The Nonlinear Contact Simulation for Manufacturing Process by Using ANSYS Program as Case study with Cylindrical rolling contact

Ammar Dawood Ghali
M.Sc. Die and Tools

Wasit University / Engineering College / Mechanical Department

Ghali_1979@yahoo.com
07808924962

Abstract :

Numerical simulation plays an indispensable role in the modern manufacturing process, especially in production operation and metal forming by using different of forming Dies Because it give perfect product design to improving quality and performance and reduce the time and cost .
Because of the most of forming operation include contact option between the product surface (contact surface) and Die surface (Target surface) then the determination of the contact surface nature and material behavior (uniform linear) or (not uniform nonlinear) that very effect on the carryout of forming process with required product .
To improving simulation accuracy, approximations of linear behavior have become less acceptable, while it is possible to perform nonlinear analysis and design more often while minimizing approximations, the nonlinear analysis capabilities in general-purpose programs such as ANSYS Mechanical.
The significance of the research is into linear simulation analysis and its limitation and development to nonlinear simulation where interest with the development manufacturing processes that including change in the geometry , properties and material behavior through forming ,the search give an example of Case study with Cylindrical rolling contact that very important for wide application to manufacturing processes such as drawing ,Rolling ,Forging , Extrusion, Bending and other .The structural nonlinearities can be classified as geometric nonlinearity, material nonlinearity, and contact or boundary nonlinearity Numerical results indicate the success of ANSYS contact technologies in solving very large scale engineering problems especially large deformation frictional contact problems with the new surface based contact elements have many advantages over the node based contact elements .

المحاكاة اللاخطية للاحتكاك في عمليات التصنيع باستخدام برنامج الانسس تطبيق لدراسة حالة تماس تدوير السطوح الاسطوانية

المدرس المساعد : عمار داود غالي
ماجستير هندسة القوالب والعدد

جامعة واسط / كلية الهندسة / قسم الهندسة الميكانيكية

الخلاصة :

تعتبر المحاكاات الرقمية ذات دورمهم وحيوي في عمليات التصنيع الحديثة وخصوصا في عمليات الانتاج وتشكيل المعادن باستخدام قوالب التشكيل المتنوعة كونها توفر التصميم الامثل لتحسين نوعية المنتجات وادائها مع توفير الوقت والكلفة .

ولكون معظم عمليات التشكيل تتضمن عملية تماس (Contact) بين سطح المنتج وسطح القالب لذا فان تحديد طبيعة السطوح المتماسة وتصرف المادة بشكل منتظم خطي (Linear) او بشكل متغير لاخطي (Nonlinear) يؤثر بشكل كبير على اجراء عملية التصنيع بالشكل المطلوب .

ولتحسين دقة المحاكاات فان الدقة التقريبية للتحليل الخطي اصبحت غير مقبولة في السنوات الاخيرة ،واصبح التحليل اللاخطي اكثر استحسانا واقرب الى الواقع العملي لتصرف المواد خصوصا بعد زيادة القدرة على اجراء هذا النوع من التحليل مع تطور برامج المحاكاات مثل برنامج الانسس(ANSYS) .

لذا تكمن اهمية البحث في تحليل المحاكاات الخطية ومحدوديتها وتطورها لتتضمن المحاكاات اللاخطية التي تهتم بعمليات التشكيل المتطورة التي يحصل فيها تغير في الشكل والخواص وتصرف المادة عند التشكيل واعطى البحث دراسة لحالة تماس السطوح الاسطوانية مع السطوح الاخرى والتي تعتبر من التطبيقات الواسعة الاستعمال في عمليات التصنيع مثل عمليات السحب والدرفلة والطرق والبثق والحني وغيرها .

يمكن اعتماد البناء اللاخطيعند السلوك اللاخطي لشكل الموديل (Nonlinear Model Geometry) ، لاخطية لتصرف المادة (Material Nonlinearity)، لاخطية الظروف المحيطية للتماس(Boundary Condition B.C Nonlinearity).لقد اثبتت نتائج المحاكاات نجاح ادواتها المتمثلة ببرنامج الانسس لمعالجة العناصر المحدد (.Finite Element F.E) وخصوصا في العمليات التي تتضمن التشوية الكبير في التماس الاحتكاكيمع تطور الخصائص الحديثة لعناصر التماس لتحليل السطوح (Surface) والتي اثبتت فعاليتها مقابل الاسس المعتمدة لتحليل العقد (Node) وامكانية تمثيلها للسطوح والزوايا المشكلة ، بالإضافة الى الخواص المتعددة التي تميزها عن باقي عناصر التماس .

