Int J Adv Manuf Technol (2005) 25: 409–419
DOI 10.1007/s00170-003-1796-6

ORIGINAL ARTICLE

Nicholas Amaral · Joseph J. Rencis · Yiming (Kevin) Rong

Development of a finite element analysis tool for fixture design integrity verification and optimisation

Received: 15 March 2003 / Accepted: 11 May 2003 / Published online: 25 August 2004

Abstract Machining fixtures are used to locate and constrain a workpiece during a machining operation. To ensure that the workpiece is manufactured according to specified dimensions and tolerances, it must be appropriately located and clamped. Minimising workpiece and fixture tooling deflections due to clamping and cutting forces in machining is critical to machining accuracy. An ideal fixture design maximises locating accuracy and workpiece stability, while minimising displacements.

The purpose of this research is to develop a method for modelling workpiece boundary conditions and applied loads during a machining process, analyse modular fixture tool contact area deformation and optimise support locations, using finite element analysis (FEA). The workpiece boundary conditions are defined by locators and clamps. The locators are placed in a 3-2-1 fixture configuration, constraining all degrees of freedom of the workpiece and are modelled using linear spring-gap elements. The clamps are modelled as point loads. The workpiece is loaded to model cutting forces during drilling and milling machining operations.

Fixture design integrity is verified. ANSYS parametric design language code is used to develop an algorithm to automatically optimise fixture support and clamp locations, and clamping forces, to minimise workpiece 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 FEA · Finite · Element · Analysis · Fixture · Optimisation

N. Amaral (✉)
V-Engine Manufacturing Engineering,
Ford Motor Company, Powertrain Operations,
21500 Oakwood Boulevard, Dearborn, MI 48124-4091 USA
E-mail: namaral@ford.com
Fax: +1-313-2486734

J.J. Rencis · Y. Rong
Worcester Polytechnic Institute,
100 Institute Road, Worcester, MA, 01609-2280 USA

1 Introduction

Machining fixtures are used to locate and constrain a workpiece during a machining operation. To ensure that the workpiece 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 workpiece and machine tools. Minimising workpiece and fixture tooling deflections due to clamping and cutting forces in machining is critical to machining accuracy. The workpiece deformation during machining is directly related to the workpiece-fixture system stiffness. An ideal fixture design maximises locating accuracy, workpiece stability, and stiffness, while minimising 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.

The majority of finite element analysis (FEA) research conducted in fixture design considers workpiece boundary conditions to be rigid and applied loads to be concentrated. In all cases where friction is considered, rigid Coulomb friction is assumed. Cutting tool torque, which results in a trend of workpiece rotation, is not considered. Clamping forces are considered to be constant point loads.

This study acknowledges that workpiece boundary conditions are deformable and influence the global stiffness of the workpiece-fixture system. The boundary conditions of the workpiece, the locators, are modelled as multiple springs in parallel attached to the actual workpiece-fixture contact area on the surface of the workpiece. 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. An indepth discussion of the work presented herein can be found in Amaral [1].

In this study, both the finite element analysis and optimisation are conducted in ANSYS. Within the analysis, a workpiece is imported in initial graphics exchange specification (IGES) format. Material properties, element type, and real constants are defined. The workpiece 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 optimised to minimise workpiece deflection [1]. The advantage of the method developed herein is that an external software package for optimisation is not required, thus compatibility between two packages is not a concern.

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 represent applied boundary conditions and loads has not been developed. Tables 1 and 2 summarise the precedent research conducted on FEA and fixture design.

Lee and Haynes [2] used FEA to minimise workpiece deflection. Their workpiece was modelled 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 workpiece. Their study considers the importance of part deformation with respect to the necessary number of fixturing elements and the magnitude of claming 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 workpiece boundary conditions. Three-dimensional tetrahedral elements were used to mesh the finite element model of the solid workpiece. All contacts between the workpiece 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 optimisation. Trappey et al. [5] developed a procedure for the verification of fixtures. FEA was used to analyse the stress-strain behaviour of the workpiece when machining and clamping

Table 1. Literature survey of workpiece models

Reference	Workpiece model				
	Material				Element type
	Type	E (Pa)	ν	μ	
Lee and Haynes [2]	Steel homogeneous Isotropic linear elastic	6.9×10^{8}	0.3	U/A*	3-D solid 8-node brick
Pong et al. [3]	Aluminium homogeneous Isotropic linear elastic	6.9×10^{10}	0.3	U/A	3-D solid 10-node tetrahedral; ANSYS SOLID92
Trappey et al. [5]	Aluminium homogeneous Isotropic linear elastic	6.9×10^{10}	0.3	0.3	U/A
Cai et al. [6]	Steel Isotropic linear elastic	2.1×10^{11}	0.3	U/A	2-D 4-node rectangular element; MSC NASTRAN QUAD4
Kashyap and DeVries [7]	Aluminium homogeneous Isotropic linear elastic	6.9×10^{10}	0.3	U/A	3-D solid tetrahedral elements

