International Research Journal of Engineering and Technology (IRJET) **e-ISSN: 2395-0056**
Volume: 02 Issue: 09 | Dec-2015 **www.irjet.net** **p-ISSN: 2395-0072**

STRESS ANALYSIS OF INTERFERENCE FIT BY FEM

Rahul G.Karmankar

Assistant Professor, Mechanical Engineering department,Nagpur University, Maharashtra,India

***

ABSTRACT

This paper shows the results of an Interference fit by using finite element method. By considering 001 mm allowance in pin connection through hole the results are carried out for contact stress and Von –mises stress. Typical examples of interference fits are the press fitting of shafts into bearing or bearings into their housings and the attachment of watertight connectors to cables . Interference fit also results when pipe fittings are assembled and tightened. ln our applications with interference fit pins in plated through holes, it is necessary to determine regions of high stress. The regions of high stress may be a source of which lead to crack failure. In order to understand the source of these high stress regions, an analysis of the process of the insertion of a pin into a hole is considered. Hereby using FEM Von-mises stress and contact stress has been studied with respect to various allowances .Inertia effects are not considered in this analysis. The finite element method is used to determine the solutions of this analysis. Shear stress set up at this interface is not large enough to cause cracking.

Keywords:- Interference fit, stress analysis,FEM,contact stress.

1.INTRODUCTION

Interference-fit pin connections have wide applications ranging from aerospace structures to electric hardware systems and the telephone industry. In order to derive the maximum benefit by the use of interference-fit pin in all such applications, a complete understanding of their behavior in the regions of joints is essential.

Definition:-

The opposite of clearance , for mating cylindrical parts in which the internal member is larger than the external member. Means shaft is larger than hole.

Also called a shrink or press fit. It gets its name because the bore is actually smaller than the shaft it is to be mated with. It is the strongest fit possible but requires heat or a hydraulic press to install.Interference fit refers to parts that must be compressed to mate.

Often the edges of shafts and holes are chamfered (beveled). The chamfer forms a guide for the pressing movement, helping

(a) to distribute the force evenly around the circumference of the hole, and

(b) to allow the compression to occur gradually instead of all at once, thus helping the pressing operation to be smoother, to be more easily controlled, and to require less power (less force at any one instant of time).Most materials expand when heated and shrink when cooled. Enveloping parts are heated (e.g., such as with torches or gas ovens) and assembled into position while hot, then allowed to cool and contract back to their former size, except for the compression that results from each interfering with the other. Railroad axles, wheels, and tires are typically assembled in this way. Alternatively, the enveloped part may be cooled before assembly such that it slides easily into its mating part. Upon warming, it expands and interferes. Cooling is often preferable as it is less likely than heating to change material properties, e.g. assembling a hardened gear over a shaft, where heating the gear would alter its hardness. Interference-fit pin connections have wide applications ranging from aerospace structures to electric hardware systems and the telephone industry. In order to derive the maximum benefit by the use of interference-fit pin in all such applications, a complete understanding of their behavior in the regions of joints is essential.

2 ANALYSIS BY FEM.

Dr. John Swanson founded ANSYS,inc. in 1970 with vision to commercialize the concept of computer-simulated engineering ,establishing himself as one of the pioneers of finite element analysis [FEA]. ANSYS Inc supports the ongoing development of innovative technology and deliver flexible, enterprise-wide engineering system that enable companies to solve the full range of analysis problem, maximizing their existing investment in software and hardware..ANSYS Inc. continues its role as a technological innovator. It also supports a process-centric approach to design and manufacturing, allowing users to avoid expensive and time consuming "build and break cycle". ANSYS analysis and simulation tools give customer ease of use, data compatibility, multiplatform support and coupled-field multi-physics capabilities.

2.1 **EVOLUTION OF ANSYS PROGRAM**

ANSYS has evolved into multipurpose design analysis software program, recognized around the world for its main capabilities. Today, the program is extremely powerful and easy to use. Each release hosts new and enhanced capabilities that make the program more flexible, more usable and faster. In this way, ANSYS helps engineers meet the pressure and demand of the modern product development.

