Journal of Advances in Science and Technology
Vol. 12, Issue No. 25, (Special Issue) December-2016, ISSN 2230-9659

Design Optimization of Spring Lock Mechanisms in Cable Lock Adaptor Systems Using Finite Element Method

M. A. Hipparagi[1]*, N. M. Ukkali[2], Biradar Anilkumar[3]

[1]Department of Mechanical Engineering, Hirasugar Institute of Technology, Nidasoshi, Belagavi, Karnataka, India

[2]Department of Mechanical Engineering, Hirasugar Institute of Technology, Nidasoshi, Belagavi, Karnataka, India

[3]Department of Mechanical Engineering, Hirasugar Institute of Technology, Nidasoshi, Belagavi, Karnataka, India

Abstract – Din Rail system is used for cable locking purpose. The cables should maintain certain pretension to maintain its straightness from contact between each other. Dins are clamps placed at intermittent distance to maintain pretension in the cables. So the din should take this wire tension loads. So structural analysis is required to prevent any possible failure of the din material. Din should be electrically resistant and soft material for easier clamping (High strength materials does not deflect easily. They require huge forces for clamping). So ABS-Plastic is considered which is a soft material requires minimum force for clamping. Structural analysis is required to observe the stress concentration regions and to improve it. Due to the advances in computer based Finite element techniques, the modification in design is simplified. Optimization of Adaptor spring lock system using finite element analysis is the main definition of the work. The present work involves analysis of adaptor lock system for its present design and modifications from finite element analysis in improving its structural strength for longer life.

---------------------------♦-----------------------------

INTRODUCTION

A clamp is a device used to join, grip, support, or compress mechanical or structural parts. Its opposing and often adjustable parts provide for bracing objects or holding them together. The clamps ensure a powerful and consistent clamping force and enable components to be loaded and unloaded simply by suitable actuation mechanism. Clamping elements may be either manually operated or actuated by pneumatic, hydraulic or a combination of other power facilities. They are also classified according to the mechanism by which a mechanical advantage is attained. For locking, two basic principles are I) Application of inclined plane theory, i.e. wedges, screws, cams, etc .ii) Application of lever principle, i.e. levers, toggles, etc.

FIG. 1 CLAMPS

2. LITERATURE SURVEY

Contact analysis is a computational bottleneck in many tasks. The difficulties are due to the large number of contacts and to the complexity of the resulting constraints, especially in systems with curved parts and contacts changes, such as chain assemblies, car engines, and VCR mechanisms. Dynamical simulators need to detect contact changes to compute contact forces. Robot path planners need to achieve and maintain contact with target parts, while avoiding collisions with the environment. Designers need to assure that products can be assembled despite small manufacturing variations in the parts. Analogous problems occur in other contact analysis applications. The complexity of contact analysis grows rapidly as the number of parts and part features (vertices, edges, and faces) increases. A pair of touching parts interacts via one or more contacts between feature pairs. For the features to touch without overlap, the part configurations (position and orientation coordinates) must satisfy semi-algebraic constraints.

The geometry of the touching features determines the individual contact constraints, which combine to constrain the system configuration. Hundreds of features per part are the norm, which leads to thousands of potential contacts per pair of parts and to a combinatorial explosion of contacts in most changing contact systems. Manual contact analysis is error-prone and time-consuming at best and is often infeasible. Further complications arise in applications with significant part deformation. The analyst needs to pick a deformation model, to estimate material properties, and to study continuum effects Contact problems and the study of the load transfer in mechanical assemblies are of considerable importance in a wide range of engineering applications. Since Hertz published his work on normal contact between elastic bodies. Many methods are available to study contact problems. Most famous are iterative technique (trial and error). This method consists of calculating the increment of loading and verifying contact condition at each step [2]. To a greater extent, the convergence and computing efficiency of the method depends on the iterative scheme employed. The second is the penalty method. Using this method, we can directly transform the inequalities to equalities by enforcing the constraint condition through penalty functions. The advantage of penalty method is that the method is simple and readily interpreted from a physical standpoint. However, it is well know that the method suffers from ill-conditioning that worsens as penalty values are increased, while constraints are satisfied exactly only in the limit of infinite penalty values. One more method of solving the problems is direct method which consists of reducing the contact problem to a special case of the mathematical programming problem and consequently uses some optimization technique for the resulting constrained quadratic mathematical programming problem.

