International Journal of Innovative Technology and Exploring Engineering (IJITEE)
ISSN: 2278-3075 (Online), Volume-8 Issue-12, October 2019

The Mechanics and Snap in Force Estimation of Cantilever Snap Fit Joint of Lock Plate by FEA

Hrishikesh S Kakade, FB Sayyad, VG Patil

Abstract: ***Snap fit joints are widely used in industry for assembling different parts. Nowadays snap fits are found in wide range of application ranging from household appliances, toys, telephones to automobiles. This study is carried out to find the most accurate snap in force of cantilever snap fit by using finite element analysis. Snap in force is the most important term in design of cantilever snap fit because while in manual assembly process it is very much important to perceive the snap fit engagement. This study focuses on the nonlinear FE analysis method to calculate insertion force of the plastic snap fit part. Behavior of plastic material is nonlinear so it difficult and time consuming to calculate the snap in force by mathematical formulae only so nonlinear FE analysis will be best method to get the realistic snap in force value. This research work also included the mathematical model and linear FE analysis calculations for the estimation of snap in force. In nonlinear analysis contact, geometric and material nonlinearity is defined in the FE model. For the verification of the results the physical testing is also carried out with the help of force gauge tool. The analytical calculations and linear FE analysis estimates higher snap in force than the actual, while nonlinear FE analysis estimates more accurate snap in force value.***

Keywords: Snap Fit, Snap in force, cantilever snap, Lock plate, Plastics, Polypropylene, FEA, material non linearity, ANSYS.

I. INTRODUCTION

This document investigates the tactile feedback of cantilever snap fit joints by using nonlinear finite element analysis. Lock plate is one of the most important parts of door assembly in dish-washer. Function of the lock plate is to lock the door on its place and do not allow to open once get locked. Snap fit feature has been provided on the lock plate which will make the assembly process of lock plate fast, easy and cost effective.

Design of snap fit feature is most important task which makes the assembly and disassembly process feasible & effortless. Cantilever hook type snap feature is commonly used in snap fit joints. Strength, constraint, compatibility, and robustness have been identified as the key requirements of snap-fit design.

Revised Manuscript Received on October 30, 2019.

***** Correspondence Author

Hrishikesh S kakade*, Design Engineering, Dr. D Y Patil School of engineering, Pune, India. Email: kakadehrishi@gmail.com

Dr. F B Sayyad, Mechanical Engineering, Dr. D Y Patil School of engineering, Pune, India. Email: farooksayyad@dypic.in

Asst.Prof. V G Patil, Mechanical Engineering, Dr. D Y Patil School of engineering, Pune, India. Email: vinodpatil@dypic.in

Robustness refers to the tolerance of dimensional variation. It is unpredictable because dimensions vary according to fabrication and assembly conditions. The most critical dimension variation is the interference between snap-fit hook and mating part. Although very small, the amount of interference determines the insertion force and hence the quality of the snap-fit. Since snap-fit connectors behave like cantilevers, the insertion force increases with increasing interference. The failure of parts is more costly. Like many contact-aided mechanisms, a successful design of snap-fit connectors requires accurate calculation of interference-induced insertion force.

Nonlinear finite element analysis estimates the more accurate value of snap in force. Situations in which contact occurs are common to many different snap-fit applications. Cantilever hook there actually exist several contact pairs depending on contact locations. Since the deformation of local contact areas it's very difficult to identify the point when these contact pairs change. But the frictional contact can exactly capture the physics present between mating parts of cantilever snap. Large deformations option in ANSYS includes the geometric non linearity which will include the actual trajectory of deflection in the simulation. It is very much important to define an actual stiffness matrix in FEA. The material of snap fit is polypropylene 30 percent glass filled which is highly nonlinear. The nonlinear material model takes care of actual stiffness of model.

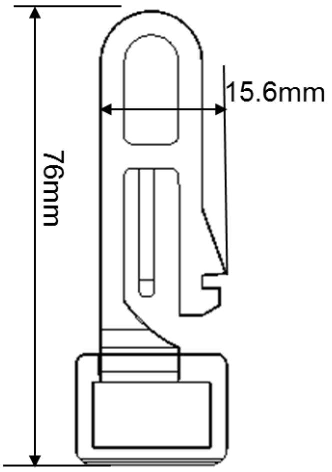

Fig. 1.Cantilever snap fit on lock plate model

II. METHODOLOGY

The main object of this project work is to find out the snap in force of cantilever snap joint which will resemble with the actual force observed at assembly line by the operator.