2.2 **OVERVIEW OF THE PROGRAM**

The ANSYS program is flexible, robust design analysis and optimization package. The software operates on major computer and operating system from PCs to workstation to supercomputers. ANSYS features file computability throughout the family of product-aided design models into ANSYS, eliminating report work. This ensures enterprise wide, Flexible engineering solution for all ANSYS users.

2.3 **REDUCING DESIGN AND MANUFACTURING COSTS WITH ANSYS, FEM**

The ANSYS program allows engineers to construct computer models or transfer CAD models of structures, products, components, or systems, apply operating loads or other design performance conditions and study physical response such as stress levels, temperature distribution or impact of electromagnetic field. In some environments, prototype testing is understand or impossible. ANSYS design optimization enables the engineers to reduce the number of costly prototypes, rigidly and flexible to meet objective and the proper balance in geometric modifications. Competitive companies look for ways to produce the highest quality product at lowest cost. ANSYS FEA can help significantly by reducing the design and manufacturing cost and by giving engineers added confidence in the conceptual design. It is also useful when used later in manufacturing process to verify the final design before prototyping

2.4 ELEMENT INPUT

This table usually contains the following items:

1.Element Name

2.Nodes

3. DOF

4.Real constant

5. Material properties

.

6. Special feature.

7. KEYOPTS

2.5 TYPE OF ELEMENT USE TO ELABORATE INTERFERENCE FIT.

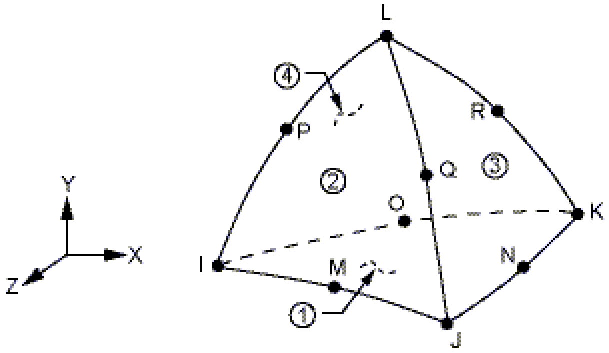
1] Solid-92
2] Target 170
3] Conta174 etc

2.6 SOLID2 ELEMENT DESCRIPTION

SOLID92 has a quadratic displacement behavior and is well suited to model irregular meshes (such as produced from various CAD/CAM systems The element is defined by ten nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element also has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities.

Figure 4A-SOLID92

Geometry



2.7 SOLID92 INPUT DATA:-

The geometry, node locations, and the coordinate system for this element are shown in Geometry". Beside the nodes, the element input data includes the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Systems. Element loads are described in Node and Element Loads. Pressures may be input as surface loads on the element faces as shown by the circled numbers on. Positive pressures act into the element.. Pressure load stiffness effects are included in linear eigenvalue buckling automatically. If an unsymmetrical matrix is needed for pressure load stiffness effects, use NROPT, UNSYM.

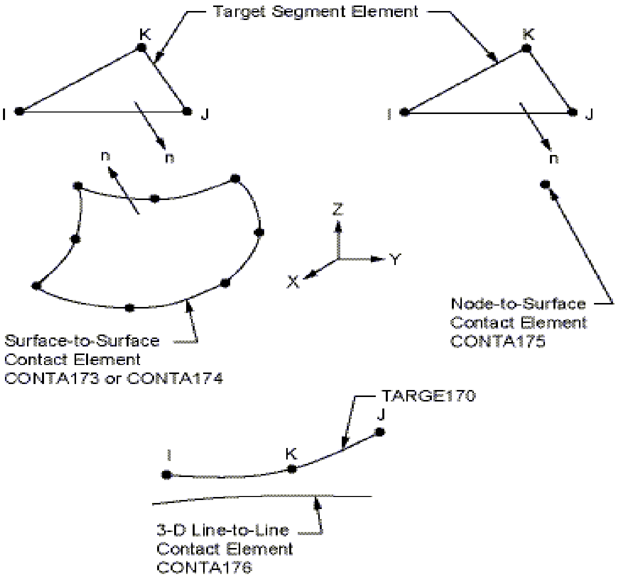
Special Features

- Plasticity
- Creep
- Swelling
- Stress stiffening
- Large deflection
- Large strain
- Birth and death
- Adaptive descent
- Initial stress import