It should be noted that the applications of mathematical programming techniques to contact problem has been the subject of fruitful scientific preoccupation of many a distinguished scientist Seireg. His scheme can handle the contact problem with small displacements and linear elastic material. Zong and Sun extended the topic to the case of elastic plastic material and small displacements. Sometimes those methods mentioned above also suffer from ill conditioning and the number of iterations for the minimization process on impenetrability condition enforce and the associated algorithm used because the relation between stresses and relative displacements of the contact interface are expressed in the form of a penalty function in those methods i.e. the contact element is introduced as an element with large springs. The springs however to a great extent are artificial. Moreover, the introduction of the penalty function often results in an unsymmetrical stiffness matrix. To overcome the drawback Hung and Saxe presented a general condition of non-interpenetration through introducing a concept of representing the space separating the contacting surfaces, as a fictitious perfectly locking material. The contact will occur if the space disappears. The introduction of such a symmetrical Non-interpenetration condition results in a symmetrical stiffness matrix and does-not suffer from the drawback mentioned above. However the finite element formulation and associated numerical results only apply to plane stress or plane strain case where small displacements are assumed and the contact occurs only on the surfaces between an elastic body and a rigid foundation. We think the major difficulty for the extension of Hung and Saxce's method to contact-impact problems consists in the fact that not only does there exists the highly stress gradient region, but the contact zone is unknown. It may deform during loading and varies with both time and load. Besides some special techniques have also found applications in the analysis of contact problems, such as boundary elements [24], artificial interface element [4], augmented lagrangian technique, Newton method and special search algorithm. From the proceeding survey of the development on the contact problems, an observation may be made. With the third approach, namely the approach about the coupling of finite element methods and mathematical programming, researchers have in the past focused their attention mainly on the development of contact algorithms for static cases in which the associated finite element formulation is derived, based on a simple virtual work principle with some optimization parameters. Small displacement algorithms capable of providing meaningful solutions to contact problems in engineering more commonly observed in practice, such as contact impact problems, geometrically and materially nonlinear contact problems. On the other hand it is obvious that the conventional FEM based on the virtual work principle is inefficient for modeling highly stress gradient problems. Therefore, a finite element methodology, based on the multi-field variation principles is needed to develop for efficiently treating such a category of the problems.

3. STEPS IN A CONTACT ANALYSIS

The basic steps for performing a typical surface-to-surface contact analysis are as follows.

Creating the model geometry and meshing. Identifying the contact pairs. Designating contact and target surfaces. Defining the target surface. Defining the contact surface. Generating the contact elements Setting the real constants Selecting the contact Algorithm. Determining the contact stiffness selecting the friction model applying necessary boundary conditions. Selection of the solver (Symmetric or Unsymmetrical solver) Defining solution options and load steps. Solving the contact problem. Reviewing the results

3.1 Materials

In the present problem, ABS-Plastic is used for din material. Steel is used for rail due to its strength. The properties of materials are as follow

ABS-PLASTIC

Young's Modulus=7672N/mm2

Poison's ratio=0.35

Table 1 Multilinear Properties of ABS-PLASTIC

Strain	Stress (MPa)
0.005 8	44.5
0.0087	65.8
0.0116	84.3
0.0145	100.0
0.0174	112.0
0.0203	123.0
0.0232	131.0
0.0261	137.0
0.0290	139.0