*U/A: unavailable

Table 2. Literature survey of boundary conditions and loading

Reference	Fixture component model		Steady-state load model	
	Locators	Clamps	Drilling	Milling
Lee and Haynes [2]	Rigid area constrain, Rigid coulomb friction	U/A*	U/A	Normal and shear point loads
Pong et al. [3]	3-D spring-gap interface element, Rigid coulomb friction	N/A**	Normal point loads	N/A
Trappey et al. [5]	3-D solid deformable constraints	Point loads	Normal point loads	Normal and shear point loads
Cai et al. [6]	Rigid point constraints	N/A	Normal point loads	Normal and shear point loads
Kashyap and DeVries [7]	Rigid point constraints	Point loads	Normal point loads	Normal and shear point loads

*U/A: unavailable
**N/A: not applicable

forces were applied. A mathematical optimisation model was formulated to minimise workpiece deformation with a feasible fixture configuration.

Cai et al. [6] used FEA to analyse sheet metal deformation and optimised support locations to minimise resultant displacements. Kashyap and DeVries [7] used FEA to model workpiece and fixture tool deformation, and developed an optimisation algorithm to minimise deflections at selected nodal points by considering the support and tool locations as design variables.

A summary of research on FEA and fixture design optimisation is shown in Table 3. The majority of research conducted in finite element analysis and fixture design optimisation, resulted in the development of a mathematical algorithm. Pong et al. [3] used the ellipsoid method to optimise support locations and minimise 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 optimisation of N-2-1 support locations to minimise sheet metal deflection. Kashyap and DeVries [7] developed a discrete mathematical algorithm for optimisation.

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 modelling package, the workpiece 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 optimised to minimise workpiece deflections.

3.1 Workpiece model

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

Geometry – The workpiece model, created in Pro/ENGINEER or other solid modelling 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

Table 3. Literature survey of optimisation analysis

Reference	Optimization analysis		
	Method	Objective function	Software package
Pong et al. [3]	Ellipsoid method	Nodal deflection	N/A*
Trappey et al. [5]	Non-linear mathematical algorithm	Nodal deflection	GINO [8]
Cai et al. [6]	Sequential quadratic programming algorithm	Nodal deflection normal to sheet metal surface	VMCON [9]
Kashyap and DeVries [7]	Discrete mathematical algorithm	Nodal deflection	N/A

*N/A: not applicable

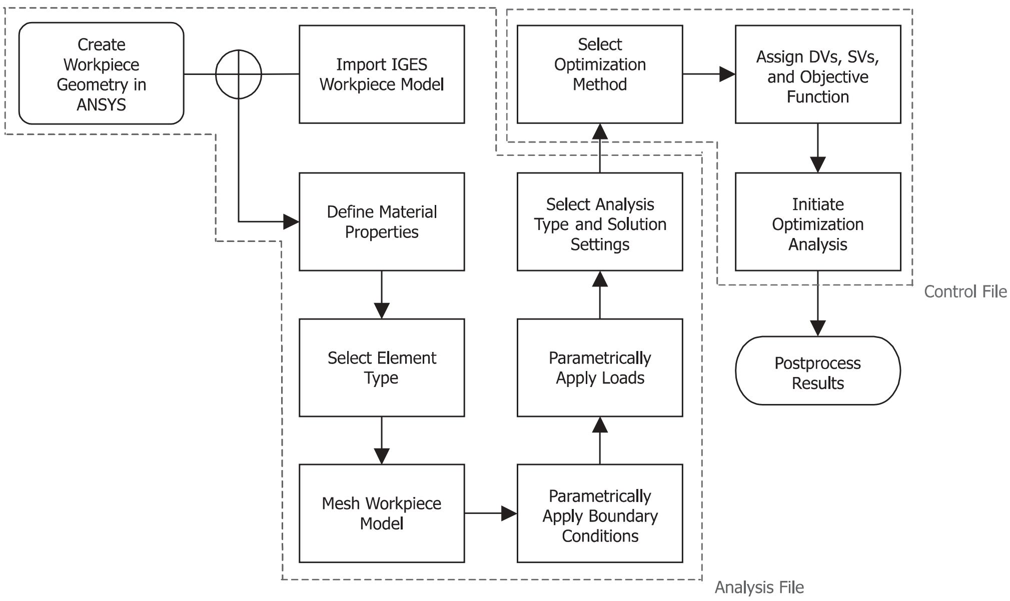


Fig. 1. Fixture design analysis methodology

Table 4. Workpiece and locator material properties

	Material	E (Pa)	ρ (kg/m^3)	ν	σ_y (Pa)
Workpiece	AISI 1212	2.0×10^{11}	7861	0.295	2.3×10^8
Locators	AISI 1144	2.0×10^{11}	7861	0.295	6.7×10^8

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.

Material properties – The workpiece material in this study is homogenous, isotropic, linear elastic and ductile; this is consistent with the material properties of most metal workpieces. The material selected is SAE/AISI 1212 free-machining grade(a) carbon steel with Young's modulus, $E = 30 \times 10^6$ psi Poisson's ratio, $\nu = 0.295$, and density, $\rho = 0.283$ lb/in^3, 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 machinability, any material could be used for the workpiece by simply changing the isotropic material properties in ANSYS. Table 4 lists the material properties selected in this study for the workpiece and locators.