1-1 Introduction :

In general, engineering problems are mathematical models of physical situations , mathematical model are differential equations with a set of corresponding boundary and initial condition . The differential equations are derived by applying the fundamental laws and principles of nature to a system or a control volume ,These governing equations represent balance of mass , force , or energy . When possible ,the exact solution of these equation renders detailed behavior of a system under a given set of condition .The analytical solution are composed of two parts : Homogenous part and Particular part [1]. For engineering design and application, it is essential to understand and accurately characterize material behavior. It is a challenging, complex science. Lemaitre and Chaboche express the complexity in a dramatic manner [2].

Numerical simulation plays an indispensable role in the manufacturing process, speeding product design time while improving quality and performance. Recently, analysts and designers have begun to use numerical simulation alone as an acceptable means of validation. In many disciplines, virtual prototyping-employing numerical simulation tools based on finite element methods- has replaced traditional build-and break prototyping. Successful designs leading to better prosthetic implants, passenger safety in automotive crashes, packaging of modern electronic chips, and other advances are partly a result of accurate and detailed analysis. With the trend toward ever improving simulation accuracy, approximations of linear behavior have become less acceptable; even so, costs associated with a nonlinear analysis prohibited its wider use in the past. Today, rapid increases in computing power and concurrent advances in analysis methods have made it possible to perform nonlinear analysis and design more often while minimizing approximations. Analysts and designers now expect nonlinear analysis capabilities in general-purpose programs such as ANSYS Mechanical.

All of the following components are necessary for a reliable nonlinear analysis tool: (a) element technologies for consistent large-deformation treatment, (b) constitutive models for a variety of metals and nonmetals, (c) contact interaction and assembly analysis, (d) solution of large-scale problems (where multiple nonlinearities interact in a complex manner), and (e) infrastructure[2].

The element library of Release 5.3 (circa 1994) was diverse and comprehensive in its capabilities. A clear need existed, however, for a new generation of elements to address the growing needs of multiplicity in material models and application complexities, and to bring about a higher level of consistency.

A small set of elements was develop (the 180 series) having these characteristics:

- Rich functionality
- Consistency with theoretical foundations employing most advanced algorithms
- Architectural flexibility.

The application of conventional isoperimetric fully integrated elements is limited, In linear or nonlinear analyses, serious locking may occur. As a general analysis tool, ANSYS Mechanical uses elements in wide range of applications. The following factors influence the selection of elements:

- Structural behavior (bulk or bending deformation)
- Material behavior (nearly incompressible to fully incompressible).

The indicated factors are not necessarily limited to nonlinear analysis; however, nonlinear analysis adds to the complexity. For example, an elasto-plastic material shows distinctly different patterns of behavior in its post-yield state [2] .

In the Sheet Metal Forming (SMF) industry, especially the car manufacturing industry, a lot of effort is put into finite element (FE) simulations of processes like deep drawing, stretching and bending. The objective of this is to minimize the trial and error cost in the development phase of a product and to analyses problems during the production phase. For accurate computer simulations a good model of the process has to be available. Friction forces between sheet and forming tools play an important role because of their influence on the process performance and on the final product properties. This frictional behavior is often taken into account by using a constant coefficient of friction in the FE simulations of SMF processes. From the results of 3D simulations it was found that the friction model influences the punch force/punch displacement characteristic as well as the local strains and the strain distribution. However, the contact description of the interacting tool and sheet in FE codes has to be improved in order to take full advantage of the implemented friction model [3] .

A concise survey of the literature related to contact and friction is presented together with an extension of the friction law for large deformation analysis. Starting from the principle of virtual work, the so-called total Lagrangian and updated Lagrangian formulations are derived based on some fundamental assumptions in linearizing the nonlinear equations. By introducing the Zaremba-Jaumann (co-rotational) increment to the Cauchy stress tensor, the classical Prandtl-Reuss equations are generalized for describing the elastic-plastic material behavior. To allow a proper consideration of the contact conditions in the incremental analysis, a general friction law with an associated isotropic Coulomb sliding rule is obtained by the similarity between dry friction and plasticity. Finite element discretizations and approximations are applied to the resulting formulation of the updated Lagrangian approach. [4] .

1-2 General characteristics of contact problems

In almost every mechanical device, constituent components are in either rolling or sliding contact. In most cases, contacting surfaces are non-conforming so that the area through which the load is transmitted is very small, even after some surface deformation, and the pressures and local stresses are very high. Unless purposefully designed for the load and life expected of it, the component may fail by early general wear or by local fatigue failure. The magnitude of the damage is a function of the materials and the intensity of the applied load as well as the surface finish, lubrication, and relative motion.

The intensity of the load can usually be determined from equations, which are functions of the geometry of the contacting surfaces, essentially the radii of curvature, and the elastic constants of the materials. Large radii and smaller modules of elasticity give larger contact areas and lower pressures.