2.8 TARGE170 ELEMENT DESCRIPTION:-

TARGE170 is used to represent various 3-D "target" surfaces for the associated contact elements (CONTA173, CONTA174, CONTA175, and CONTA176). The contact elements themselves overlay the solid elements describing the boundary of a deformable body and are potentially in contact with the target surface, defined by TARGE170. This target surface is discretized by a set of target segment elements (TARGE170) and is paired with its associated contact surface via a shared real constant set. It can impose any translational or rotational displacement, temperature, voltage, and magnetic potential on the target segment element. For rigid target surfaces, these elements can easily model complex target shapes. For flexible targets, these elements will overlay the solid elements describing the boundary of the deformable target body.

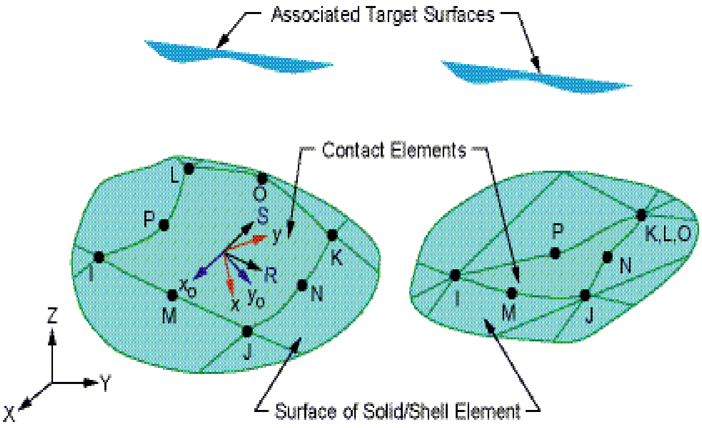
Figure 170.1 TARGE170 Geometry:-



2.9 CONTA174 ELEMENT DESCRIPTION

CONTA174 is used to represent contact and sliding between 3-D "target" surfaces (TARGE170) and a deformable surface, defined by this element. The element is applicable to 3-D structural and coupled field contact analyses. This element is located on the surfaces of 3-D solid or shell elements with midside nodes It has the same geometric characteristics as the solid or shell element face with which it is connected (see Figure 174.1:CONTA 174 " Geometry" below). Contact occurs when the element surface penetrates one of the target segment elements (TARGE170) on a specified target surface. Coulomb and shear stress friction is allowed.

Figure 174. CONTA 174 Geometry



R = Element x-axis for isotropic friction

x_0 = Element axis for orthotropic friction if ESYS is not supplied (parallel to global X-axis)

x = Element axis for orthotropic friction if ESYS is supplied

2.10 CONTA Input Data:-

The geometry and node locations are shown in Figure 174.1: CONTA 174Geometry". The element is defined by eight nodes (the underlying solid or shell element has midside nodes). It can degenerate to a six node element depending on the shape of the underlying solid or shell elements. If the underlying solid or shell elements do not have midside nodes, use CONTA173 The node ordering is consistent with the node ordering for the underlying solid or shell element. The positive normal is given by the right-hand rule going around the nodes of the element and is identical to the external normal direction of the underlying solid or shell element surface. For shell elements, the same nodal ordering between shell and contact elements defines upper surface contact; otherwise, it represents bottom surface contact. Remember the target surfaces must always be on its outward normal direction CONTA 174 supports isotropic and orthotropic Coulomb friction. For isotropic friction, specify a single coefficient of friction, MU, using either TB command input (recommended) or the MP command. For orthotropic friction, specify two coefficients of friction, MU1 and MU2, in two principal directions using TB command input. For isotropic friction, the applicable coordinate system is the default element coordinate system (noted by the R and S axes in the above figure). For orthotropic friction, the principal directions are determined as follows. The global coordinate system is used by default The first principal direction is defined by projecting the first direction of the chosen coordinate system onto the contact surface. The second principal direction is defined by taking a cross product of the first principal direction and the contact normal. These directions also follow the rigid body rotation of the contact element to correctly model the directional dependence of friction. Be careful to choose the coordinate system (global or local) so that the first direction of that system is within 45° of the tangent to the

2.11 MANAGING CONTACT PAIRS:-

This tool used for proper detection of the contact pair. It also

1. Verify that the normal of the contact and target surfaces are in the correct direction
2. Reverse normal of elements that are not oriented correctly

In addition these elements can be displayed independently or in the context of entire model. In the later case the contact elements are highlighted in a translucent plot the model. Another important function is to edit the properties of the contact pair(s) as needed. The properties include real constant values and key option values as discussed earlier. The Contact Properties button in the contact manager provides a simple to use interface that allows the properties of the selected contact pair(s) to be reviewed and modified if needed.