3.2 Meshed workpiece model

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 modelling 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 workpiece 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 workpiece 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 workpiece and to a fully constrained copied node offset from the workpiece 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 workpiece 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 $0 < x < x_L$ 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 modelling 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. 4, the method for obtaining the normal and tangential stiffness for a locator is shown. The stiffness divided by the total

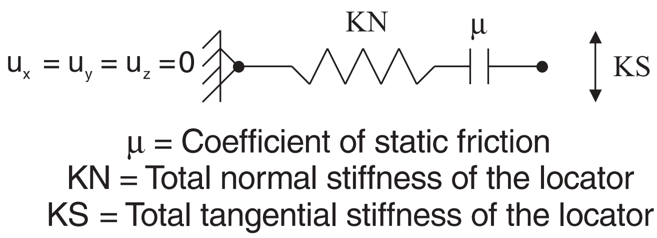


Fig. 2. CONTAC52 element used to model point contact for locators [10]

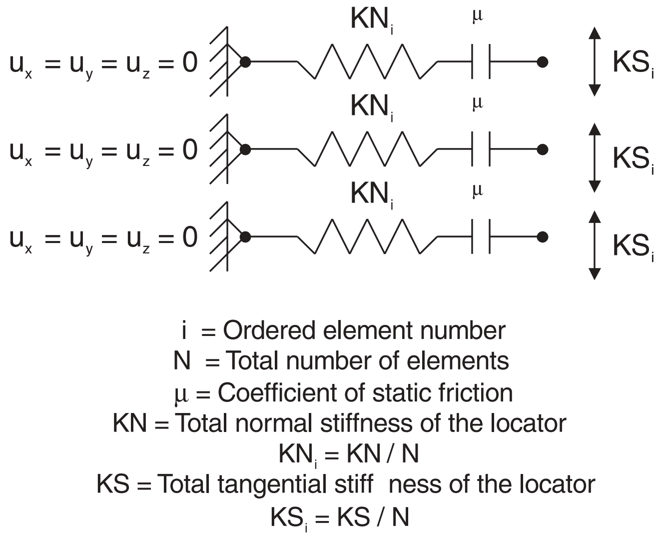
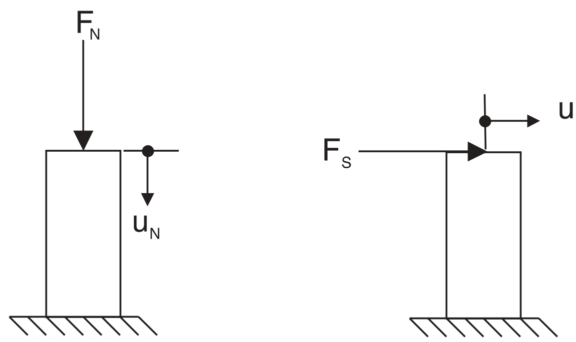


Fig. 3. CONTAC52 elements in parallel, used to model area contact for locators [10]



$$K_N = \frac{|F_N|}{|\delta_N|} \qquad K_S = \frac{|F_S|}{|\delta_S|}$$

Fig. 4. Normal and tangential stiffness for locator

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 workpiece 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 workpiece. 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-workpiece 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 workpiece surface for a toggle clamp. Both clamps may also be modelled with a single point load at the centre 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 workpiece in machining, to determine the reactions at the boundary conditions of the workpiece. 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 workpiece and transverse motion of the table in milling.

Drilling – The forces in a drilling operation include a torque, T, to generate tool rotation, shear force, V, created by tool rotation at the cutting edge contact for chip removal, and an axial load, P, due to feeding. The forces in drilling are time and position dependent and oscillatory due to cutter rotation, since the cutting edge of the tool is not in constant contact with the workpiece at a particular location. The cutting force increases monotonically during tool entry and then approaches steady-state. Fluctuations in the cutting force are due to cutting tool tooth distribution during rotation. In this study, the torque and thrust forces in feeding are applied as steady-state loads, since initial tool entry is not considered. In previous FEA fixture design research, loads were applied as a steady-state. Also neglected was cutting tool torque and subsequently the workpiece deflections due to the trend of rotation in the fixture. An initial attempt to model the distributed loading using a number of point loads applied at key points was unsuccessful, due to limitations in ANSYS. The model consisted of placing key points on a local coordinate system created on the machining surface of the workpiece. The key points were located at exact R, θ, and Z positions on the cutting tool perimeter. At each key point forces were applied to model a drilling operation. The torque was modelled with tangential forces placed at the outer radius of the cutting tool contact area. The tangential couple forces were decomposed into global X and Y components. The axial load was modelled by applying forces at each key point in the global Z direction. The reason this model failed is that the key points created on the workpiece surface are geometric entities and are not part of the finite element mesh, i.e., key points are not nodes. Due to this limitation in ANSYS, the point load model was modified to apply loads at existing nodes on the workpiece surface. Figure 5 shows the modified load model for drilling. Notice that node *i* is slightly offset from the cutting tool perimeter. Because a node may not exist in the exact location specified by R, θ, and Z, the node closest to that location in the local coordinate system is selected and forces are applied as point loads with global X, Y, Z components. The user may minimise the distance between a specified coordinate location and an existing node by