A contact is said to be conforming (concave) if the surfaces of the two elements fit exactly or even closely together without deformation. Journal bearings are an example of concave contact. Elements that have dissimilar profiles are considered to be nonconforming (convex). When brought into contact without deformation they touch first at a point, hence point contact or along line, line contact. In a ball bearing, the ball makes point contact with the inner and outer races, whereas in a roller bearing the roller makes line contact with both the races. Line contact arises when the profiles of the elements are conforming in one direction and non-conforming in the perpendicular direction. The contact area between convex elements is very small compared to the overall dimensions of the elements themselves. Therefore, the stresses are high and concentrated in the region close to the contact zone and are not substantially influenced by the shape of the elements at a distance from the contact area.[5]

Modern developments in computing have stimulated research into numerical methods to solve problems in which the contact geometry cannot be described adequately by the quadratic expressions used originally by Hertz. The contact of worn wheels and rails or the contact of conforming gear teeth with Novikov profile are the typical examples. In the numerical methods, contact area is subdivided into a grid and the pressure distribution represented by discrete boundary elements acting on the elemental areas of the grid. Usually, elements of uniform pressure are employed, but overlapping triangular elements offer some advantages. They sum to approximately linear pressure distribution and the fact that the pressure falls to zero at the edge of the contact ensures that the surfaces do not interfere outside the contact area. The three-dimensional (3D) equivalent of overlapping triangular elements is overlapping hexagonal pyramids on an equilateral triangular grid [5].

The brief Considerations in Contact Analysis can be shown in the following :

• The general goal for contact analysis is to determine : The Contacting area and Contact
stresses transmitted across contacting interface .

• Contact problems present significant difficulties

– Unknown contacting zone prior to the analysis

– Contact constraint is either active or inactive

– Friction introduces another kind of nonlinearities

• A small amount of positive or negative relative sliding can change the sign of frictional
forces/stresses completely.

• Contact finite element development covers :

– Kinematics, discretization, Inequality

– Accuracy, robustness and computational overhead

• The numerical treatment of contact problems involves:

– Formulation of geometry

– Integration of interface laws

– Vibrational formulation

– Development of algorithms

In Contact Kinematics , We can considerate the following :

• Contact problem involves a variety of geometric and kinematic situations
• Contact surface discretization
– Contact surface must be discretized because the underlying bodies discretized.
– Node-node, node-surface, surface-surface
– Smoothing provides a significant improvement in convergence behavior.
• Contact detection and searching (Global search, local search)
• Penetration/gap calculation .
In Solving Larger Assembly Models the increased computer performance in combination with efficient solver technology and parallel computing techniques has resulted in FE models which may exceed 1,000,000 elements not only for linear but also for nonlinear analysis. [6]

1-3 Material Nonlinearity

Material nonlinearities arise from the presence of time-independent behavior, such as plasticity, time-dependent behavior such as creep, and viscoelastic behavior where both plasticity and creep effects occur simultaneously. They may result in load sequence dependence and energy dissipation (irreversible structural behavior).[7]
Viscoelasticity is a nonlinear material behavior having both an elastic (recoverable) part of the deformation as well as a viscous (non-recoverable) part. Viscoelasticity model implemented in ANSYS is a generalized integration form of Maxwell model, in which the relaxation function is represented by a Prony series.
The model is more comprehensive and contains, the Maxwell, Kevin, and standard linear solid as special cases. ANSYS supports both hypo-viscoelastic and large-strain hyper viscoelasticity. The large-strain viscoelasticity implemented is based on the formulation proposed by Simo. The viscoelastic behavior is specified separately by the underlying hyper elasticity and relaxation behavior. All ANSYS hyper elasticity material models can be used with the viscoelastic option (PRONY).[2]
The application requires a robust nonlinear analysis because of these factors:

- A large (several hundred percent) strain level
- The stress-strain response of the material is highly nonlinear
- Nearly or fully incompressible behavior
- Temperature dependency
- Complex interaction of elastomeric material with adjoining regions of metal.

Nonlinear FEA allows approximate numerical solutions to a boundary value problem by solving simultaneous sets of equations with displacements, pressures, and rotations as unknowns. The experimental characterization of the material assumes a critical role. One must judiciously select a particular constitutive model among available options.

1-4 Geometric Nonlinearity

Geometric nonlinearities arise from the presence of large strain, small strains but finite displacements and/or rotations, and loss of structural stability. Large strains, over 5% may occur in rubber structures and metal forming. Slender structures such as bars and thin plates may experience large displacements and rotations with small strains. Initially stressed structures with small strains and displacements may undergo a loss of stability by buckling. Two problems are considered. The first problem involves a thin cantilever plate subjected to a point load at one of the free comers. Because the plate is thin, the resulting displacement components are in comparable order to its geometric dimensions, thus the geometry changes. This requires the stiffness matrix to be modified by accounting for the changes in the geometry. Results are compared to the solution of the same problem obtained by disregarding the nonlinear geometry effects. The second problem involves a composite plate with a circular hole subjected to compression. As the applied loading increases, the plate is expected to buckle.[7]

1-5 Equation Solvers for Nonlinear Analysis

The solver selection is a function of specific problem characteristics. Issues such as predominantly bulk or bending deformation, or material behavior being compressible and incompressible, translate into conditionality or an eigenvalue spectrum of the system matrices influencing particular choices. Other factors influence solver selection, such as the presence of a large number of constraint equations .