2.12DIFFICULTY TO SOLVE CONTACT PROBLEM:-CONTACT OVERVIEW

Contact problems are highly nonlinear and require significant computer resources to solve. It is important that you understand the physics of the problem and take the time to set up your model to run as efficiently as possible. Contact problems present two significant difficulties. First, you generally do not know the regions of contact until you've run the problem. Depending on the loads, material, boundary conditions, and other factors, surfaces can come into and go out of contact with each other in a largely unpredictable and abrupt manner. Second, most contact problems need to account for friction. There are several friction laws and models to choose from, and all are nonlinear. Frictional response can be chaotic, making solution convergence difficult In addition to these two difficulties, many contact problems must also address multi-field effects, such as the conductance of heat, electrical currents, and magnetic flux in the areas of contact.

2.12 GENERAL CONTACT CLASSIFICATIONS

Contact problems fall into two general classes: rigid-to-flexible and flexible-to-flexible. In rigid-to-flexible contact problems, one or more of the contacting surfaces are treated as rigid (i.e., it has a much higher stiffness relative to the deformable body it contacts). In general, any time a soft material comes in contact with a hard material, the problem may be assumed to be rigid-to-flexible. Many metal forming problems fall into this category. The other class, flexible-to-flexible, is the more common type. In this case, both (or all) contacting bodies are deformable (i.e., have similar stiff nesses). An example of a flexible-to-flexible contact is bolted flanges.

2.13 NODE-TO-SURFACE CONTACT ELEMENTS

Unlike the node-to-node contact elements, you do not need to know the exact location of the contacting area beforehand, nor do the contacting components need to have a compatible mesh. Large deformation and large relative sliding are allowed, although this capability can also model small sliding.

2.14:-WHY IS THE CONTACT PROBLEM SIGNIFICANTLY DIFFICULT.

Despite the importance of contact in the mechanics of solids and its engineering applications, contact effects are rarely seriously taken into account in conventional engineering analysis, because of the extreme complexity involved. Mechanical problem involving contacts are inherently nonlinear. Why is it “nonlinear” behavior? Usually the loading causes significant changes in stiffness, which results in a structure that is nonlinear. Nonlinear structural behavior arises for a number of reasons, which can be reduced to three main categories. (1) Geometric Nonlinearities (Large Strains, Large Deflections) (2) Material Nonlinearities (Plasticity) (3) Change in Status

Nonlinearities (Contact), so the contact between two bodies belongs to the case (3).Contact problems present many difficulties. First, the actual region of contact between deformable bodies in contact is not known until the solution has been obtained. Depending on the loads, materials, and boundary conditions, along with other factors, surfaces can come into and go out of contact with each other in a largely unpredictable manner. Secondly, most contact problems need to account for friction. The modeling of friction is very difficult as the friction depends on the surface smoothness, the physical and chemical properties of the material, the properties of any lubricant that might be present in the motion, and the temperature, of the contacting surfaces. There are several friction laws and models to choose from, and all are nonlinear. Frictional response can be chaotic, making solution convergence difficult (ANSYS). In addition to those difficulties, many contact problems must also address multi-field effects, such as conductance of hear and electrical currents in the areas of contact. Bodies in contact may have complicated geometries and material properties and may deform in a seemingly arbitrary With the rapid development of computational mechanics, however, great progress has been made in numerical analysis of the problem. Using the finite element method, many contact problems, ranging from relatively simple ones to quite4 complicated ones, can be solved with high accuracy. The finite element method can be considered the favourites method to treat contact problems, because of its proved success in treating a wide range of engineering problem in areas of solid mechanics, fluid flow, heat transfer, and for electromagnetic field and coupled field problems.