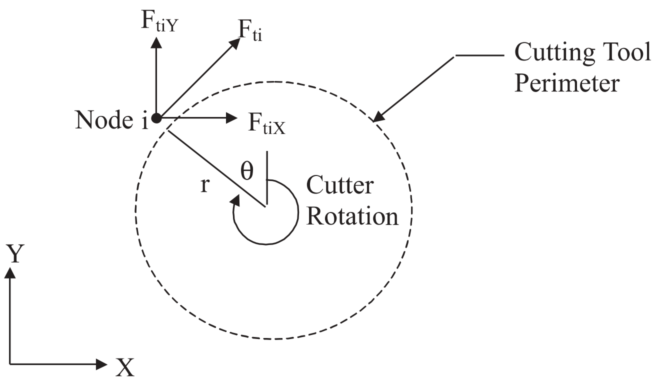


Fig. 5. Drilling load model

increasing the mesh density. The nodes are selected at equivalent θ intervals on or near the cutting tool perimeter. At each selected node, global X and Y components of the tangential couple force, F_{ti} and axial load component, F_{ci} are applied. The applied torque is equal to the sum of the tangential forces multiplied by the cutting tool radius, r. F_{tiX} and F_{tiY} are the global X and Y components, respectively, of the tangential force, F_{ti}. F_{ci} is equal to the total axial load, F_c, divided by the number of nodes over which it is applied.

A simplified model entails the use of a single point force normal to the surface of the workpiece to model the cutting tool axial load and a couple to model the applied torque. A study was conducted to determine whether multiple point forces applied along the cutting tool perimeter are actually necessary to model the axial load and assess the validity of the simplified model

Milling – The loading in a milling operation involves an axial load, a transverse load due to the linear feeding of the workpiece, a torque to generate tool rotation, which is transmitted through the workpiece, and shear force in the cutting area. Figure 6 is the loading model for end milling. 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 workpiece during the operation and tool entry are not considered, only the

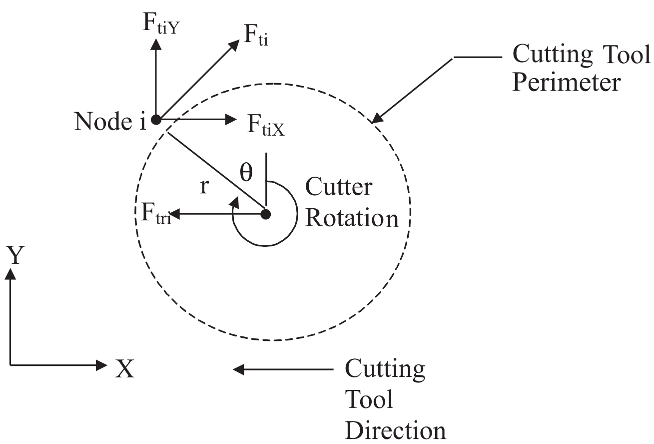


Fig. 6. Milling load model

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.

4 Fixture design optimisation

In order to minimise workpiece deformation and maximise locating accuracy, the boundary conditions (support locations and clamp location, and clamping force magnitude) of the model are optimised. The object of optimisation is to maximise machining accuracy by minimising workpiece deformation. The locators satisfy two functional requirements: (1) Locate and stabilise the workpiece, and (2) Serve as supports to minimise workpiece deflections. The optimisation analysis attempts to satisfy both functional requirements with a single design parameter, the position of the locators on the workpiece surface.

The optimisation analysis is performed in ANSYS 5.6.2. The ANSYS program offers two optimisation methods to accommodate a wide range of optimisation problems. The subproblem approximation method is an advanced zero-order method that can be efficiently applied to most engineering problems. The first-order method is based on design sensitivities and is more suitable for problems that require high accuracy. For both the subproblem 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 optimisation methods mentioned above.

The design variables, state variables, and objective function are referred to as the optimisation variables. In an ANSYS optimisation, 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 postprocessing). 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 initialises the design variables, defines the feasible design space, optimisation analysis method, and looping controls, and executes the optimisation analysis

(Looman D, ANSYS Corporate Technical Support Center, 2001, personal communication).

A loop is one pass through the analysis cycle. Output for the last loop performed is saved on file *Jobname*.OPO. An optimisation iteration is one or more analysis loops which result in a new design set. Typically, an iteration equates to one loop. However, for the first order method, one iteration represents more than one loop. The optimisation database contains the current optimisation environment, which includes optimisation variable definitions, parameters, all optimisation specifications, and accumulated design sets. This database can be saved to *Jobname*.OPT or resumed at any time in the optimiser [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 optimisation 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 minimise. 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 optimisation 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 characterised by the optimisation variable values, however, all model parameters, including those not identified as optimisation 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].