In Direct Solvers the ANSYS Mechanical program issues a sparse direct solver for all nonlinear problems. The sparse solver can address negative indefinite systems (common in nonlinear analysis due to stress stiffness and constitutive behavior) and Lagrange multipliers from a variety of sources (such as multipoint constraints, mixed u-P elements and contact elements). The sparse solver is applicable to real, complex, symmetric and non-symmetric systems. Non-symmetric systems are critically important for contact models with significant friction. The sparse solver is a robust choice for all forms of nonlinear analysis. The sparse solver supports parallel processing on all supported platforms. The speed-up factor on high-end servers ranges from 3 to 6. The sparse solver performs efficiently for a wide range of problems, including multiphasic applications.

Another key strength of the sparse solver is that it provides fully out-of-core, partially out-of-core or fully in-core support; therefore, the solver can handle even large industrial problems with limited computer resources (such as low memory or disk space).[2]

1-6 ANSYS Mechanical Nonlinear Analysis Support

The ANSYS Mechanical program's solution infrastructure is the common thread between the components of elements, materials, contact, and equation solvers. Together with the latest technologies implemented in kinematics, constitutive, and constraint treatment, the tools enable the efficient and accurate solution of complex problems.

ANSYS Mechanical provides automatic time stepping, requiring minimal manual intervention. The time-step size increases, decreases or holds constant based upon various convergence parameters. ANSYS Mechanical provides a status report and graphical convergence tracking. Although ANSYS, Inc. intended for the time-stepping schemes to contribute to robust analyses, they often provide the most efficient solution. The program also supports convergence enhancers such as a predictor and line search. ANSYS has arc length method to simulate nonlinear buckling, and trace complex load-displacement response when structural is not stable. Since the displacement vector and the scalar load factor are solved simultaneously, the arc-length method itself includes automatic step algorithm. With ANSYS Mechanical, it is possible to restart an analysis at any converged incremental step where restart files exist, allowing one to modify solution-control parameters and continue the analysis after encountering convergence difficulties. One can also modify loads to generate result data for a solved incremental step. Solution-control heuristics are tuned to problem-specific details and reflect years of accumulated experience, hence the sometimes conservative choices. Nevertheless, one can specify solution-control parameters manually for unrivaled flexibility.[2]

1-7 Nonlinear Structural Analysis

The nonlinear load-displacement relationship -the stress-strain relationship with a nonlinear function of stress, strain, and/or time; changes in geometry due to large displacements; irreversible structural behavior upon removal of the external loads; change in boundary conditions such as a change in the contact area and the influence of loading sequence on the behavior of the structure-requires a nonlinear structural analysis. The structural nonlinearities can be classified as geometric nonlinearity, material nonlinearity, and contact or boundary nonlinearity. The governing equations concerning large deformations are nonlinear with respect to displacements and velocities. The material behavior can be linear or nonlinear, and the boundary conditions can also exhibit nonlinearity.

Geometrical nonlinearity arising from large deformations is associated with the necessity to distinguish between the coordinates of the initial and final states of deformation, and also with the necessity to use the complete expressions for the strain components. The material can exhibit either time dependent or time-independent nonlinear behavior. Nonlinearity due to boundary conditions emerges from a nonlinear relationship between the external forces and the boundary displacements. The presence of contact conditions also leads to a nonlinear structural analysis because the extent of the contact region and the contact stresses are not known a priori. The solution to the nonlinear governing equations can be achieved through an incremental approach. The incremental form of the governing equations can be written as :[7]

K(u)Au = AP (10.1)

where K is the system stiffness matrix, u is the vector of unknowns in which Au andAP represent the unknown incremental displacement vector and the known incremental applied load vector, respectively. The solution is constructed by taking a series of linear steps in the appropriate direction in order to closely approximate the exact solution. Depending on the nature of the nonlinearity, the magnitude of each step and its direction may involve several iterations. The computational algorithms and the associated parameters must be chosen with extreme care. The solution to nonlinear problems may not be unique. Nonlinear analyses require more computational time. Therefore, when solving nonlinear static problems, it may be helpful to solve a preliminary version of the problem with no nonlinearities. The results from the linear solution may indicate mistakes in modeling, meshing, and application of boundary conditions in a shorter time frame. Also, the linear solution provides information about the regions where high stress gradients are expected, thus guiding the user to modify the mesh (make it more refined) in those regions.[7]

1-8 Contact between two adherent bodies (No Bonding Condition)