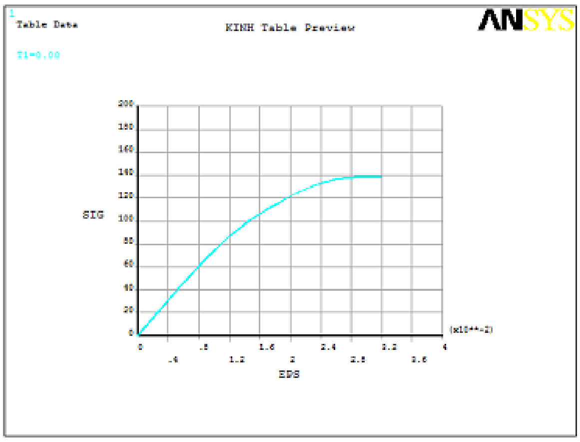


Fig: 2 ABS-PLASTIC- Material Graphs

The graph shows nonlinear material properties for ABS-Plastic. The behavior is nonlinear as the material does-not follow hooks law. Generally strain is considered as the limiting value for failure of the plastic members. In this problem the stress value of 139 Mpa corresponding to 0.029 strain is considered as the breaking point for the member

Table 2 Multilinear Properties of Steel – Material Graph

Strain	Stress(Mpa
0.0042	730
0.0048	815.0
0.0055	880.0
0.0074	1.009.
0.0095	1.100.
0.1350	1.336.

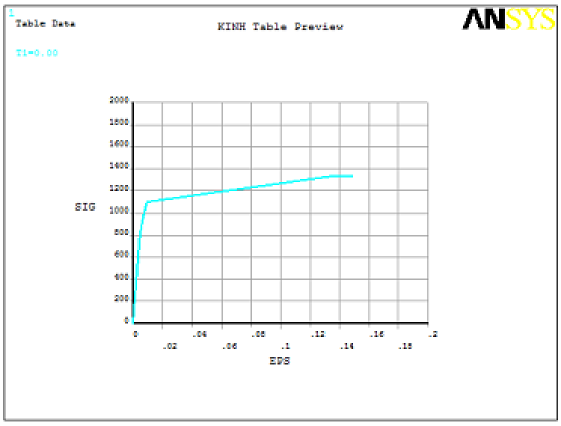


Fig3: Steel Material graph

The graph 3 shows nonlinear properties of steel. A yield stress of 730 N/mm2 can be observed for the steel member. One material goes to permanent deformation, material behavior is nonlinear. The old concept of yield limit is not applicable for the problems as there is a recoverable time based deformation called inelastic deformation is also considered safe for the materials. Due to the importance of cost reduction to meet the competing industry demands, the companies try to optimize the materials to the best possible optimum levels. Earlier no techniques are available to find inelastic deformation, so yield limit was considered as the final limiting stress for structural safety.

4. GEOMETRY OF THE PROBLEM

The figure 3.3 shows present design of the Din Rail system. All major dimensions are represented in the problem. Front top and profile views are represented to get the dimensional view of the problem. All dimensions are represented in mm. Drafting module in Catia is used for representation of the problem.

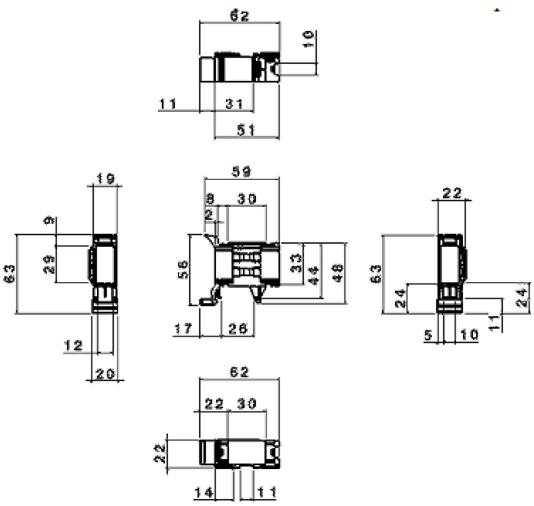


Fig 4: two dimensional view of the Din Rail system

M. A. Hipparagi[1]*, N. M. Ukkali[2], Biradar Anilkumar[3]

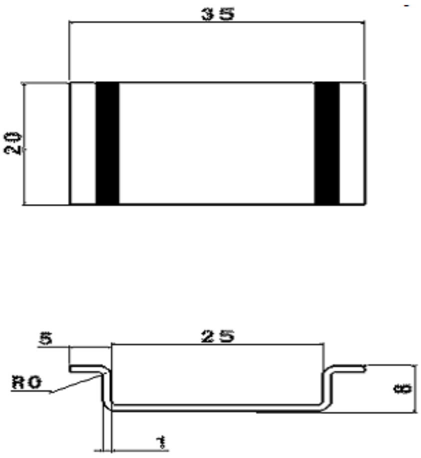


Fig 4, 5: Two dimensional view of Rail system

The figure 4 shows dimensional representation of Rail system. Maximum dimension of 35mm with maximum height of 8mm can be observed for rail system. This member is made of steel for higher strength compared to the ABS-Plastic.

