International Journal of Applied Research in Mechanical Engineering

Volume 2 | Issue 3 Article 11

January 2013

FINITE ELEMENT MODELLING AND ANALYSIS OF WORKPIECE-FIXTURE SYSTEM

N. M. KUMBHAR
Dept. of Mechanical Engineering, Govt. College of Engineering, Karad, Dist- Satara, Maharashtra, India,
nilkumbhar444@gmail.com

G. S. PATIL
Dept. of Mechanical Engineering, Govt. College of Engineering, Karad, Dist- Satara, Maharashtra, India,
gaurav.sp06@gmail.com

Sushas S. Mohite
Mechanical Department, Government College of Engg, Karad, Dist. Karad, India - 415124,
mohitess@yahoo.com

M. A. SUTAR
Dept. of Mechanical Engineering, Govt. College of Engineering, Karad, Dist- Satara, Maharashtra, India,
maheshsutar.a@rediffmail.com

Recommended Citation

KUMBHAR, N. M.; PATIL, G. S.; Mohite, Sushas S.; and SUTAR, M. A. (2013) "FINITE ELEMENT MODELLING AND ANALYSIS OF WORKPIECE-FIXTURE SYSTEM," *International Journal of Applied Research in Mechanical Engineering*: Vol. 2 : Iss. 3 , Article 11.
Available at: https://www.interscience.in/ijarme/vol2/iss3/11

FINITE ELEMENT MODELLING AND ANALYSIS OF WORKPIECE-FIXTURE SYSTEM

N. M. KUMBHAR, G. S. PATIL, S. S. MOHITE & M. A. SUTAR

Dept. of Mechanical Engineering, Govt. College of Engineering,
Karad, Dist- Satara, Maharashtra, India
E-mail: nilkumbhar444@gmail.com, mohitess@yahoo.com,
gaurav.sp06@gmail.com, maheshsutar.a@rediffmail.com

Abstract - The fixture design and analysis lead-time is always a major influencing factor for the bottleneck between product design and manufacturing. With advent analysis and simulation tools, there is a lot of potential to expand the scope of optimization of fixture elements to achieve the objectives such as ‘maximizing rigidity’ and ‘minimizing weight’ of a fixture assembly. There are many sources which affect the rigidity of fixture assembly. The rigidity of fixture assembly mainly depends upon various factors such as material properties, geometry of individual components, stability against thermal distortions, clearances or plays on fixture elements (to facilitate loading and unloading of workpiece), contact deformations at contacting surfaces, etc. Among these, contact deformations at contact interface are one of the important sources of errors in precision fixture design. These contact deformations are measure of contact stiffness and therefore needed to be studied in accordance with workpiece-fixture contact. In this paper, a numerical model of fixture unit assembly is presented using the FEA tool ANSYS. ANSYS contact elements CONTA175 (8 node element) and TARGE170 (8 node quadrilateral element) are utilized to represent contact stiffness at contact interface. This model simulates the behavior of contacting surfaces of fixture assembly for different values of surface roughness, coefficient of friction (μ) and contact stiffness with very little modelling or computational efforts and time. This work establishes the FEA model of fixture unit assembly to predict fixture unit stiffness and FEA model of rough surface contact to predict contact stiffness. Based on this study, the database of fixture stiffness and contact stiffness is built up and further can be used in computer aided fixture design (CAFixD).

Keywords *- fixture unit stiffness, contact stiffness, surface roughness, contact deformation.*

I. INTRODUCTION

Fixtures are an assemblage of fixture-baseplate, locators, clamps, supports and components which are in contact with one another and subjected to static preloads and dynamic forces of machining. The rigidity of fixture assembly mainly depends upon various factors such as material properties, surface properties, size and shape of individual components, stability against thermal distortions, clearances or plays on fixture elements, contact deformations at contacting surfaces, etc. Due to availability and use of computational tools for fixture design such as expert systems; fixture design activity leads towards optimization of fixture components. This optimization is to be done with an objective of ‘maximizing rigidity’ against external loads and ‘minimizing weight’ of fixture assembly. In order to study fixture stiffness, a general fixture assembly structure is decomposed into functional units with fixture components called as fixture units. Fixture unit stiffness is therefore defined as the force required for a unit deformation of the fixture unit in normal and tangential directions at the contact position with another unit. When a workpiece is located and clamped in the fixture, the fixture units subjected to external loads, which transmitted from the workpiece. If the external load acting on a fixture unit is known, and the displacement of the fixture unit is measured or calculated, the fixture unit stiffness can be estimated. Fig. 1 shows the definition of the fixture unit stiffness K_{Un} in the normal direction. Once the stiffness of fixture units are known, the overall fixture stiffness can be obtained regarding the tolerance sensitivity [2].