Contact between two bodies with no bonding (such as glue, solder, or weld) is a challenging problem, mainly stemming from the lack of prior knowledge of the contact regions. Another complication is that, in most of the cases, there is friction between the contacting bodies. Both of these two factors make contact analysis highly nonlinear. In addition to these, if the materials involved exhibit nonlinear material behavior or transient effects, achieving convergence becomes even more difficult.Before starting a contact analysis, the user needs to be aware of two main considerations : the difference in stiffness of the contacting bodies and the location of possible contact regions. If one of the contacting bodies is significantly stiffer than the other, then the rigid-to-flexible contact option can be used in ANSYS, resulting in a considerable reduction in computational time and, possibly, less difficulty in convergence. [7]

1-9 Cyclically loaded contacts

In such cyclically loaded contacts, the contacting surfaces are subjected to a series of deformation cycles. Examples are often seen in mechanical components or machine elements such as bearings and the contact between the rail and the wheels of a train , the elastic steady state that a material reaches during a cyclic loaded contact above the elastic limit and it can be seen after a number of cyclic plastic deformations. For elastic shakedown to occur, the maximum load is called the elastic-shakedown limit. [8]

The phenomenon will be described in four steps:

· As the contact stress exceed the yield stress of softer asperity, plastic deformation initiates.

· With the initialization of the plastic deformation protective subsurface residual stresses start to develop. At the same time the material eventually strain hardens.

· The contact area starts to increase so that the contact pressure decreases, as a result .

· Asperities come to a steady-state where the deformations are elastic, if the load is not too high.

The factors responsible for shakedown are:

1. Residual stresses being developed under the contacting surfaces.
2. Geometrical changes that lead to an increase in the conformity of the contacting bodies and decrease in the contact pressures.
3. Strain hardening that makes the material harder (increase in yield strength).

1-10 Contact Models

There are three contact models in ANSYS: node-to-node, node-to-surface, and surface-to surface.Each of these models requires different types of contact elements. The node-to-node contact model is used when the contact region is accurately known a priori and when the nodes belonging to either contact surfaces are paired, thus requiring these nodes to have same the coordinates. If a large amount of sliding between the contact surfaces is expected, node-to node contact is not suitable. The node-to-surface contact model is used when a specific point on one of the surfaces, the expected to make contact with a rather smooth surface. This model does not require accurate a priori knowledge of the contact region, and the mesh pattern on either surface does not need to be compatible. The surface-to-surface contact model is used when contact regions are not known accurately and a significant amount of sliding is expected. In this model, one of the surfaces is called the contact surface and the other, the target surface.[7]

The ANSYS Mechanical program over in the last two decades the elements CONTAC12 and ONTAC52 simulated node-to-node contact in two and three dimensions, respectively. Initially, the elements were based upon a penalty function approach and elastic Coulomb friction model. later developed the CONTAC48 and CONTAC49 node-to-surface contact elements for general contact problems. The elements allow for large sliding, either frictionless or with friction. In addition to solid mechanics, the elements support thermal analysis as well. The elements allowed one to solve highly nonlinear contact problems (for example, metal forming and rolling contact) ; however, When the elements were used for large surfaces, the visualization also suffered due to the numerous line elements generated like the models of curved surfaces .

A series of surface-to-surface contact elements (169-174) allowed one to model rigid-to-flexible surface interaction. Where the difference in penalty stiffness that selected by default in many factors (including the size of adjoining elements and the properties of underlying materials). It is not necessary to provide an absolute value of stiffness, but one may override default values by a scaling non-dimensional factor. This option is necessary in bending-dominated situations.[2]

As a result of ongoing research and development, a new contact models elements ,show in the following with their specifications : [6]

2D/3D Node-Node Contact Element CONTA178

- It is the simplest and least expensive (in terms of solution CPU) contact element available.
- Easy of use and intelligent default settings ,Chattering& penetration control, contact stiffness.
- More efficient and more stable than (Node-Surf and Surf-Surf) contact elements .
- The contact nodal forces are assumed to be directly conjugate to the nodal gaps.
- The approach can not support 3D higher order element contact .
- The approach is capable of passing so-called contact patch tests due to matching mesh pattern

Limitation

- Requires matching mesh pattern on both side of contact face .
- Requires 8 nodes hexahedron .
- Only supports small sliding and rotation .
- Does not support multi-physics contact .
- Contact results can not be visualized .

2D/3D Node-Surface Contact Element CONTA175

- The element offers many advantages of surface-surface contact elements but without the disadvantages of the 48 and 49 elements.

The primary application for the new element is edge-to-surface, corner-contact analysis.

In addition, it forms a basis for upcoming Lagrange multiplier-based contact capabilities.

• Supports dissimilar mesh on both sides of contacting surface that CONTA178 cannot handle.
• Solve contact problems when surface-surface contact elements have difficulties
• Point-surface contact & edge-surface contact
• This element is typically used only in highly specialized niche applications:
– In 3D, contact between a line (or sharp edge) and a surface.
– In 2D, contact between a node (or sharp corner) and a surface.
Limitation
• Does not support 3D higher order element contact
• Requires 8 nodes hexahedron
• Contact results can not be visualized
• The approaches is incapable of passing so-called contact patch tests.