Table 5. Optimisation variables

Design variables	Position of locators
	Locator 1 (X_1, Y_1, Z_1)
	Locator 2 (X_2, Y_2, Z_2)
	Locator 3 (X_3, Y_3, Z_3)
	Locator 4 (X_4, Y_4, Z_4)
	Locator 5 (X_5, Y_5, Z_5)
	Locator 6 (X_6, Y_6, Z_6)
	Position of clamps
	Clamp 1 (X_1, Y_1, Z_1)
	Clamp 2 (X_2, Y_2, Z_2)
	Clamping force magnitude
	Clamp 1 (Fcl_1)
	Clamp 2 (Fcl_2)
State variables	Vonmises effective stress (VONMISES)
Objective function	Maximum resultant displacement (DMAX)

Because there are a finite number of positions where the modular tooling can be fastened to the base plate, the optimisation 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 workpiece stability and locating accuracy the supports on the primary reference plane should be placed as far apart as possible. However, to minimise workpiece deformation, the supports should be placed as close to the loads normal to the primary surface as possible. The support locations are optimised where workpiece deflections are minimised and locating accuracy is highest. Locating accuracy, workpiece stability, and workpiece deformations are all affected by the support locations and contribute to the overall fixture stiffness and subsequently, the machining accuracy [3].

5 ANSYS optimisation study

A sample optimisation analysis shown in Fig. 7 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 optimisation analysis is used to minimise the maximum resultant displacement in the workpiece, by optimising support locations, clamp locations, and clamping force magnitudes. The same 3-2-1 fixture configuration used for the workpiece in the loading study, was used as the initial configuration in the optimisation analysis. The algorithm for selecting initial support locations is explicitly described in the loading study. Three feasible design sets resulted from the optimisation analysis. The results are listed in Table 6. Design set 1 is the initial fixture configuration. Design set 2 is the optimised 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 optimisation analysis resulting in design set 2 is shown in Fig. 7 as a dashed square. The design space for the optimisation 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 workpiece material, AISI 1212 Steel, $\sigma_y = 58\,015$ 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 workpiece 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, SEQV5, and SEQV6) remain relatively constant, since the axial thrust force magni-

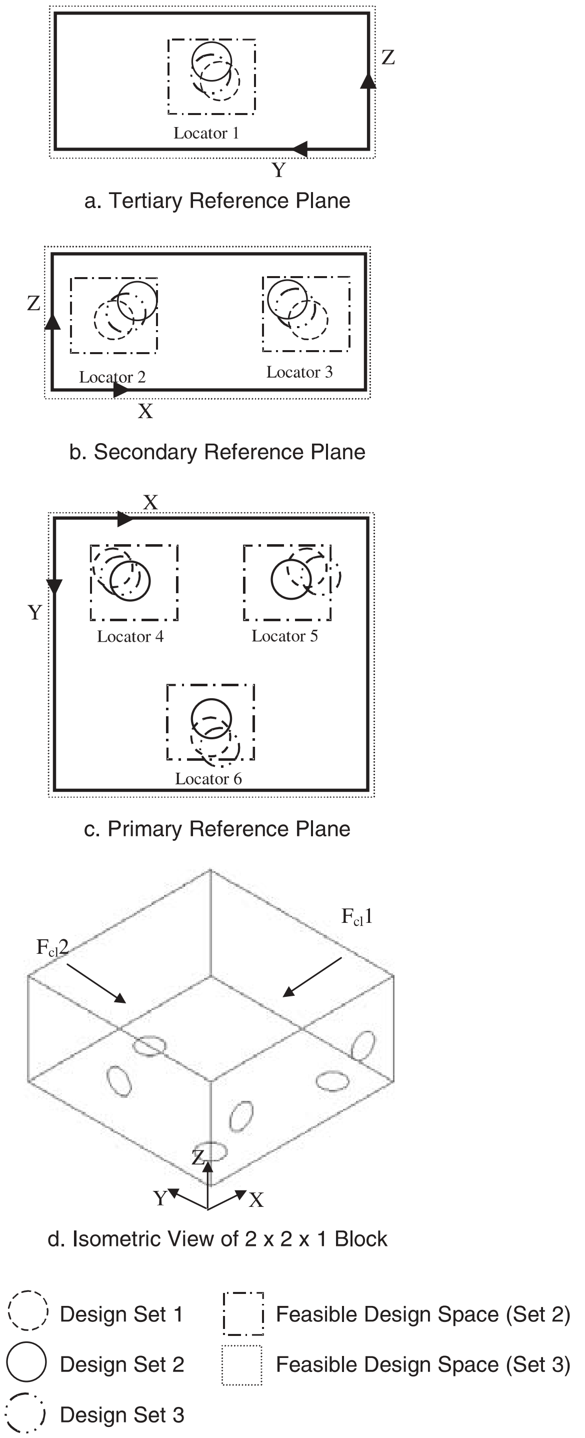


Fig. 7a–d. Benchmark fixture design configurations **a** Tertiary reference plane, **b** Secondary reference plane, **c** Primary reference plane, **d** Isometric view of $2 \times 2 \times 1$ block

tude 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 optimised 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.