The mathematical calculations are carried out by using the emperical formulae. Finite element model is built in ANSYS workbench. Fine mesh with higher order element is generated to achieve the mesh convergence. Geometric ,contact and material non linearity is defined the FE model. Frictional contact has been defined with appropriate coefficient of friction value.

Non linear maerial model of PP30GF is defined so that the stiffness matrix will give us the more realistic force value. Elasto plasic non linear material model is used in ANSYS workbench.

The confirmation test of the snap fit feature of lock plate is caried out and it is ensured that the installation force calculated by nonlinear FE analysis is very close to physical test value.

III. PROBLEM STATEMENT

Snap fit feature has been provided on lock plate to make the connection with collar and to make the assembly process easy, cost effective & fast. Analytical calculations and linear finite element analysis estimates the approximate value of snap in force of snap joint.

So aim of this paper is to find out the snap in force of cantilever snap fit which will be most realistic by using the nonlinear finite element analysis.

IV. NUMERICAL FORMULATION

The type of the snap fit in lock plate in cantilever type. It is suggested to design the finger so that either its thickness (h) or width (b) tapers from the root to the hook; in this way the load-bearing cross section at any point bears a more appropriate relation to the local load. The maximum strain on the material can therefore be reduced, and less material is needed.

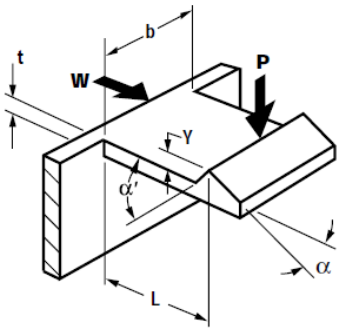

Fig. 2.Cantilever snap fit

If the snap fit has to insert without any permanent deflection (plastic deformation), the retention or pull-off force must be below a maximum value, the elastic strain limit, but high enough to retain engagement under normal service load. The insertion or assembly force is defined as,

$$W = P * \frac{\mu + tan\alpha}{1 - \mu * tan\alpha}$$

And

$$P = \frac{bt^2E\epsilon}{6L * Q}$$

Where,

W=Assembly or insertion force

P=Perpendicular force

u=coefficient of friction

α=lead angle

b=beam width

t=Beam thickness

L=Beam length

E=Flexural modulus

e=Strain at base

Table- I: Calculation for snap in force

Sr no	Parameter	Value	unit
1	beam width	4.5212	mm
2	beam thickness	4.699	mm
3	beam length	27.5	mm
4	deflection	2.159	mm
5	lead angle	15.6	degree
6	coefficient of friction	0.3	--
7	flexural modulus	10500	Mpa
8	Deflection magnifying factor	3	--
9	Perpendicular force	133	N
output	**Snap in force**	**44.2**	**N**

V. FINITE ELEMENT ANALYSIS (FEA)

This chapter describes the finite element model of cantilever hook with a high retention angle. Firstly three different nonlinearities of finite element analysis were described: Moving Contact, Geometric Nonlinearities and Material Nonlinearities. Then finite element analysis techniques applied in this thesis were explained because in a typical FEA simulation usually 80%-90% time is spent to find the best parameter settings. Finally a three dimensional, contact FEA model was created in ANSYS workbench, and the results were compared to the physical testing results with the same geometry to verify the accuracy of the FEA model.

Material Properties

A number of material-related factors can cause your structure's stiffness to change during the course of an analysis. Nonlinear stress-strain relationships of plastic, multilinear elastic, and hyper elastic materials will cause a structure's stiffness to change at different load levels. All materials are assumed to be homogeneous and isotropic. Polypropylene 30% Glass filled (PP30%GF) material has been used for lock plate and steel material has been used for collar. For the purpose of numerical modelling, it was assumed that the material was time independent. For all design guides of snap-fits none of these material nonlinearities were included in their design guides. All of them assumed that the maximum stress always under the yield stress that means no plastic deformation occurred during the retention. In order to overcome the inaccuracy caused by the ignoring of plastic deformation, it's necessary to include the material nonlinearity in the FEA model. Material of lock plate is plastic (PP30%GF) so to ensure the realistic resuts from FEA, multi-linear isoropic properties has been defined for PP30%GF. As collar is not area of interest to reduce solver time, linear proprties has been defined for steel material.