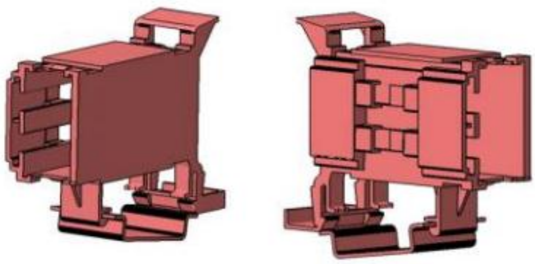


Fig 6 Three Dimensional view of the Din Rail System

The figure shows three dimensional representation of the problem. Catia is used for three dimensional modeling. Initially through sketcher two dimensional views are represented and later converted to three dimensional model using part modeling. Later members are assembled through assembler module available in Catia. Catia is the top level three dimensional modeling software in the industry.

4.1 Meshed Views

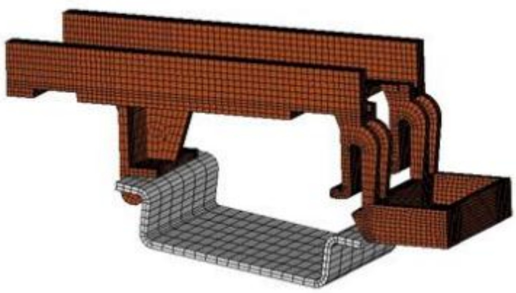


Fig 7: Brick mesh of the Din Rail system

The figure3.6 shows modeled Din Rail system. The model is imported to hyper mesh in step file format and meshed with brick elements. Hyper mesh is having extensive options for better brick meshing. Brick meshing is time taking process and consumes computer resources due to its higher stiffness matrix. But better results, the structure is brick meshed. Material properties are applied through component manager option and also loads are applied through hyper mesh tool option.

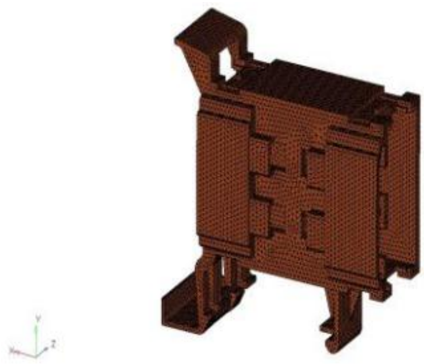


Fig 8: Complete Assembly mesh

The figure 8 shows three dimensional mesh of Din Rail and adaptor system. The model is imported to hyper mesh in 'step' file format for meshing. A tetrahedral mesh is used for analysis. 4 nodded tetrahedral elements defined with solid185 elements are defined. Total number of elements used are 66059 and number of nodes are 20155. Due to the nonlinear material behavior number of elements is minimized to reduce execution problems and memory requirements.

5. THEORETICAL VALIDATION OF THE PRESENT PROBLEM

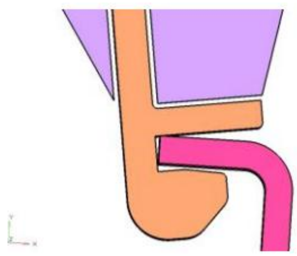


Fig 9: Initial Configuration

The figure 9 shows rail and din assembly. The assembly is almost a clearance fit. This type of joint will not give good results as the members slide with very little forces. And also an erratic contact behavior will be observed during clamping. Dimensional stability Check through basic formulae's:

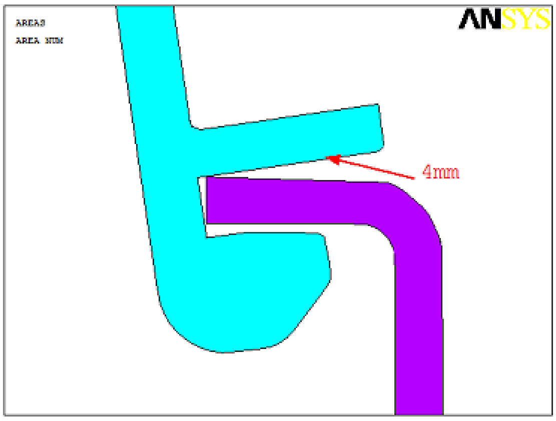


Fig: 10 Contact regions at fixing end

Table 3 Dimensional stability Check through basic formulae's

Allowable stress for ABS-Plastic	Σ=139N/mm^2.
Possible Maximum concentrated load	=200N
Assuming as concentrated cantilever beam M_{max}	=200*2=400N-mm
Width of the problem	B=12.5mm
Sectional modulus required Z	Z=400/139=2.877 =bh^2/6=2.877
H	=(2.877*6/12.5)$^{1/2}$=1.175mm
Thickness provided for the member So1.2mm.structural thickness is safe.	1.2mm.
If load is distributed like uniformly distributed load (udl) Udl	= w= 200/4=50N/mm
Mmax	=wL^2/8=50*4^2/2=80*16/8=400N-mm

Allowable stress for ABS-Plastic Σ=139N/mm2.

Possible Maximum concentrated load =200N

Assuming as concentrated cantilever beam Mmax =200*2=400N-mm

Width of the problem B=12.5mm

Sectional modulus required

Z Z=400/139=2.877

=bh2/6=2.877

H =(2.877*6/12.5)1/2=1.175mm

Thickness provided for the member

So1.2mm.structural thickness is safe. 1.2mm.

If load is distributed like uniformly distributed load (udl)

Udl = w= 200/4=50N/mm

Mmax =wL2/8=50*42/2=80*16/8=400N-mm

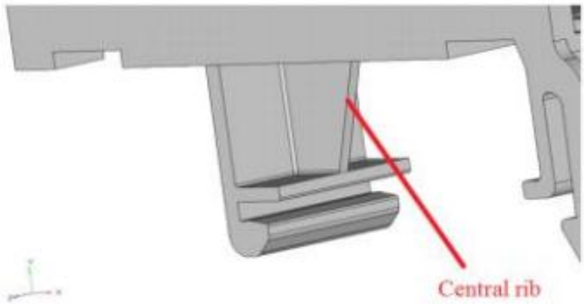


Fig 11: Provision of central rib to increase the strength

For the water sensor circuit, the plate will be used as the component to sense the existing of water. However, this sensor was not very effective as it senses the water, the system will automatically ON the wiper at only one speed. The second conceptual design is a two lines plate with a high and low line. The function of the low line plate is to detect the showery rain. The high line plate is to detect the heavy rain. The third conceptual used the stamping concept. The stamping sensor is a good sensor for sensing the moist, any kind of water or raindrops. Nevertheless the sensor is very expensive. After considered all these criteria, second conceptual design was chosen to construct the circuit. The circuit was changes from single relay to double relay as an input to PIC controller system.

6. RESULTS & DISCUSSION

The din rail system is analyzed using Finite element software Ansys for improvement in the design. Initially the existing design is analyzed and later analysis is carried out using improved methodology. Due to complexity involved in the three dimensional problem with multilinear nonlinear problem which consumes extensive solution time and memory requirements and improper convergences, the problem is analyzed using two dimensional domain and results are presented.

Three dimensional analysis shows stress reaching to the critical value of 139 Mpa. This stress distribution is mainly near the corner regions. The figure shows Hexahedra mesh analysis. But due to contact difficulties only contact pairs are created at the sliding ends. Nonlinear properties are applied to check the realistic stress distribution in the structure due to locking mechanism. So structure will fail as the stresses are reaching to the critical values. So some modification is required for the problem for proper functioning. So by giving proper fillets, this stress raise can be reduced. A three dimensional problem considers the actual geometries of design will be considered. But geometrical variation and execution is a long process which is not suitable for initial design stages. So further a two dimensional analysis is considered to improve the problem.