Table 6. ANSYS problem optimization analysis results

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
SEQV5	SV	1.71×10^{8} Pa	1.43×10^{8} Pa	1.70×10^{8} Pa
SEQV6	SV	3.34×10^{8} Pa	3.25×10^{8} Pa	3.09×10^{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	0.0373 mm	0.0341 mm	0.0370 mm

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 minimise deflections due to the applied axial load. The locators on the secondary and tertiary reference planes were moved up to minimise deflections due to the applied torque.

It is obvious that without some knowledge base in fixture design, the optimisation 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 optimisation 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. It is more appropriate to declare the entire surface on each reference plane as feasible design space, but the analysis is more time intensive than if the design space is limited to a smaller range of values.

6 Industrial optimisation case study

An industrial case study was conducted to validate the fixture design analysis method developed in this study. The workpiece model is a simplified die cast aluminium brake caliper taken from Delphi Automotive Systems. The model is simplified to protect proprietary features and dimensions. The locators are placed in a 3-2-1 configuration. Three locators are placed on the primary reference plane (one on the bottom of the caliper and two directly below the slide bushing holes). Two locators are placed on the secondary reference plane, which is on the side of the caliper, and one locator is placed on the tertiary reference plane, directly behind 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 workpiece to the locator without generating any bending moments. Because the tertiary reference plane is perpendicular to the direction of applied loading, no clamp is necessary opposite the locator. A list of brake caliper model parameters and results is listed in Table 7. Table 8 lists the locator and clamp positions in mil-

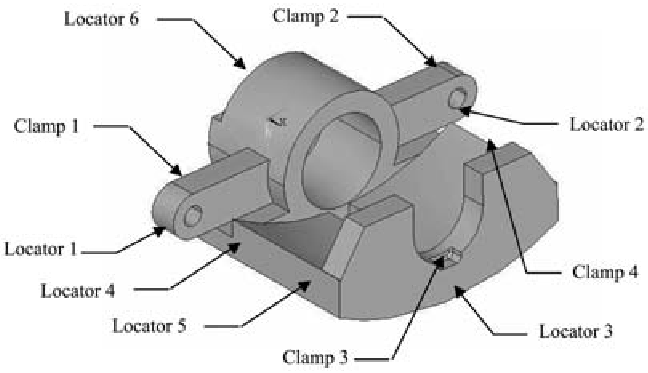


Fig. 8. Simplified brake caliper model

limetres relative to the global coordinate system origin. Delphi Automotive Systems provided the initial fixture configuration, clamping force magnitude, machining forces, and locator stiffness values. The locators were modelled with multiple ANSYS CONTAC52 spring-gap elements in parallel, attached to a circular contact area at specified fixturing points on the brake caliper. The loading is representative of a boring operation.

The maximum resultant displacement in the preloaded workpiece model is 0.0032 mm, and increases slightly to 0.0036 mm in the fully loaded workpiece 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 increases significantly, by as much as 100%, but does not exceed the maximum resultant displacement in the preloaded workpiece model. Figures 9 and 10 are the resultant displacement and von Mises stress plots, respectively, for the preloaded model (clamping loads, no machining

Table 7. Brake caliper model parameters and results

Element type	ANSYS SOLID45 4-node tetrahedral
Mesh type	Free tetrahedral
Workpiece material type	6061-T6 aluminium
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
Workpiece material yield strength, σ_y	1.7×10^8 Pa
Poisson's ratio, ν	0.35
Coefficient of static friction, μ	0.61
Thrust force, F_c	249.1 N
Torque, T	18 865 Nmm
SEQV1	7.67×10^5 Pa
SEQV2	5.95×10^{-5} Pa
SEQV3	7.40×10^5 Pa
SEQV4	1.31×10^5 Pa
SEQV5	2.66×10^5 Pa
SEQV6	4.11×10^5 Pa
Clamping force, FCL1	200 N
Clamping force, FCL2	200 N
Clamping force, FCL3	200 N
Clamping fForce, FCL4	200 N
DMAX	0.0036 mm

Table 8. Optimised brake caliper locator and clamp positions

	Initial configuration (mm)*			Optimized configuration (mm)		
Locator	X	Y	Z	X	Y	Z
1	37.95	17.00	-89.50	37.95	17.00	-89.50
2	37.95	17.00	89.50	37.95	17.00	89.50
3	133.85	48.00	0.00	133.85	48.00	0.00
4	78.42	17.51	-76.00	78.42	17.51	-76.00
5	126.84	17.51	-76.00	126.84	17.51	-76.00
6	0.00	0.00	0.00	0.00	0.00	0.00
Clamp	X	Y	Z	X	Y	Z
1	37.95	-10.00	-89.50	37.95	-10.00	-89.50
2	37.95	-10.00	89.50	37.95	-10.00	89.50
3	141.85	27.00	0.00	141.85	27.00	0.00
4	102.61	17.51	76.00	102.61	17.51	76.00