2.15:-HOW TO SOLVE THE CONTACT PROBLEM

In order to handle contact of pin and hole problems in interference fit with the finite element method, the stiffness relationship between the two contact areas is usually established through a spring that is placed between the two contacting areas. This can be achieved by inserting a contact element placed in between the two areas where contact occurs. There are two methods of satisfying contact compatibility: (i) a penalty method and (ii) a combined penalty plus a Lagrange multiplied method. The penalty method enforces approximate compatibility by means of contact stiffness. The combined penalty plus Lagrange multiplier approach satisfies compatibility to a user-defined precision by the generation of additional contact forces that are preferred to as Lagrange forces. It is essential to prevent the two areas from passing through each other. This method of enforcing contact compatibility is called the penalty method. The penalty allows surface penetrations, which can be controlled by changing the penalty parameter of the combined normal contact stiffness. If the combined normal contact stiffness is too small, the surface penetration may be too large, which may cause unacceptable errors. Thus the stiffness must be big enough to keep the surface penetrations below a certain level. On the other hand, if the penalty parameter is too large, then the combined normal contact stiffness may produce several numerical problems in the solution process or simply make a solution impossible to achieve. For most contact analyses of huge solid models the value of the combined normal contact stiffness may be estimated [ANSYS] as.$Kn = fEh$

Where f is a factor that controls contact compatibility. This factor is usually be between 0.01 and 100.

E = smallest value of Young's Modulus of the contacting materials H = the contact length

The contact stiffness is the penalty parameter, which is a real constant of the contact element. There are two kinds of contact stiffness, the combined normal contact stiffness and the combined tangential or sticking contact stiffness. The element is based on two stiffness values. They are the combined normal contact stiffness Kn and the combined tangential contact stiffness $K.l$. The combined normal contact stiffness Kn is used to penalize interpenetration between the two bodies, while the combined tangential contact stiffness Kl is used to approximate the sudden jump in the tangential force, as represented by the Coulomb friction when sliding is detected between the contacting nodes. However, serious convergence difficulties may exist during the vertical loading process and application of the tangential load often results in divergence. A details examination of the model's nodal force during the vertical loading may indicated the problem. Not only are friction forces developing but they develop in random diffractions. This is due to Poisson's effect causing small transverse deflections of the nodes in the contact zone. These deflections are enough to activate the friction forces of the contact elements [1]. The friction forces are developing in various directions because the generation of a tangential friction force facing right on one node would tend to pull the node on its left to the right. This would generate a friction force facing left on this node, pulling back on the other node. This continual tug-of-war cause the poor convergence This problem was eliminated by applying a small rotation to the above cylinder model forces as it was displaced and loaded vertically, this rotation ensured that the friction forces would develop in the proper direction.Interference contact problem cab be solved by using the above method.Here for solving the contact problem contact element of CONTA174 is used.-----Following fig. Shows the basics relationship between the contact and target surface.By using this method we can solve the problem of Interference fit for flexible to flexible contact.

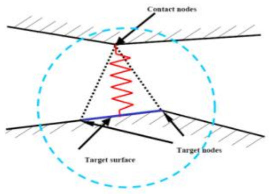


Figure 3-1 Point-to-surface contact element

IMAGES

. Fig. shows the stress analysis of interference fit for pin and hole.