Surface-Surface Contact Element CONTA171-174. A New Revolution Technology

• The surface-to-surface elements are the most widely used contact elements in ANSYS, due to the many advantages that they have over the other contact elements.
• Extend family of ANSYS contact elements (instead of node-node, node-surf contact) .
• Compatible with both lower order and higher order elements.
• Support large deformations with significant amounts of sliding and friction efficiently.
• Provide better contact results (easier to postprocessor contact pressure and frictional stresses).
• Intelligent default settings, Contact Wizard (easy to use).
• Metaphysics contact capability.
• Provide higher-order contact/target elements
• Mesh generation of 10 nodes tetrahedral is not an issue
• Represent curve surfaces (non-faceted)
• Provide general thermal/electric contact analysis capability
The figure (1) shows how to Represent the Target and contact surface as Contact Pair Concept.

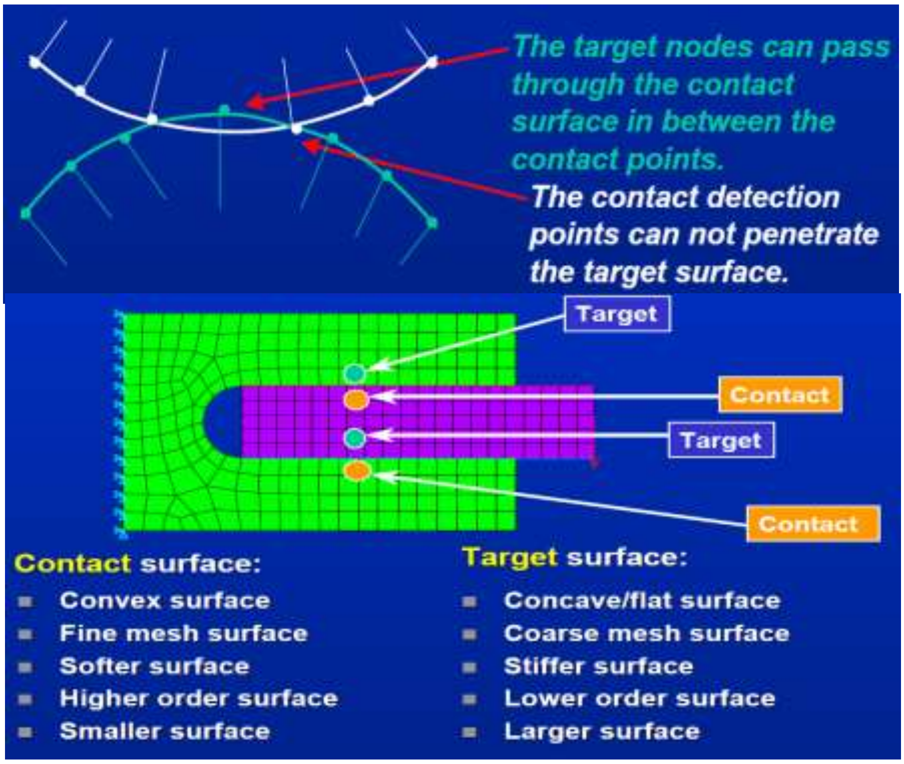


Fig (1) Contact Pair Concept

1-11 Friction

Friction is a complex physical phenomena that involves the characteristics of the surface- Surface roughness, temperature, pressure, and relative velocity. Integration of Friction Law

• The matrix is unsymmetrical which corresponds to the no associativity of Coulomb's frictional law.

• The combination of the frictional interface law with return mapping algorithm leads to a consistent matrix.

• Consistent Stiffness Matrix has a consistent and Unsymmetrical behavior where it makes system equation converge quadratically and Maintain limit pressure if sliding .[6]

1-12 Causes of Contact Chattering

Contact stiffness's are too higher: reduce the initial stiffness or redefine during load step. Often occur when the model has long, flexible parts with small contact pressures .[6]

• When only a few nodes are in contact, we should be refine the underlying mesh of the contact surface or reduce the contact stiffness to distribute the contact over more nodes.