*Optimized experimentally

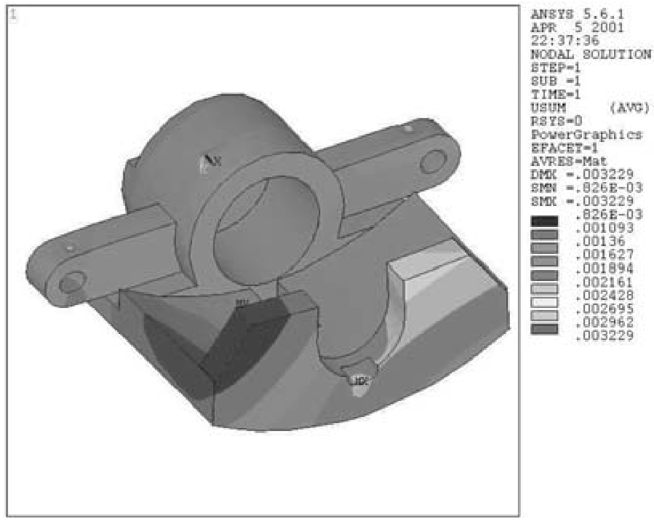


Fig. 9. Pre-loaded brake caliper resultant displacement (mm) contour plot

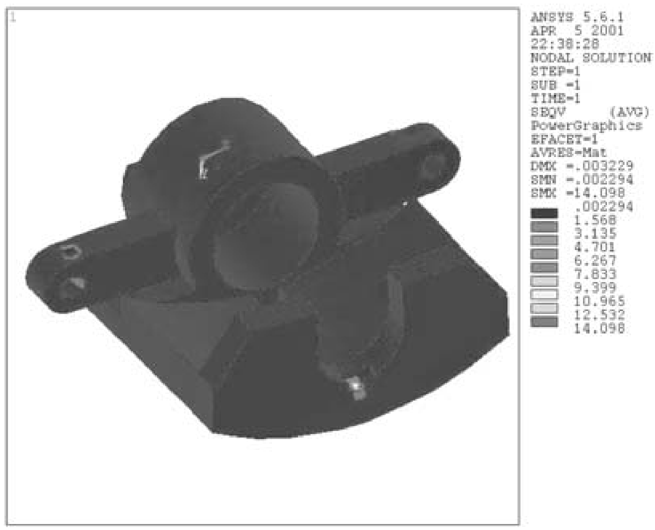


Fig. 10. Pre-loaded brake caliper von Mises stress (MPa) contour plot

loads). Figures 11 and 12 are the resultant displacement and von Mises stress plots, respectively, for the loaded model.

There is a stress concentration at the bottom of the cylinder bore, as shown in Fig. 12, during machining due to bending moments generated by the thrust force. The maximum von Mises stress occurs at the contact area of clamp 3, located opposite locator 3 on the primary reference plane.

An optimisation analysis was conducted to validate the optimisation tool developed in ANSYS. Because the fixture configuration for the caliper has been optimised experimentally, the desired result of the optimisation analysis is that ANSYS will produce the same fixture configuration. As expected, the support location optimisation resulted in the same fixture configuration. However, ANSYS further reduced the maximum resultant displacement in the workpiece by minimising the clamping force magnitude. The clamping force was reduced to 100 N, subsequently reducing the maximum resultant displacement by 31% to 0.0025 mm. The von Mises stresses at the supports, which are located directly opposite the clamps, were also reduced significantly as shown in Table 9. The von Mises stress at locator 6, SEQV6 remained the same, since locator 6 is not reacting to the clamping forces, but rather to the applied machining loads, which remained constant.

