.

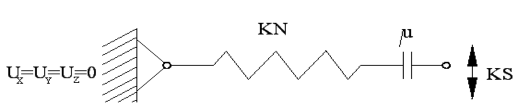

Figure.2

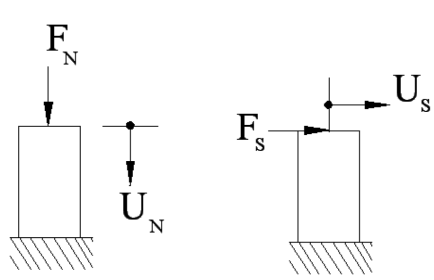

Figure .3

The end milling model is the same as the drilling model, with the transverse load added. Because the objective of the analysis is to determine the maximum resultant displacements and equivalent stresses in the work piece during the operation and tool entry are not considered, only the average steady-state load magnitude is addressed. In this study, the cutting forces are applied as steady-state loads. In previous FEA research, forces in milling were traditionally modelled as steady-state single point loads and torque was neglected. The axial load due to feeding can be applied as multiple point loads on the cutting tool perimeter or as a single point load. The transverse load, F*tri*, is applied as a single point load at the centre of the cutting tool.

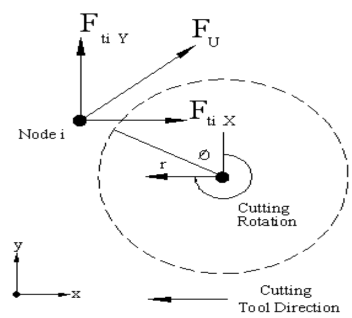

Figure.4

4. Fixture Design Optimization

In order to minimize work piece deformation and maximize locating accuracy, the boundary conditions (support locations and clamp location, and clamping force magnitude) of the model are optimized. The object of optimization is to maximize machining accuracy by minimizing work piece deformation. The locators satisfy two functional requirements: (1) Locate and stabilize the work piece, and (2) Serve as supports to minimize work piece deflections. The optimization analysis attempts to satisfy both functional requirements with a single design parameter, the position of the locators on the workpiece surface. The optimization analysis is performed in ANSYS 11. The ANSYS program offers two optimization methods to accommodate a wide range of optimization problems. The sub problem approximation method is an advanced zero-order method that can be efficiently applied to most engineering problems.

The firstorder method is based on design sensitivities and is more suitable for problems that require high accuracy. For both the sub problem approximation and first-order methods, the program performs a series of analysis-evaluation-modification cycles. That is, an analysis of the initial design is performed, the results are evaluated against specified design criteria, and the design is modified as necessary. This process is repeated until all specified criteria are met. In addition to the two optimisation techniques available, the ANSYS program offers a set of strategic tools that can be used to enhance the efficiency of the design process. For example, a number of random design iterations can be performed. The initial data points from the random design calculations can serve as starting points to feed the optimization methods mentioned above.

The design variables, state variables, and objective function are referred to as the optimization variables. In an ANSYS optimization, these variables are represented by user-named variables called parameters. The user must identify which parameters in the model are design variables (DVs), which are state variables (SVs), and which is the objective function. The analysis file is an ANSYS input file that contains a complete analysis sequence (preprocessing, solution, and post processing). It must contain a parametrically defined model, using parameters to represent all inputs and outputs, which will be used as DVs, SVs, and the objective function. The loop file resides in the working directory and is used by the control file to build the model. The control file initializes the design variables, defines the feasible design space, optimization analysis method, and looping controls, and executes the optimization analysis

A loop is one pass through the analysis cycle. Output for the last loop performed is saved on *Jobname*.OPT or resumed at any time in the optimizer [10]. DVs are independent quantities that are varied in order to achieve the optimum design. Upper and lower limits are specified to serve as “constraints” on the design variables. The design variables in the optimization are locator and clamp positions, and clamping force. SVs are quantities that constrain the design. They are also known as “dependent variables”, that are functions of the design variables. A state variable may have a maximum and minimum limit, or it may be “single sided”, having only one limit. The state variable is the von Mises effective stress. The objective function is the dependent variable that you are attempting to minimize. It should be a function of the DVs, that is, changing the values of the DVs should change the value of the objective function.