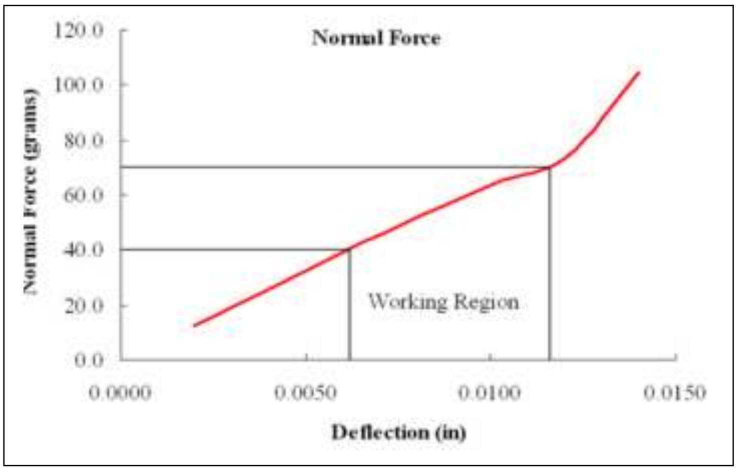
• The size of the region in contact is rapidly changing .
• The target surface is not sufficiently smooth .
• A contact node is sliding off the target surface .
- If none of the above cases seem to apply, we need to using the transient option

1-13 Linear and Non-Linear Contact Analysis

A linear FEA analysis is undertaken when a structure is expected to behave linearly. The stress is proportional to the strain, and the structure will return to its original configuration once the load has been removed. A structure is a load bearing member and can normally be classified as a bar, beam, column, or shaft. Conversely, a non-linear FEA is used to predict the behavior of a structure that is loaded beyond the elastic limits of the material of interest. The structure experiences plastic deformation and will not return to its original configuration or shape. Always perform a linear FEA on any structure whose behavior is unknown. A comparison of the Von Mises' stress level in structure and the yield strength of the choice material will determine if a non-linear FEA is required. The results of a linear FEA will be accurate and acceptable if the proper boundary conditions and meshing have been applied. The Von Mises' stress level must be below that of the yield strength of the material. Remember, stress is a function of geometry only while non-linear behavior is a function of stress and material properties. In the event that the Von Mises' stress level is greater than the yield strength of the chosen material, you have three options. First, perform a non-linear analysis to determine the degree of the plastic behavior and its suitability for the application. Second, look for a better material. Third, change your geometry to better manage the stress. Changing boundary conditions and load stiffening may also dictate the use of a non-linear FEA, but they are beyond the scope of this particular analysis. [9]

Contact Normal Force:

In contact design, the normal force is an important factor in maintaining the proper connection between two contact mating surfaces. A force deflection curve is a plot of the normal force and the contact deflection. An example in Figure (2) shows, the normal force generates a working region which is the area in which the contact provides an adequate amount of normal force for its application. In a good design, this working region should be maximized to account for the variation in contact compression in customer applications. A large working region will also compensate for tolerances in the manufacturing of the connectors. The area of deflection beyond the working region shows an increase in the normal force slope because the contact begins to interfere with the connector body.[9]



Fig(2)
An example to measured Normal force versus deflection.

1-14 Contact Finite Element Analysis:

Finite Element Analysis, or FEA, is often used in the initial design stages of a new contact. This analysis will test the contact in the computer to generate a normal force curve based off the stress-strain curve provided by the material manufacturer. To obtain effective results from a FEA study, the restraints and contact conditions must be carefully laid out. Some other important items to set up prior to running the analysis are to ensure the material for both bodies are set. The contact material should be selected according to the desired choice, and the contact pad should be set to a different material that has a much harder surface to prevent the deflection of the pad's surface from having any effect on the results. Once the material is defined, the contacts/gaps should all be defined. The global contact set must be changed to no penetration to ensure that the pusher and contact are treated as separate entities. The next step is to define a contact set between the contact and contact pad surfaces to ensure there is no penetration and no gap between the two objects. Once the contact is fully restrained and defined, the FEA can be run. Both linear and non-linear studies may be conducted, but it is important to keep in mind that the linear FEA does not account for any permanent set (or yielding) of the material. However, the linear analysis may be performed much quicker and can be used as a tool to check for any setup mistakes.

The FEA also records the reaction (or normal force) the contact generates as it is deflected. The generated values have distinct differences between the linear and nonlinear FEA methods . Figure (3) shows a graph of an example for the normal force values generated by the linear FEA, non-linear FEA, and also the calculated beam reactions, This yielding will cause permanent set and must be considered when designing the contact.[9]

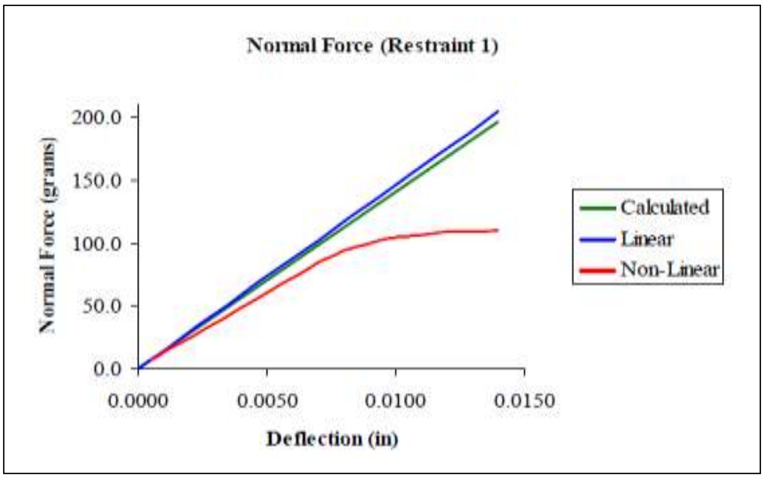


Fig (3) Normal Force and Deflection.