Table- II: Material Properties

Part	Material	Young's Modulus (MPa)	Tensile Strength (MPa)	Strain at break %
Lock Plate	PP 30%GF	10500	140	2.5
Collar	Steel	210000	250	0.45

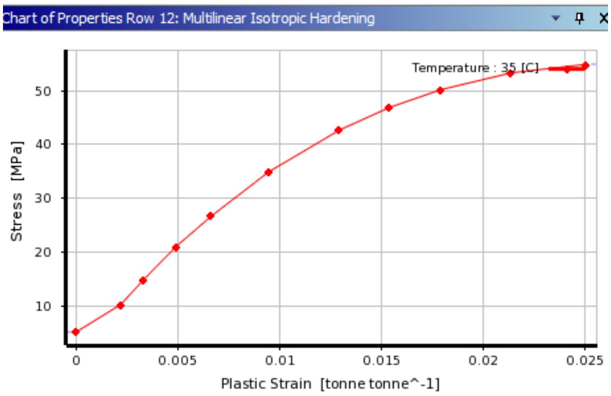

Fig. 3.Multi-linear isotropic stress-strain curve PP30GF

Cotact non linearity:

Many common structural features exhibit nonlinear behaviour that is status dependent. Status changes might be directly related to load, or they might be determined by some external cause. Situations in which contact occurs are common to many different snap-fit applications.

During the retention of cantilever hook there actually exist several contact pairs depending on contact locations. Since the deformation of local contact areas it's very difficult to identify the point when these contact pairs change.It is very much important to include the coefficient of friction in FE model. Also the stiffness of the contact pair should be enugh so that very negligible penetration will occure between mating surfaces and it chattering will also avoided.

FE model information:

For finite element modelling second order tetra elements are used. Typical solid 187 elements are selected from ANSYS element library. Fine mesh is used in the simulation to achieve the mesh convergence and more accurate results will be obtained. Total node count of the model is 620602 and number of elements in the model are 420423.

SOLID187 element is a higher order 3-D, 10-node element. SOLID187 has quadratic displacement behaviour and is well suited to modelling irregular meshes (such as those produced from various CAD/CAM systems). The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyperelasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

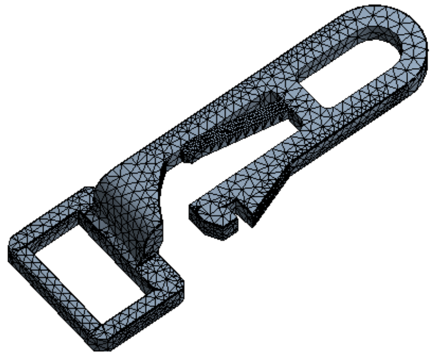

Fig. 4.FE model of lock plate

Loads and Boundary conditions

Collar is fixed at its edges in all DOF. Displacement in axial direction is applied at the end of lock plate so that it will get inserted into the collar.The reaction force is measured at the displacement location which is the required snap in force.Following figure shows the loading and boundary conditions for lock plate.

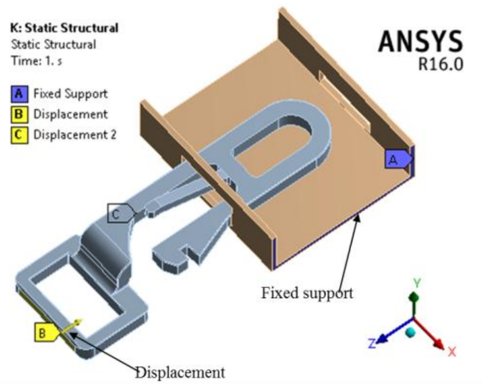

Fig. 5.Loads and Boundary conditions

Analysis results of linear model

FE model is solved by considering the linear material model of polypropylene 30% glass filled. The modulos of elesticity considered is constant through out the analysis. Non linear frictional contact has been defined in the model. Following are the results obtained from the linear material model of cantilever snap fit.

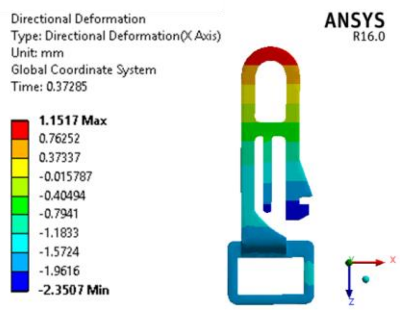

Fig. 6.X deflection Plot

As shown in the above image the deflection obverd on snap in x direction is 2.3mm.

Following figures shows the von mises stress plot. The stress occured on lock plate is 94.8 MPa which is less than yield strength of 140MPa.