The objective function is the maximum resultant displacement in the model. Table 5 lists all the optimization variables used in this study. A design set is simply a unique set of parameter values that represents a particular model configuration. Typically, a design set is characterized by the optimization variable values, however, all model parameters, including those not identified as optimization variables, are included in the set. A feasible design is one that satisfies all specified constraints on the SVs as well as constraints on the DVs. If any one of the constraints is not satisfied, the design is considered infeasible. The best design is the one that satisfies all constraints and produces the minimum objective function value. If all design sets are infeasible, the best design set is the one closest to being feasible, irrespective of its objective function value [10]. Because there are a finite number of positions where the modular tooling can be fastened to the base plate, the optimization algorithm is discrete. There are also geometric constraints on the locators and clamps. For example, although it would be ideal to position the primary reference plane supports directly under the applied load during machining, since the forces would be transferred directly through the support and the bending moment would be zero, it is impractical in some instances, such as in the drilling of a through hole, because of interference with the support. For maximum work piece stability and locating accuracy the supports on the primary reference plane should be placed as far apart as possible.

However, to minimize work piece deformation, the supports should be placed as close to the loads normal to the primary surface as possible. The support locations are optimized where work piece deflections are minimized and locating accuracy is highest. Locating accuracy, work piece stability, and work piece deformations are all affected by the support locations and contribute to the overall fixture stiffness and subsequently, the machining accuracy [3].

5. ANSYS Optimization Study

A sample optimization analysis was conducted to demonstrate the validity of the ANSYS parametric design language (APDL) batch code. As mentioned in the fixture design analysis methodology section in Part I, the optimization analysis is used to minimize the maximum resultant displacement in the work piece, by optimizing support locations, clamp locations, and clamping force magnitudes. The same 3-2-1 fixture configuration used for the work piece in the loading study, was used as the initial configuration in the optimization analysis. The algorithm for selecting initial support locations is explicitly described in the loading study. Three feasible design sets resulted from the optimization analysis. The results are listed in Table 6. Design set 1 is the initial fixture configuration. Design set 2 is the optimized configuration given a limited design space, as shown in Fig. 7. Design set 3 is the optimised configuration given an extended design space. The design space for the optimization analysis resulting in design set 2 is shown in Fig. 7 as a dashed square. The design space for the optimization analysis resulting in design set 3 was extended to include the entire surface on each reference plane. The von Mises stress at each support location is compared to the yield stress of the work piece material, AISI 1212 Steel, $\acute{o}y$ = 58 015 psi, to ensure that the material does not exhibit plastic deformation during machining. The von Mises stress is treated as a state variable and is not allowed to exceed the work piece material yield strength.

The von Mises stresses at the locators on the secondary and tertiary reference planes (SEQV1, SEQV2, and SEQV3) vary between design sets due to their position and the magnitude of the clamping forces. Notice that on the primary reference plane, the von Mises stresses (SEQV4) remain relatively constant, since the axial thrust force magnitude is constant.

The clamping force is increased to 249 lbf in design set 2 from 100 lbf in design set 1. In design set 3, it is only increased to 112 lbf. The maximum resultant displacement was subsequently reduced by 8.4%, from 1.47×10-3 in. (design set 1) to 1.34×10-3 in (design set 2). In design set 3, the optimized fixture configuration did not vary significantly from the initial configuration. The maximum resultant displacement was only reduced by 0.75% from 1.47×10-3 in to 1.46×10-3 in.

Design set 2, note that the locators on the primary reference plane (4, 5, and 6) were moved closer to the centre of the plane to minimize deflections due to the applied axial load. The locators on the secondary and tertiary reference planes were moved up to minimize deflections due to the applied torque. It is obvious that without some knowledge base in fixture design, the optimization analysis is meaningless. An initial fixture configuration must be provided. If all of the supports are initially placed at the global coordinate system origin, for example, the optimization analysis will not result in a feasible design set. The user must also specify the design space, by selecting the range of values for the design variables.

The locators were modelled with multiple reference plane is perpendicular to the direction of applied loading, no clamp is necessary opposite the locator. A list of brake work piece to the locator without generating any bending moments. Because the tertiary ANSYS CONTAC52 spring-gap elements in parallel, attached to a circular contactarea at specified fixturing points on the brake caliper. The loading is representative of a boring operation. The maximum resultant displacement in the preloaded work piece model is 0.000297 mm, and increases slightly to 0.000297 mm in the fully loaded work piece model, thus it is evident that the preloading due to clamping is the major contribution to the resultant displacement throughout the machining operation.