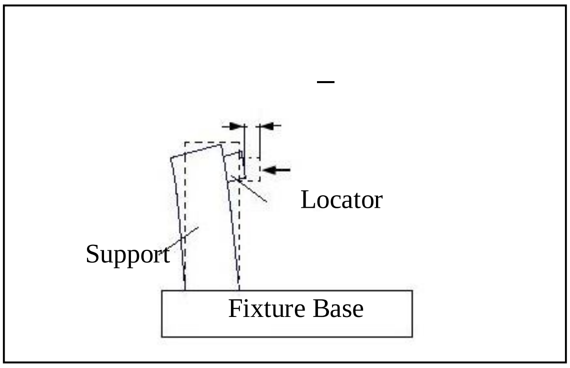

Fig.1 : Definition of fixture unit stiffness

II. LITERATURE REVIEW

While studying the fixture–workpiece systems, prime focus is on analyzing workpiece- fixture contact interactions. Contact problems are traditionally classified in terms of surface friction, initial undeformed geometry, initial conditions, relative rigidity and behavior under loading.

Liao and Hu [1] developed an integrated model of fixture-workpiece system. The commercial FEA codes ABAQUS and NASTRAN along with DMAP (Direct Matrix Application Program) and FORTRAN programs were used to simulate an engine block subjected to fixture clamping and a face milling operation. The developed simulation procedure was used to determine the effects of clamping preloads, machining forces and forced vibrations of the fixture–workpiece system on the machined surface flatness. Kang et al. [2] presented the methodologies of fixturing stability analysis in CAFixDV (Computer Aided Fixture Design and Verification). A kinetic model of workpiece-fixture system was created to formulate the stability problem to calculate the minimum clamping forces required in a machining operation and the effect of the clamping sequence on fixturing stability. J. Asante [3] computed and investigated the effect of fixture compliance and cutting conditions on workpiece stability and used it as a basis for selecting a suitable fixture among several alternatives using analytical approach. Cioata and Kiss [4] presented the simplified analytical model of contact deformation between locators and workpiece and a finite element model in order to estimate the contact deformation at workpiece-locator contact. Zheng [5] developed an FEA model of fixture unit stiffness with contact elements for solving contact problems in workpiece-fixture assembly.

From above literature review following remarks can be drawn.

- Despite of extensive work in field of mathematics and engineering, frictional rough surface contact remains one of the most challenging problems due to nonlinear formulations and solution procedures.
- Most of the researchers assumed fixture elements as rigid and few of them considered as elastic at local contact points only.

I. *FEA Model of Fixture Unit Assembly*

The main objective of this work is to develop the fixture unit stiffness database which to be used in Computer Aided Fixture Design (CAFixD). To achieve this objective a typical fixture unit assembly is modeled in ANSYS. The FEA model of typical fixture unit assembly is developed in ANSYS.

A.FEA Model Formulation

Consider a general fixture unit with two components I and J, as shown in Figure 2. For multi-component fixture units, the model can be expanded. The fixture unit is discretized into finite element models using a standard procedure, except for the contact surfaces, where each nodes on the finite element mesh for the contact surface is modeled by a pair of nodes at the same location belonging to components I and J, respectively, which are connected by a set of contact elements (CONTAC175 and TARGE170). The basic assumptions include that material is homogenous and linearly elastic, displacements and strains are small in both components I and J, and the frictional force acting on the contact surface follows the Coulomb’s law of friction.

When the two components I and J are in contact with each other, a number of three-dimensional contact elements are in effect on the contact surfaces. It should note that the problem is strongly nonlinear, partially due to the fact that the number of contact elements may vary with the change of contact condition. The original contacting nodes might separate or recontact after separation, based on the deformation condition on the contact surface; also contact stiffness may not constant either. The contact elements are capable of supporting a compressive load in the normal direction and tangential forces in the tangential directions. While the two components are in contact and the displacements in the tangential and normal direction are assumed as independent, the element itself can be treated as three independent contact springs: two having stiffness k_t and k_t in the tangential directions of the contact surface at the contact point and one having stiffness k_n in the normal direction as shown in Fig. 2.