Fig.1. Initial condition

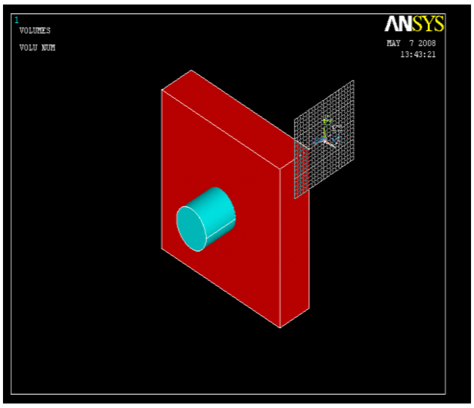
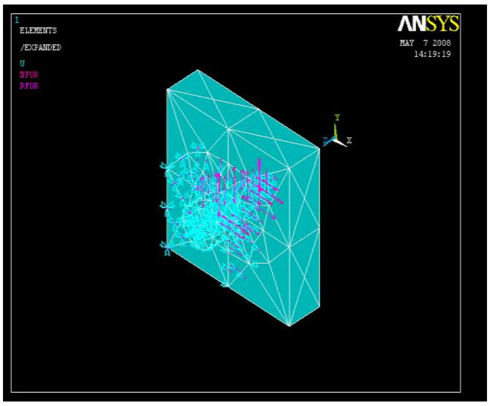
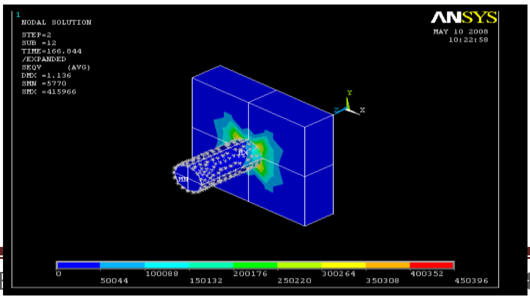
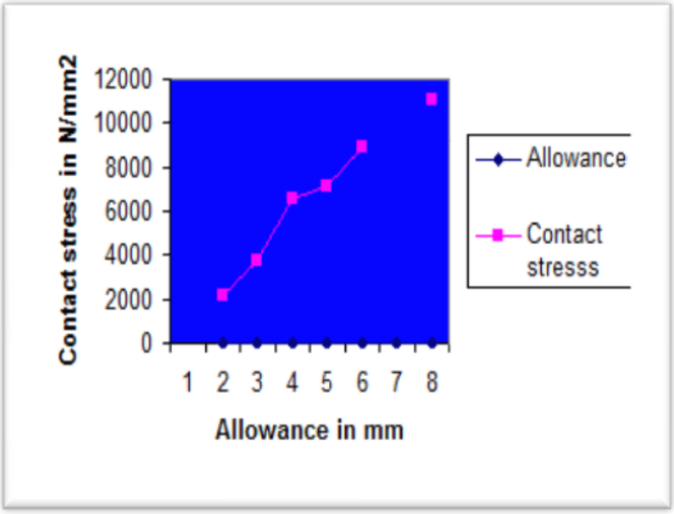


Fig.4 Stress at the location of interference fit .







PROCESS FOR INTERFERENCE FIT ASSEMBLY

A metal part will expand when it is heated and will shrink back to its original size when it cools again. This phenomenon is used to advantage for the assembly of some components – particularly in the automotive drive train and medical industries. For example, it is common for a gear to be attached to a shaft by heating the gear so that its inner diameter, which is slightly smaller than the diameter of the shaft, expands to be larger than the diameter of the shaft. The hot gear is slid over the shaft, and then as it cools it shrinks and sticks in place. If the gear is not hot enough it will not expand enough to fit over the shaft. If the gear is overheated it may warp or temper. Clearly, precision temperature control is required.

FEM RESULT

Force fit suitable for parts which can be highly stressed or for shrink fits where the heavy pressing forces required are impractical. Hole Basis - H7/u6, Shaft Basis - U7/h6

A]GRAPH SHOWS THE RELATION BETWEEN ALLOWANCE AND VON-MISES STRESS.

Here form above graph it is clear that value of Von-mises stress are increases as allowance are increase.But from study it is clear that value of Von-mises stress decrease as if allowance are further increase, thus there are standard value for Interference should be take as per our requirement.

3. SCOPE FOR FUTURE WORK

1] Though there are standard values for the allowance but it should be test for various conditions like effect of temperature, heat conductivity, inertia effect.

2] Allowance plays an important role in various leak proof application Like Gas, air, liquid etc. And hence study of Interference is very essential to avoid the future failure.