The displacement near the cylinder bore at the centre of the cylinder. The configuration is shown in Fig. 8. The clamps are placed directly opposite the locators on each reference plane, so that the clamping force is transferred directly through the cylinder bore increases significantly, by as much as 100%, but does not exceed the maximum resultant displacement in the preloaded work piece model. Figures 9 and 10 are the resultant displacement and developed in this study. The work piece model is a simplified die cast aluminium brake caliper two locators are placed on the secondary reference plane.

Optimization variable	Variable type	Design set 1 (feasible)	Design set 2 (feasible)	Design set 3 (feasible)
SEQV1	SV	1.51×10^7 Pa	1.51×10^8 Pa	2.24×10^8 Pa
SEQV2	SV	6.39×10^7 Pa	1.16×10^8 Pa	7.21×10^7 Pa
SEQV3	SV	3.50×10^6 Pa	2.94×10^8 Pa	2.05×10^6 Pa
SEQV4	SV	1.56×10^8 Pa	1.89×10^8 Pa	1.81×10^8 Pa
FCL1	DV	444.8 N	1.107×10^3 N	498.2 N
FCL2	DV	444.8 N	1.107×10^3 N	498.2 N
DMAX	OBJ	0.000297 mm	0.00026 mm	0.000295 mm

Table 6. Brake caliper model parameters And results

6. Industrial Optimization Case Studies

An industrial case study was conducted to validate the fixture design analysis method which is on the side of the caliper, and one locator is placed on the tertiary reference plane, directly behind the von Mises stress plots, respectively, for the preloaded model (clamping loads, no machining loads).

Figure5.Work piece with Fixture

Table 7.Optimised brake caliper locator and clamp Position

Element type	ANSYS SOLID45 4-node tetrahedral
Mesh type	Free tetrahedral
Work piece material type	6061-T6 aluminum
Locator material type	AISI 1144 steel
Locator normal stiffness	1.75×10^5 N/mm
Locator tangential stiffness	1.75×10^4 N/mm
Young's modulus, E	7.0×10^{10} Pa
Work piece material yield strength, $^s y$	1.7×10^8 Pa
Poisson's ratio,	0.35
Coefficient of static friction, μ	0.61
Thrust force, Fc	249.1 N
SEQV1	7.67×10^5 Pa
SEQV2	5.95×10^{-5} Pa
SEQV3	7.40×10^5 Pa
SEQV4	1.31×10^5 Pa
Clamping force, FCL1	200 N
Clamping force, FCL2	200 N
DMAX	0.000297 mm

	Initial configuration (mm)			Optimized configuration (mm)		
Locator	X	Y	Z	X	Y	Z
1	47.95	27.00	-99.50	47.95	27.00	-99.50
2	47.95	27.00	99.50	47.95	27.00	99.50
3	143.85	48.00	0.00	143.85	58.00	0.00
4	0.00	0.00	0.00	0.00	0.00	0.00
Clamp	X	Y	Z	X	Y	Z
1	47.95	-20.00	-99.50	47.95	-20.00	-99.50
2	47.95	-20.00	99.50	47.95	-20.00	99.50

Optimized experimentally

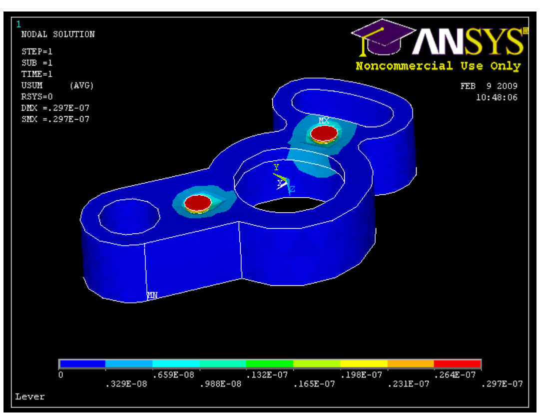

Fig. 5 Resultant displacement (mm) Contour plot

Displacement and von Mises stress plots, respectively, for the loaded model. The maximum von Mises stress occurs at the contact area of clamp 3, located opposite locator 3 on the primary reference plane. An optimization analysis was conducted to

Because it results in slightly larger displacements. Because the local state of stress is not of concern, the point load is as appropriate as a distributed load for the purpose of work piece deflection analysis. The torque component of the load model is critical to work piece deformation.