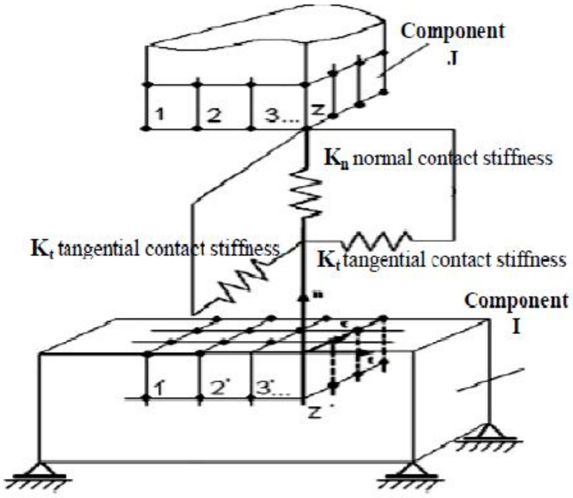

Fig. 2 Contact model of two fixture unit assembly

B.FEA Model description

Fig. 3 shows typical fixture unit developed for the static, structural (Large deflection) finite element analysis. For the sake of simplicity of model formulation, the contact stiffness in all three directions is assumed to be same. The ANSYS software package is used to solve the defined problem. The model with two deformable components (500 x 500 x 100 mm as fixture base and 100 x 100 x 300 mm as support) which simulates fixture body and fixture element such as support or locator of machining fixture respectively.

The bottom of the fixture base is fixed. The evenly distributed load, F_C is applied to the nodes on the top of support, simulating the fastening force in the fixture. A concentrated load F_M, parallel with the fixture base, is applied to the node on the top of the support, in simulating the external machining force passed through the workpiece being fixtured. The fixture unit deflection is measured as δ_{uf} (at the position of x = 150mm, y = 300mm, and z = 400mm) in z direction at the top of the support, as shown in Fig. 4. The material properties are as shown in Table 1.

TABLE 1 Material Properties for FEA Model of Fixture Unit Assembly

Material	AISI 4150
Modulus of elasticity (E)	200 GPa
Poisson's ratio (ν)	0.3

The results of developed FEA model of fixture unit assembly are discussed in following sections.

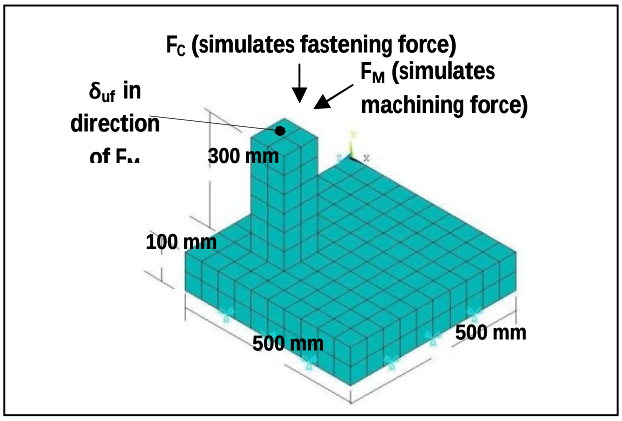

Fig. 3 FEA model of fixture unit assembly

C. FEA Model of Rough Surface Contact

The FEA model with rough surface contact is generated using standard formulation procedure for probabilistic random rough surface available from literature [6] in ANSYS environment. The material properties are same as listed in Table 1. Three values of surface roughness are taken for analysis viz., 1.44, 3.08, 5.55 μm. The results of FEA model of rough surface contact are discussed in next section.