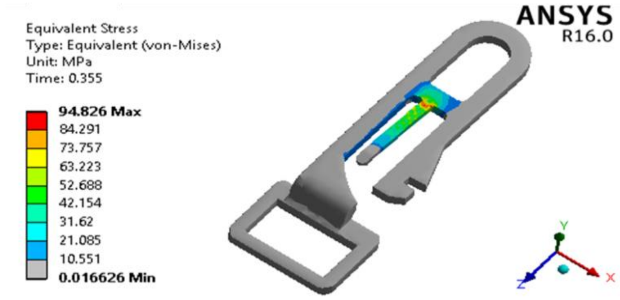

Fig. 7.Von mises stress plot- linear model

The reaction force is checked at the end of the lock plate, the graph is shown in the following figure. The force is increased gradualy and it is reached to a peak value of 43.7N. We can say that force required to insert the lock plate having snap fit joint is 43.7N.

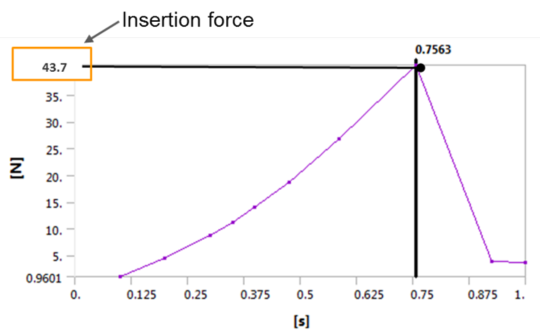

Fig. 8.Reaction force plot: Snap In force

Analysis results of nonlinear model

Now in this step, FE model is solved by considering the nonlinear material model of polypropylene 30% glass filled. Material model assumed is multi linear isotropic hardening. The plastic strain is plot againes the stress value, so the curve starts at the yeild point of material as shown in the figure 4.Non linear frictional contact has been defined in the model. Following are the results obtained from lnear material model of cantolever snap fit.

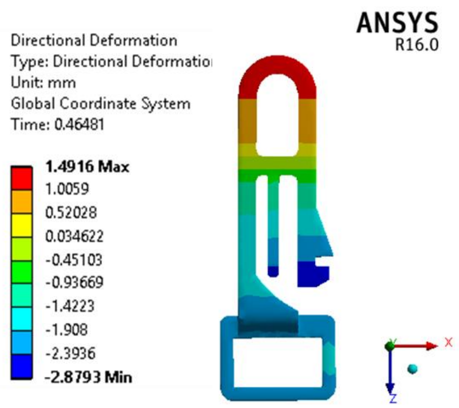

Fig. 9. X-directional deflection plot

As shown in the above image the deflection oberved on snap in x direction is 2.8mm.

Following figures shows the von mises stress plot. The stress occured on lock plate is 85.4 MPa which is less than yeild strength of 140MPa.

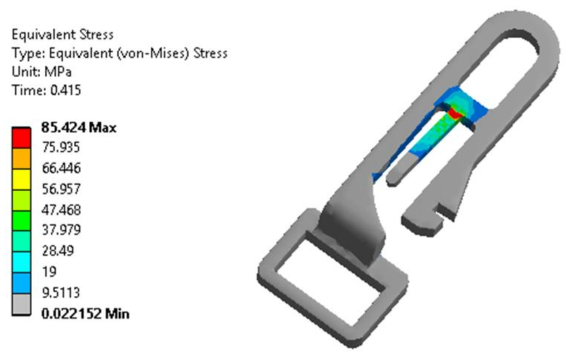

Fig. 10. Von mises stress plot- nonlinear model

Fig. 11.

The reaction force is checked at the end of the lock plate, The graph is shown in the following figure. The force is increased gradualy and it is reached to a peak value of 40.6N. We can say that force required to insert the lock plate having snap fit joint is 40.6N. The curve obtained here for nonlinear model is smoother than the linear material model.

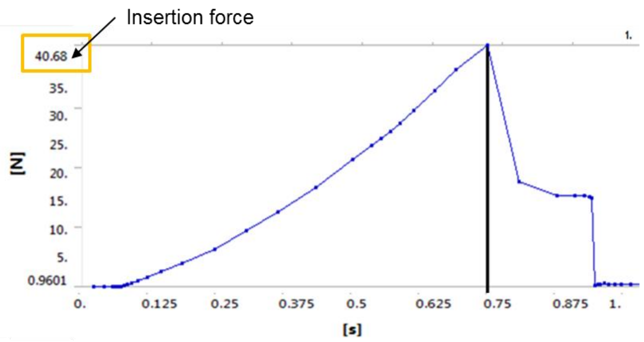

Fig. 12. Reaction force plot: Snap In force

VI. PHYSICAL TESTING:

The force measurement is carried out with the help of Digital Force gauge. A digital force gauge is basically a handheld instrument that contains a load cell, electronic part, software and a display. A load cell is an electronic device that is used to convert a force into an electrical signal. Through a mechanical arrangement, the force being sensed deforms a strain gauge. The strain gauge converts the deformation (strain) to electrical signals. The software and electronics of the force gauge convert the voltage of the load cell into a force value that is displayed on the instrument