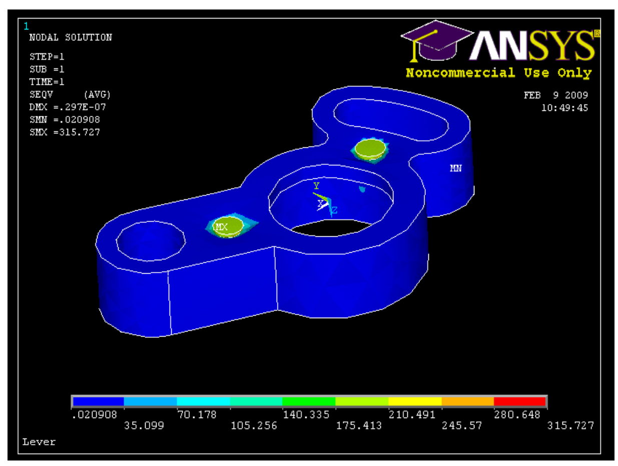

Fig. 6 Von Mises stress (Mpa) Contour plot

7. Conclusions

In this study a finite element model was developed for fixture work piece boundary conditions and applied loads in machining using ANSYS. As opposed to preceding finite element analysis research in fixture design, in this study, boundary conditions modelled as both area and point constraints were considered to determine whether a single point constraint model is appropriate. Only Pong et al. [3] modelled boundary conditions to be elastic and deformable, but this research only considered elastic point constraints. His research does not specify whether an elastic area constraint model was considered.

A more accurate representation of machining loads was also developed. The load model developed in this study includes torque, which is neglected in all preceding research. Distributed and concentrated loading is considered in this study, whereas in previous research all machining forces are applied as single point loads. Because the model boundary conditions and loads are applied parametrically, APDL code can be used for solid models with planar locating surfaces and user defined (1) support locations, (2) clamp locations, (3) clamping force magnitude,(4) cutting tool location, (5) axial load, (6) transverse load, and(7) torque magnitude.

The following analysis specific conclusions are realized based on the research conducted throughout this study:

Work piece elements. The SOLID45, 8-node brick element, is suitable for meshing prismatic geometry. The SOLID45, 4-node tetrahedral element is not as accurate as the brick element, but is suitable for displacement analysis of non-prismatic geometry.

Load model. It is appropriate to model the cutting tool axial load with a single point load for large work piece surface area to cutting tool contact area ratios. In addition to being easier to apply, the single point load is more conservative.

References

1. *Amaral N, (2001) "Development of a finite element analysis tool for fixture design integrity verification and optimization", Dissertation, Mechanical Engineering Department, Worcester Polytechnic Institute, Worcester, MA*

2. *Lee J D Haynes L S (1997), "Finite element analysis of flexible fixture system", Trans ASME J Eng Ind Vol. 109(2), 134–139.*

3. *Pong P C Barton R R Cohen P H (1993), "Optimum fixture design" Proceedings of the 2nd Industrial Engineering Research Conference, Los Angeles, 26–28 May-1993, IIE Norcross, GA, pp 6–10.*

4. *Menassa R DeVries W (1997), "Optimization methods applied to selecting Support positions in fixture design" Trans ASME J Eng Ind Vol. 113,412–418.*

5. *Trappey A J C Su C S Hou J L (1995), "Computer-aided fixture analysis using finite element analysis and mathematical optimization modeling", Proceedings of the 1995 ASME International Mechanical Engineering Congress and Exposition, Part 1, 12–17 Nov 1995. ASME, New York, pp 777–787*

6. *Cai W Hu S J Yuan J X (1996), "Deformable sheet metal fixturing: principles, Algorithms and simulations", Trans ASME J Eng Ind Vol.118, 318–324.*

7. *Kashyap S DeVries W R (1999), "Finite element analysis and optimization In fixture design", Struct Optim Vol.18(2–3),193–201.*

8. *Bradly Associates (2001), GINO Graphics. Bradly Associates, Crowthorne,Berkshire.*

9. *Hillstrom K E Minkoff M (2001), VMCON. Applied Mathematics Division.*

10. *ANSYS, Inc. (2001), "Alternate online manual set", Texas A&M University Supercomputing Facility, College Station, TX.*