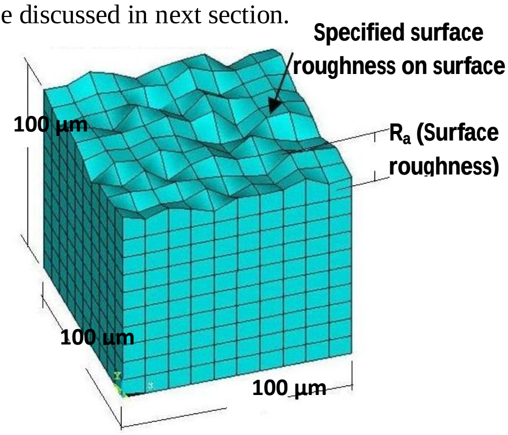

Fig. 4 Random probabilistic rough surface generated in ANSYS

I. Results Analysis

Finite element modelling of typical fixture unit assembly is carried out using a FEA tool, ANSYS. In static analysis of fixture unit model; first, effects of external machining forces (F_M and F_C), coefficient of friction (μ) and contact stiffness (K_c) on fixture unit deformation (δ_{uf}) are studied. And later on fixture unit stiffness (K_{uf}) is calculated as defined, load per unit fixture deformation in tolerance sensitive direction (in direction of applied external force).

A.Analysis of FEA Model of Fixture Unit Assembly

- *Effect of External Machining Force (F_M) on Fixture Unit Deformation (δ_{uf}):* A typical curve of fixture unit deformation (δ_{uf}) of FEA model against the external machining force (F_M) is shown in Fig. 5. The curve can be divided into three stages: the linear first stage (I), the second nonlinear stage (II), and the third linear stage (III). In the first stage, for a small machining force F_M, the deflection of the fixture components contributes to elastic deformation of fixture unit. When external load increases, support begins to separate from the fixture base; which causes a decrease of actual contact area and a rapid increase in fixture unit deformation (δ_{uf}). When the external force (F_M) exceeds; separation of contact interface becomes stabilized and the deformation tends to be linear again in the third stage. Fig. 6 summarizes the deflection curves of the typical fixture unit assembly under different fastening force (F_C).

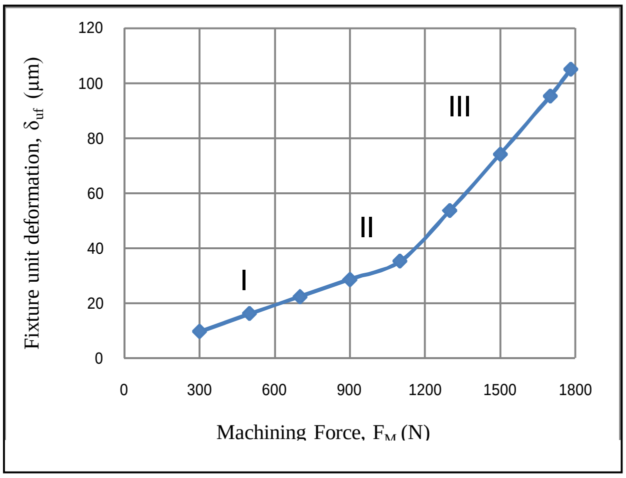

Fig.5 Typical deflection curve for fixture unit FEA model.

- *Effect of coefficient of friction (µ) on fixture unit deformation (δuf):* To analyse the effect of an input parameter, coefficient of friction (µ) on fixture unit deformation (δ_{uf}); three values of coefficient of friction (µ) are input to FEA model, viz., 0.1, 0.2, 0.3. The graph is as shown in Fig. 7. For all load cases, an increase in coefficient of friction (µ) between contacting surfaces decreases the fixture unit deformation (δ_{uf}). This is due to the fact that, as coefficient of friction increases, the frictional force increase; which opposes the external machining force (F_M). Therefore, the fixture unit deformation (δ_{uf}) decreases with increase in coefficient of friction (µ).

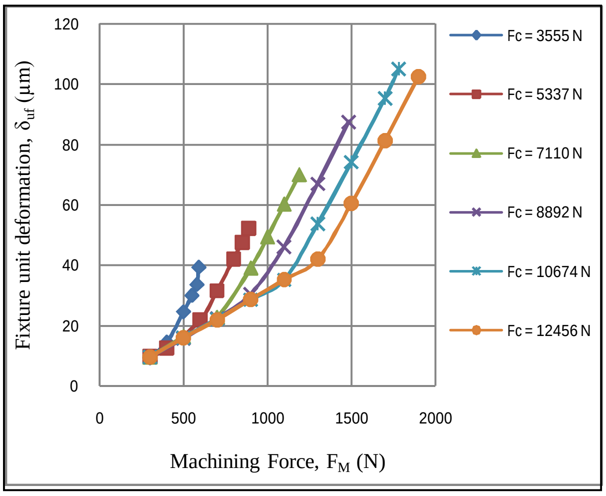

Fig.6 Deflection curve under different fastening force.