The physical setup is as shown in the fig. below. The collar is fixed on the table top with the help of clamps which ensures the firm holding of collar on table top. The lock plate is inserted into the collar by pushing it with the stylus of strain gauge. Care has been taken that the lock plate will gently inserted into the collar and no other noise will affect the insertion force measurement and more accurate force will be measured.

Fig. 13. Physical test setup

From the physical testing the snap in force observd on the force gauge is 39.6N. The above image indicates the physical test setup and the force gauge used to measure the snap in force.

VII. RESULT AND DISCUSSION

In this study an attempt is made to calculate the snap in force for the cantilever snap fit joint. Then numerical formulation has been established with the help of emperical formulae and analytically snap in force is calculated. The snap in force calculated by analytical method is 44.2 N.

Further the FE model has been built and limear material model is defined in the finite element analysis. The snap in force observed in linear finite element analsis is 43.7 N.

In next step to accurately simulate the actual snap in force the non linear finite element analysis is carried out. Frictional contacts has been modeled and multi linear isotropic hardening material curve is defined in FE model. The snap in force observed in non linear FE analysis is 40.6N.

The physical testing is carried out to verify the results obtained from finite element analysis. From physical testing the observed snap in force is 39.6N.

All the results obtained in this atudy are tabulated in following table.

Table- III: Result summary

Model	Snap in force by Linear **FEA (N)**	Snap in force by **nonlinear FEA (N)**	Snap in force by **physical tesing (N)**
Cantilever snap fit	43.7	40.6	39.6

Physical testing results of cantilever snap fit and nonlinear FE analysis are nearly equal. There is only 2.5% variation in nonlinear FE analysis results and physical testing results.

VIII. CONCLUSION

The snap in force required for cantilever snap fit is 40.6N which is very close to physical testing results. So calculated snap force by nonlinear finite element analysis is more realistic and reliable.

The present work has successfully developed a non linear finite element analysis method to calculate the *snap in* force of plastic cantilever snap fit.

The analytical calculations and linear FE analysis estimates higher *snap in* force than the actual. So the *snap in* force of plastic snap fit joint can be estimated most accurately by using nonlinear finite element analysis method.

REFERENCES

1. Eamon Whalen, et al, "Snap fit optimisation for achieving desired insertion and retension forces," Altair Engineering Inc [2018].
2. Jingjing Ji Kok-Meng et al, "Cantilever Snap-Fit Performance Analysis for Haptic Evaluation" ASME 121004-8 / Vol. 133, December 2011.
3. L Goteti et.al, "design optimization of snap-fit integral parts by fea and statistical Analysis", ASME International Mechanical Engineering Congress and Exposition November 5-10, 2006, Chicago, Illinois, USA.
4. Abbet et.al, "Nonlinear FE Analysis "National agency for finite element methods and standards Training course, Oct 17 2016.
5. BASF " Snap-fit Design manual,"http//web.mit.edu/2.75/resourses/random/snap fit
6. Rohit K et.al., "Design and analysis of snap fit joint in plastcs" International Journal of Scientific & Engineering Research, Volume 4, Issue 10, October-201

AUTHORS PROFILE

Hrishikesh kakade, He is presently PG Student of Design engineering in Dr. D Y Patil School of engineering, Pune. He has more than 5 years of working experience in the field of mechanical design. He has published research papers in reputed journals.

Dr. Farook Sayyad, He is presently Professor in mechanical department of Dr. D Y Patil School of engineering, Pune. With nearly two decades of teaching experience, he is author of books Theory of Machine, Dynamics of Machinery, Mechanical Vibration & Noise Engineering. He published more than 30 books for various universities in India. He has published several research papers in national and international reputed Journals and conferences. Also he has published two Patents in Intellectual Property India. He is also a member of various national bodies on technical education and also working as editorial board member and reviewer of various international journals.

Prof. Vinod G Patil, He is Assistant Professor in mechanical engineering department of Dr. D Y Patil School of engineering, Pune. He has 7 years of teaching experience and has published several national and international research papers.