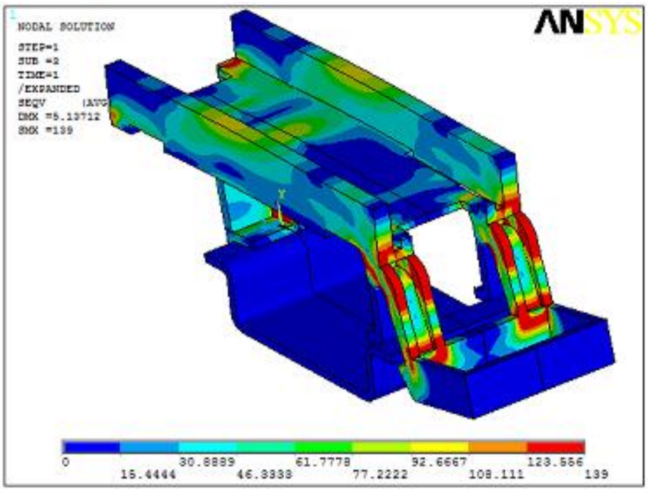


Fig12; Stress distribution in the member

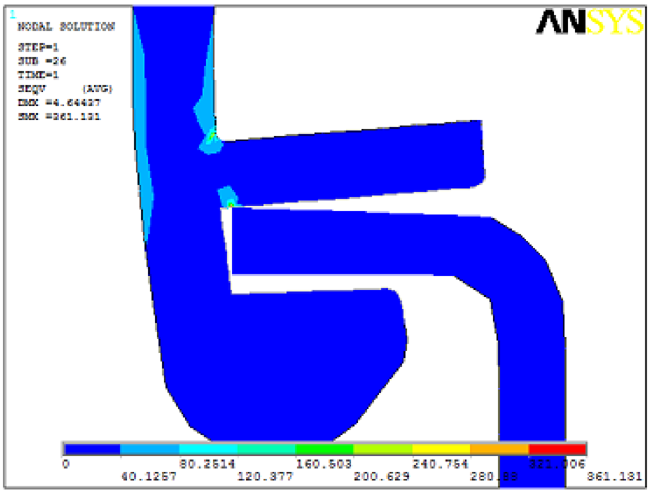


Fig13: Vonmises stress in the member

The figure 13 shows vonmises stress development in the structure. Maximum stress is around 361.131Mpa which is more than the allowable stress of 139Mpa for the material. This is due to unpredicted contact mechanics problem where localization of herzian stress takes place which are the causes of failure. These concentrated stresses are the main source of weakness of structure and causes the cracking of the member. So this region needs to be modified.

6.1 Improvement Analysis

Geometry is modified at the following regions: The geometry is modified in such a way that it form a push fit at the joint. No clearance is provided between din and rail system at the fixing end. The modified mesh is shown the end with node to node connection for proper convergence.

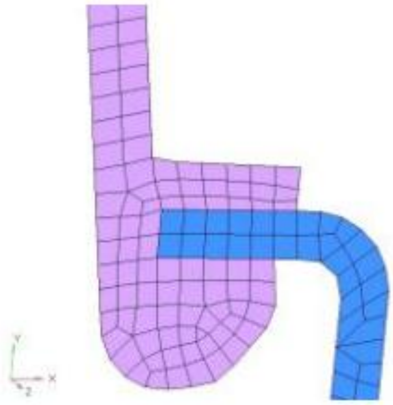


Fig14; Improvement at the fixing end

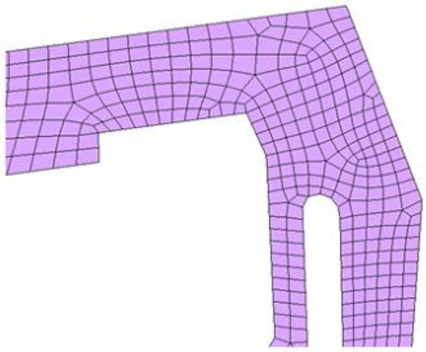


Fig: 15: Improvement in the fillet end

The figure 15 shows modified fillet region to reduce stress localization. The mesh in the region is also shown.