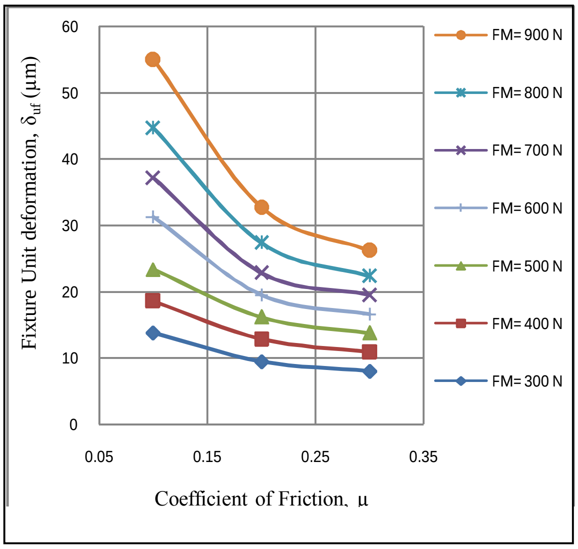

Fig.7 Effect of coefficient of friction (µ) on fixture unit deformation (δ_{uf}).

- *Effect of contact stiffness (K_c) on fixture unit deformation (δ_{uf}):* The FEA model of fixture unit assembly has contact elements CONTA175 (8 node quadrilateral) and TARGE170 (8 node quadrilateral) at contact interface. These elements require contact stiffness (K_c) as an input to the FEA model. To study the influence of contact elements and contact stiffness; model with contact elements at contact interface is compared with an integrated body without contact elements. The results are graphed in Fig. 8. As if contact stiffness increases by 10 %, the fixture unit deformation (δ_{uf}) decreases exponentially about 82 %. This is due to the fact that, as contact stiffness (Kc) increases, the contact interface becomes more and more rigid. At large value of contact stiffness (K_c) (about 1000 kN/mm); the structure behaves like single continuous structure.

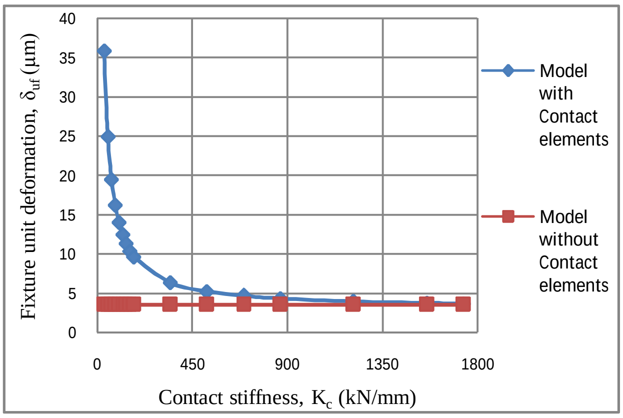

Fig.8 Effect of contact stiffness (K_c) on fixture unit deformation (δ_{uf}).

B. Rough surface analysis results

For analysis of effect of surface roughness (R_a) three values of surface roughness are taken into consideration, viz., 1.44, 3.08, 5.55 µm. These values are chosen from literature articles; which are found to be realistic in practical situations.

- *Effect of surface roughness (R_a) on contact deformation (δ_c) and contact stiffness (K_c):* Effect of surface roughness (R_a) on contact deformation (δ_c) is shown in Fig. 9. This nature of graph is due to the fact that as surface roughness increases, fewer asperities will come into contact. In this situation, due to the existence of surface irregularities the real area of contact is very tiny fraction of the nominal or apparent contact area. Therefore, with increase in surface roughness; the real area of contact reduces drastically; and thus contact deformation increases linearly. Similar trend was observed for all other value of surface roughness. The graph shows linear increase in contact deformation with normal load. The contact stiffness (K_c) is therefore estimated from contact deformation as load (F_N) per unit contact deformation (δ_{uf}). Effect of surface roughness on contact stiffness is graphed in Fig. 10.