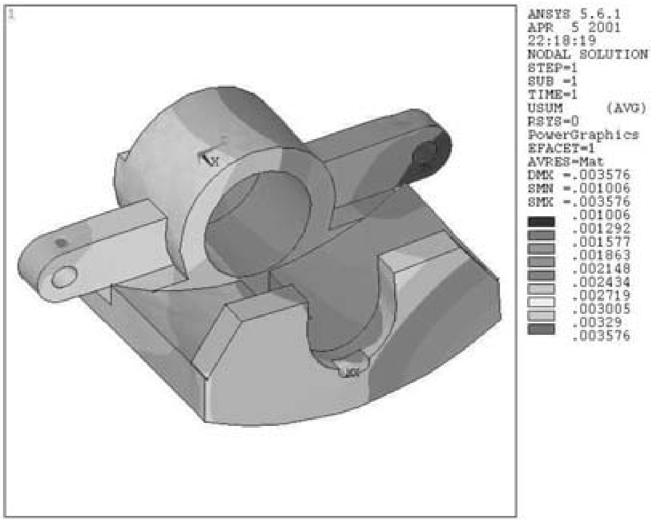


Fig. 11. Loaded brake caliper resultant displacement (mm) contour plot

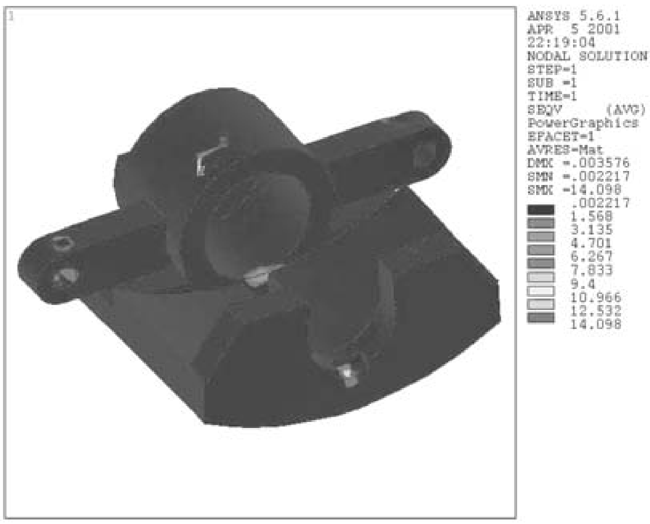


Fig. 12. Loaded brake caliper von Mises stress (MPa) contour plot

Table 9. Optimised brake caliper results

Optimization variable	Variable type	Initial configuration	Optimized configuration
SEQV1	SV	7.67×10^5 Pa	4.72×10^5 Pa
SEQV2	SV	5.95×10^5 Pa	1.98×10^5 Pa
SEQV3	SV	7.40×10^5 Pa	3.68×10^5 Pa
SEQV4	SV	1.31×10^5 Pa	0.68×10^5 Pa
SEQV5	SV	2.66×10^5 Pa	1.41×10^5 Pa
SEQV6	SV	4.11×10^5 Pa	4.11×10^5 Pa
FCL1	DV	200 N	100 N
FCL2	DV	200 N	100 N
FCL3	DV	200 N	100 N
FCL4	DV	200 N	100 N
DMAX	OBJ	0.0036 mm	0.0025 mm

7 Conclusions

In this study a finite element model was developed for fixtured workpiece boundary conditions and applied loads in machining using ANSYS 5.6.2. 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 realised based on the research conducted throughout this study:

Workpiece 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.

Locator model. It is appropriate to model locators with a single elastic point constraint for large workpiece surface area to locator contact area ratios. If the surface area to locator contact area ratio is small, the multiple spring-gap element model using ANSYS CONTAC52 elements must be used.

Load model. It is appropriate to model the cutting tool axial load with a single point load for large workpiece surface area to cutting tool contact area ratios. In addition to being easier to apply, the single point load is more conservative 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 workpiece deflection analysis. The torque component of the load model is critical to workpiece deformation.

Optimisation. In this study, a method of fixture design optimisation was developed. The method is valid for solid workpieces with planar locating surfaces and may be used to optimise support locations, clamp locations, and clamping force magnitude. The user must have some basic fixture design knowledge to define the initial fixture configuration and design space. A configuration, which is not feasible, will not be solved by ANSYS. The method is capable of minimising the maximum resultant displacement and assessing workpiece stability. If the workpiece is not stable, it will enter a state of rigid body motion and will not be solved by ANSYS.

This study focused on the minimisation of the maximum resultant displacement in the workpiece as a result of applied machining loads to demonstrate the capabilities of the modelling methods developed. However, the displacement results can be retrieved parametrically at any user-specified location in the workpiece, critical to the quality of the finished part. Although the displacements in the workpiece are elastic, the concern is local displacements occurring during a machining operation, which are critical to the accuracy of the machined feature. The total machining error, which should be within specified workpiece design tolerances, is the sum of the locating error of the fixture, the machine tool resolution, machine error, cutting tool deflection, fixture component deflection, and the workpiece deflections due to machining loads. Cutting tool resolution and machine error for milling centres can vary significantly depending on machine component quality.

Acknowledgement Special thanks to Steve Campoli, Dino Ciobannati, Dave Looman, Al Olsen, and Michael Stowe, of ANSYS, Inc. for their technical assistance throughout this research.

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 JD, Haynes LS (1987) Finite element analysis of flexible fixture system. Trans ASME J Eng Ind 109(2):134–139
3. Pong PC, Barton RR, Cohen PH (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 (1991) Optimization methods applied to selecting support positions in fixture design. Trans ASME J Eng Ind 113:412–418
5. Trappey AJC, Su CS, Hou JL (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 SJ, Yuan JX (1996) Deformable sheet metal fixturing: principles, algorithms, and simulations. Trans ASME J Eng Ind 118:318–324
7. Kashyap S, DeVries WR (1999) Finite element analysis and optimization in fixture design. Struct Optim 18(2–3):193–201
8. Bradly Associates (2001) GINO Graphics. Bradly Associates, Crowthorne, Berkshire, http://www.bradassoc.co.uk/home.htm
9. Hillstrom KE, Minkoff M (2001) VMCON. Applied Mathematics Division, Argonne National Laboratory, Argonne, IL, http://www.nea.fr/abs/html/nesc0922.html *and* Crane RL, RCA David Sarnoff Research Center, Princeton, NJ
10. ANSYS, Inc. (2001) Alternate online manual set. Texas A&M University Supercomputing Facility, College Station, TX, http://sc.tamu.edu/softwareDocs/ANSYS/realtoc.html
11. Carr Lane Manufacturing Company (2001) Online catalog. Carr Lane Manufacturing Company, St. Louis, MO, http://www.carrlane.com/