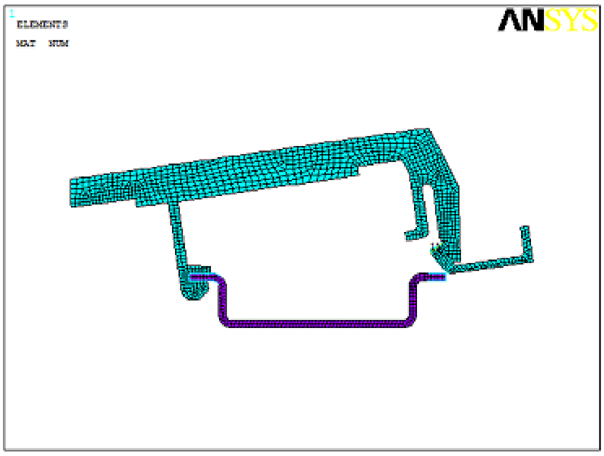


Fig 16 Dimensional improved mesh

The figure 16 shows two dimensional mesh of the geometry. Elements are minimized due to the nonlinear material and geometrical requirements. Fine mesh is used only in the region of stress localization. Contact pairs are created at both ends for estimation of stresses and contact pressure distribution.

6.2 Analysis Results

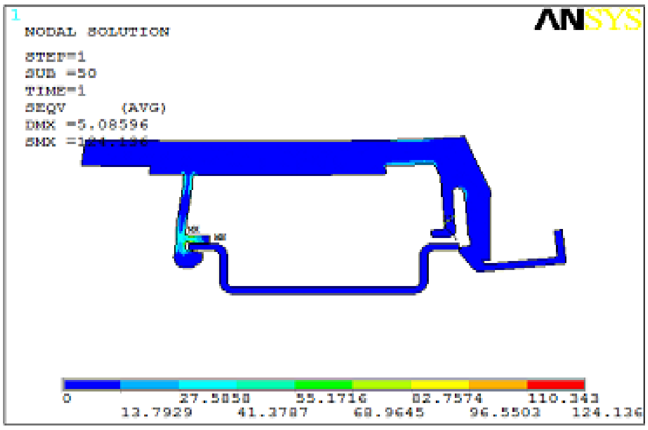


Fig17: With modification at fixed end (Push fit)

The figure shows maximum stress development of 124.136Mpa. This stress is less than the allowable stress of 139 Mpa. So the region is improved for stress analysis. 'MX' symbol shows region of maximum stress. Symbol 'MN' shows minimum stress region.

The member shows improper contact during locking process. So further improvement is required in this region. During sliding the hard steel region is coming in contact certain regions and stress distribution is improper and increasing the value to 124.136 Mpa.

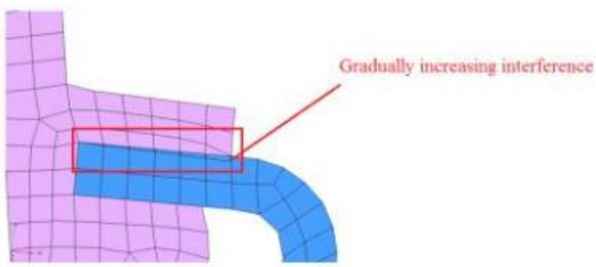


Fig 18: Interference in the Assembly

The figure shows press fit of the joint. Gradually increasing interference of 0.1 mm is provided at the end. The interference is provided to increase the stiffness of the structure. Further analysis is carried out to check the results. Generally press fits increases the stress pattern of the problem and also increases holding capacity

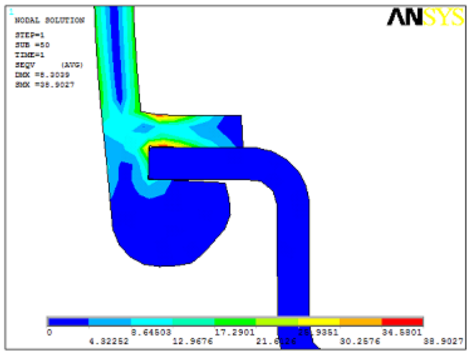


Fig 19; Vonmises stress at the joint

The figure 19 shows reduction of stress in the region compared to the previous configuration. The stress value is reduced from 124Mpa to 38.9Mpa. This is due to interference and uniform distribution of load. Also very little opening can be observed compared to the previous configurations where almost separation can be observed in the assembly.