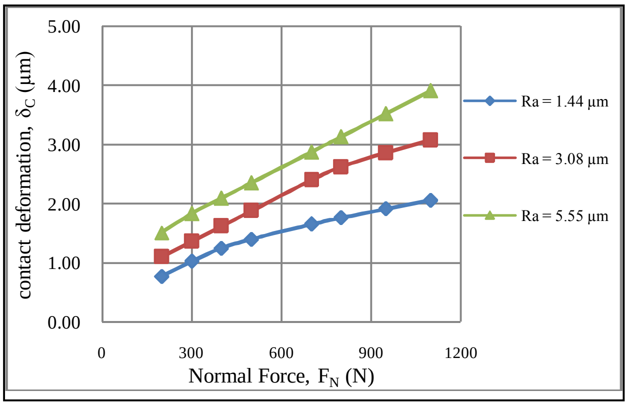

Fig.9 Effect of surface roughness (R_a) on contact deflection (δ_c).

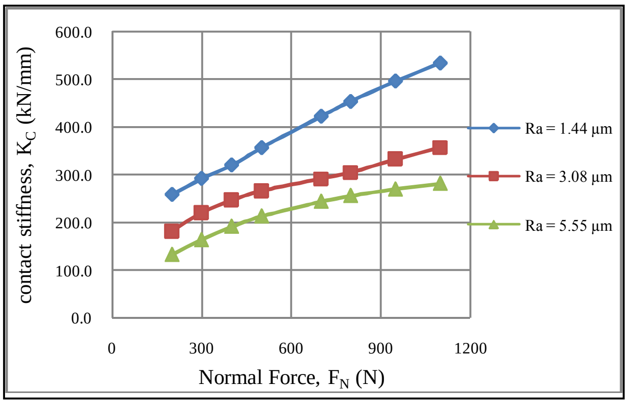

Fig.10 Effect of surface roughness (R_a) on contact stiffness (K_c).

II. *Validation of FEA Results*

The FEA model of fixture unit assembly and rough surface contact is validated using experimental results from literature studied.

A. Validation of FEA model of fixture unit assembly

An experimental work by Zhu [7] was done on modular T-slot modular fixtures. Fig. 12 Shows the experimental configuration; a basic assembly unit, where structural supports are bolted to a baseplate. When external forces (F_M and F_C) are exerted on the upper portion of the supports in the horizontal and vertical directions, the fixture component deformation is measured in the horizontal direction. The experiment showed that, as the exerted external force increases; fixture deformation also increases in manner as shown in Fig. 13. From results, it is found that FEA results show similar trend as that of the experimental results. The difference between experimental and FEA results is caused by the simplification of the FEA model of fixture unit assembly. The major assumptions are:

i. Material is homogeneous and linearly elastic.

ii. The friction force acting at contact interface follows Coulomb law of friction.

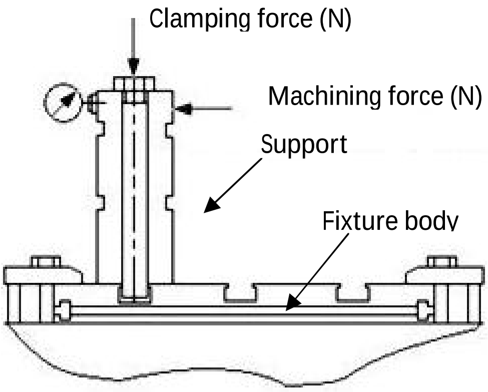

Fig.11 Experimental setup developed by Zhu [7].

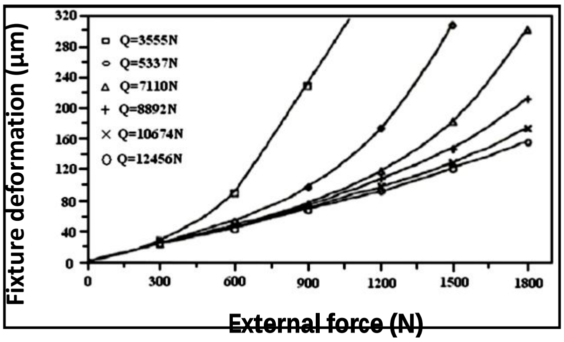

Fig.12 Deflection curves from experimental work by Zhu [7].

B. Validation of FEA model of rough surface contact

Zheng [5] carried out an experimental research of measurement of contact stiffness of two rough surface contacts. The results of experimental work are shown in Fig. 14. The trend is similar to that of results of FEA model of rough surface contact. The difference in the results of FEA model and experimental model are due to;

1. The simulated surface roughness of FEA model does not represent actual surface roughness of specimen.
2. The effects of environmental factors such as vibrations of test bed structures, dust particles at contact interface, etc. are not accounted in FEA model.
3. The contact area of FEA model is different than that of experimental one.