3] Future work can be carried out by considering the Inertia effect and we can find the reasonable value for allowance

APPLICATION

1.Tyre fitting 2. Railway wheel and axel fitting

3. Gear and shaft fitting 4. Shaft and coupling fitting.

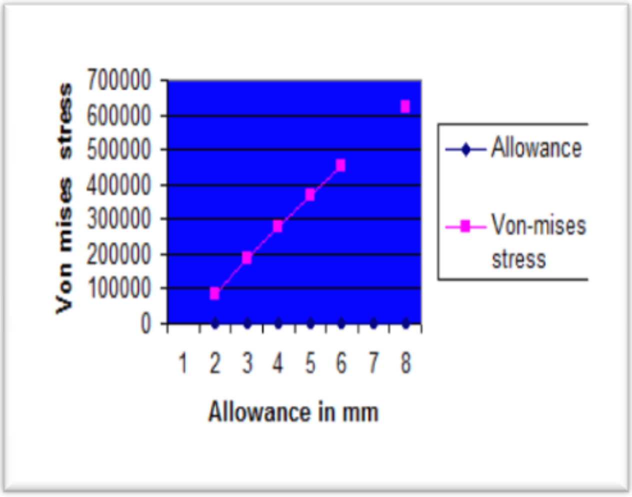
5.Shaft and Bush fit. 6. Cable joint fit for water proof.

B] This graph shows the relationship between contact stress and allowance.

3.1RESULT:- Result has been carried out for exact size of hole and shaft and noted the values of von-mises stress for perfect size of hole and shaft.Also analys is carried out for different no.of allowance .Result show the relation between contact stress and allowance which can be cleared from above graph.Main objective of this analysis is to find out the exact size of hole and shaft for interference fit which is applicable for press fit or force fit.

VON MISES STRESS

In this case, a material is said to start yielding when its von Mises stress reaches a critical value known as the yield strength, σ_y. The von Mises stress is used to predict yielding of materials under any loading condition from results of simple uniaxial tensile tests. The von Mises stress satisfies the property that two stress states with equal distortion energy have equal von Mises stress.Hencky (1924) offered a physical interpretation of von Mises criterion suggesting that yielding begins when the elastic energy of distortion reaches a critical value. For this, the von Mises criterion is also known as the maximum distortion strain energy criterion.This analysis show the best combination for hole and shaft with perfect allowance .It also help to avoid failure at the location of interference fit..This Analysis has been carried out for different no. of holes and allowance with respect to shaft diameter.Here Shaft basis basis system is used.



No	Allo. in mm	Contact-stress	Von.Mis Stress
1	0.001	2160 E-06	8.28E-02
2	0.002	3717 E-06	189948 E-06
3	0.003	6598 E-06	280476 E-06
4	0.004	7139 E-06	366706 E-06
5	0.005	8861 E-06	452983 E-06
6	0.006	1.01E-02	5.03E-01
7	0.007	11092 E-06	622790 E-06

.3..2CONCLUSION

By using the strategy of Interference fit in plated through hole we found the value of high stress which leads the crack failure criteria. Generally negative allowances are provide for press fit. By using this method we can find suitable value for allowance which will help to design reliable product. This method is used to find the critical value of the Von misses stress, contact stress etc. It help to investigate the critical value of high stress which can avoid the future failure of the product like matting of shaft and bearing, watertight connectors to cable, aerospace structures. Here the value of Von-mises stress for Interference fit has been found by considering 0.01mm allowance. Works has been also carried out for various allowance means from 0.001mm to 0.007 mm improve the accuracy of the results.There are mathematical procedure are also available for the study of the Interference fit but like others FEM also give best results which will help to study the Interference fit with various consideration like temperature , contact pressure, thermal conductivity ,inertia etc .

3.3REFERENCE

1] Tirupathi R. Chandrupatla, Ashok D. Belegundu” Introduction of finite elements in Engineering” PHI New Delhi-1991

2] S.S.Rao,” The finite element method in Engineering “Pergamon press, 1989

3] O.C. Zienkiewicz, “Finite element method “-Mc Graw Hill, New York, 1977

4] Eliahu Zahavi,” The finite element in Machine design “Prentice hall, Englewood Cliffs, New Jersey, 1992

5] Panday and Shah,’ Machine Design “, Charotar Publishing house, India 1994

6] V.B. Bandhari” Introduction to Machine Design “Tata Mc. Grawhill.

7] Khurmi and Gupta ‘Machine Design’

8] Web site- www.innovative .com

www.design engineer.com

www.sciencedirect.com