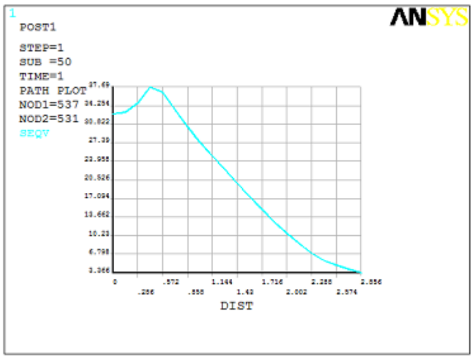
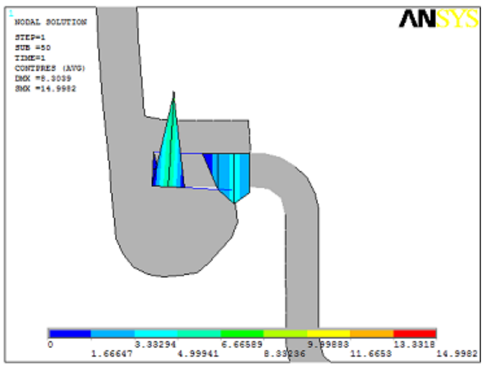


Fig 20: Vonmises stress variation along the fixed end

The figure 20 shows variation of stress along the fixed end. The stress pattern is similar to the previous configuration, but values are less. Since higher factor of safety 3.6 (139/38.5) can be observed in the problem the geometrical changes are complete for structural safety of the din rail adaptor system.



The figure 21 shows contact pressure at the fixed end.

Maximum of 14.9982 Mpa contact pressure can be observed in the problem. The contact pressure is lesser then the previous configuration indicating even distributions of load along the interface between din and rail system.

REFERENCES

H. Hertz, J.F. Math. 92,156-171,1881.

S.H. Chan and I.S. Tuba, A finite element method for contact problems of solid bodies, Internet, J. Mech. Sci, 13,615-639,1971

T.F. Conry and A. Seireg, A mathematical programming method for design of elastic bodies in contact, J. Appl. Mech. 38, 387-392,1971

L.R. Herrmann, Finite element analysis of contact problems, J. Eng. Mech., ASCE104,1043-1057,1978

N.D. Hung and G.De Saxce, Frictionless contact of elastic bodies by finite element and mathematical programming techniques, Comput. Structure. 11,55-67,1980

J.T.Oden, Exterior penalty methods for contact problems in elasticity, in: K.J. Bathe, E. Stein and W. wunderlich, eds., Nonlinear Finite element Analysis in Structural Mechanics (Europe-US Workshop, Springer, Belin,1980

J.A. Landers and R.L. Taylor, An augmented Lagrangian formulation for the finite element solution of the contact problems, SESM Rep., 85/09, University of California, Berkeley,1985.

J.G. Hallquist, Sliding interface with contact-impact in large scale Lagrangian computations, Comput. Methods Appl Mech. Engg. 51,107-137,1985.

W.X. Zhong and S.M.Sun, A finite element method for elastic-plastic structure and contact problem by parametric quadratic programming, Internet, J. Number. Methods, 26,2723 /2738, 1988

Q.H. Qin, "Variation Principles, FE and MPT for analysis of non-linear impact contact problems", Comput. Methods Appl, Mech. Engg. 122, 1994

Nagi Elabbasi, "On the reliable solution of contact problems in Engineering design", International Journal of Mechanics and Materials in Design 1:3-16, 2004.

S. Ramamrutham, R. Narayan, Strength of Materials, Dhanpat Rai & Sons, 11th editions 2006.

Corresponding Author

M. A. Hipparagi*

Department of Mechanical Engineering, Hirasugar Institute of Technology, Nidasoshi, Belagavi, Karnataka, India

E-Mail – ahmaheshwar@gmail.com