Thus, the developed FEA model of fixture unit assembly is validated using experimental results and can be used for analysis with confidence.

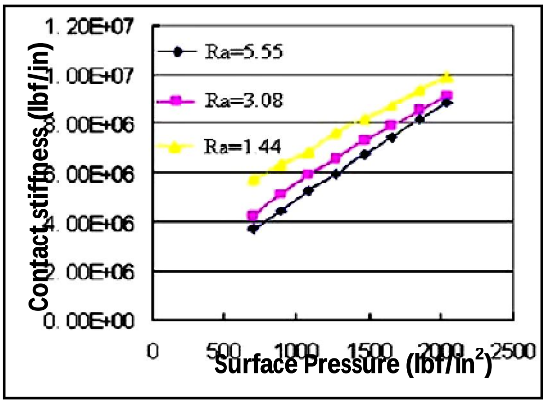

Fig.13 Contact stiffness results from experimental work [5].

CONCLUSIONS

A FEA model simulates the behaviour of contacting surfaces of fixture unit assembly for different input values of loads (F_M and F_C), coefficient of friction (μ) and contact stiffness (K_c). Initially, FEA model of fixture unit assembly is simulated and result; i.e., fixture unit deformation (δ_{uf}) studied.

- Fixture unit deformation (δ_{uf}) of FEA model with coefficient of friction, μ=0.2 (δ_{uf} =22.88 μm) reduces as compared to model with coefficient of friction, μ=0.1 (δ_{uf} =37.183 μm) by 38%. Similar trend was observed for the model with coefficient of friction μ=0.3, (δ_{uf} =19.52 μm). Therefore, larger coefficient of friction offers higher fixture unit stiffness and thus better structural stability against external loads.
- In analysis of rough surface contact model, FEA model with surface roughness, R_a =3.08 μm, (δ_c=2.39 μm) contact deformation (δ_c) increased by about 31 % as compared to that of model with surface roughness, R_a =1.44 μm, (δ_c=1.65 μm). Similar nature was observed in model with surface roughness, R_a = 5.55 μm, (δ_c=2.86 μm).
- The contact stiffness (K_c) in FEA model with surface roughness, R_a=3.08 μm, (291.93 kN/mm) is decreased by 30.98 % as compared to that of model with surface roughness, R_a=1.44 μm, (423.01 kN/mm). The similar nature was observed in model with surface roughness, R_a=5.55 μm, (244.07 kN/mm).
- The results of developed FEA models are validated using previous experimental results. Thus, models are validated.

In concluding lines, this proposed FEA model of fixture unit assembly develops fixture unit stiffness database for integrated Computer Aided Fixture Design (CAFixD). This database is to be used to design a machining fixture with integrated computer aided fixture design system.

REFERENCES

[1] Y. G. Liao and S. J. Hu, “An Integrated Model of a Fixture–Workpiece System for Surface Quality Prediction”, Int. J. Adv. Manuf. Technol., Vol. 17, 2001, pp 810–818.

[2] Y. Kang, Y. Rong and J. C. Yang, “Computer-Aided Fixture Design Verification. Part 3. Stability Analysis”, Int. J. Adv. Manuf. Technol., Vol. 21, 2003, pp 842–849.

[3] J. N. Asante, “Effect of Fixture Compliance and Cutting Conditions on Workpiece Stability”, Int. J. Adv. Manuf. Technol. 48 (2010) 33–43.

[4] V. G. Cioata, I. kiss, “The machining error due to contact deformation of workpiece-fixture system”, ACTA Technical Bulletin of Engineering, 2009, pp 33-36.

[5] Y. Zheng, “Finite Element Analysis for Fixture Stiffness,” Worcester Polytechnic Institute, Worcester, MA, USA, PhD. Dissertation Rep., 2005.

[6] M. K. Thompson, “A Multi-scale iterative Approach for Finite Element Modelling of Thermal Contact Resistance,” Massachusetts Institute of Technology, PhD. Dissertation Rep., 2007, 18-28.

[7] Zhu, Y., S. Zhang and Y. Rong, "Experimental study on fixturing stiffness of T-slot based modular fixtures", NAMRI Transactions XXI, NAMRC, Stillwater, OK, 1993, 231-235.

◆ ◆ ◆