2- Case study with Cylindrical rolling contact :

2-1 Introduction:

Mechanical components often involve contacting surfaces. The contact can be either in the form of a dry or a lubricated sliding contact (piston ring cylinder liner, slider crank mechanisms, etc.) or a rolling contact (i.e. ball bearing: ball-ring, train: wheel-rail, etc.). Both in sliding and rolling contacts, wear (removal of material during contact of surfaces) and plastic deformation are the two critical mechanisms observed that may lead to failure of the contacting components. [8]

It is important to design and manufacture the cylindrical roller for the working performance and service life of a cylindrical bearing. The early stage of contact fatigue often occurs in the ends of the raceways and of the roller because the cylindrical bearing with the straight roller loaded inevitably is influenced by edge stresses, stress concentration, at the ends of the roller. This greatly reduces the working performance and service life of the cylindrical bearing.[10]

2-2 Contact Theory Basics

Contact between two solids can be broken into to categories. Conforming contact occurs when the mating surfaces of the two bodies have identical (or nearly identical) contours. Under normal load, the contact area is large and the pressure is distributed. Non-conforming contact occurs when two solids are brought together that will initially have an infinitesimal contact area, such as a point (sphere contacting a plane or another sphere), or a line (cylinder contacting a plane or another cylinder). The non-conforming bodies will deform under pressure and a relatively small area of contact will be created. Contact stress will be concentrated in this small area and will thus be high compared to stress in conforming contact. In practice, no surface is truly planar, cylindrical, spherical, due to manufacturing irregularities. Therefore, non-conforming contact is of the greatest interest, especially when rough surfaces are involved.[11]

2-3 Bearing Loading and Motion

Rolling element bearings are used in commercial and industrial applications to support and locate rotating shafts, and greatly reduce friction. This prolongs equipment life and can drastically cut the energy input required to keep the mechanism moving. Cylindrical roller bearings are used where high load carrying capability is required in a compact space. Cylindrical roller bearings are intended to carry radial loads (perpendicular to the shaft axis) only. The roller has a greater contact area with the race than a traditional ball bearing, to better distribute the applied load.[11]

2-4 The research object Description:

Configuration of the contact between cylinder and two blocks is shown in Figure (4,5).This is a typical contact problem, which in engineering applications is represented by a cylindrical rolling contact bearing. Also, the characteristic feature of the contact is that, nominally, surface contact takes place between elements. In reality, this is never the case due to surface roughness and unavoidable machining errors and dimensional tolerance. There is no geometrical interference when the cylinder and two blocks are assembled.[5]

This is a 3D analysis and advantage could be taken of the inherent symmetry of the model. Therefore, the analysis will be carried out on a half-symmetry model only. The objective of the analysis is to observe the stresses in the cylinder when the initial gap between two blocks is decreased by 0.05 cm. The dimensions of the model are as follows: cylinder radius=0.5 cm; cylinder length=1 cm; block length=2 cm; block width=1 cm; and block thickness=0.75 cm. Both blocks are geometrically identical. All elements are made of steel with Young's modulus (EX)=2.1×109 N/m2, Poisson's ratio (PRXY) =0.3 and are assumed elastic. Friction coefficient at the interface between cylinder and the block is 0.2.

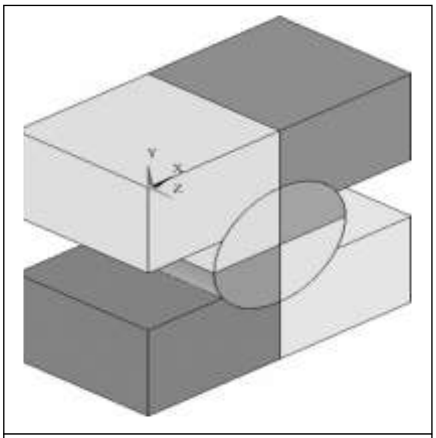


Figure (4)
cylinder and two blocks

Figure (5)
Unit of bearing pillow

2-5 MODEL CONSTRUCTION

For the intended analysis a half-symmetry model is appropriate. It is shown in Figure (6).The use of two 3D primitives, namely block and cylinder, is made. The model is constructed using GUI facilities only.

In order to create a model , We will following the steps below :

- Create the model block .
- Create cylinder and make overlap between the block and cylinder and then subtracted to get final shape of required block as shown in fig (7) .
- Create another cylinder to completed the model where the cylinder created earlier is reproduced In order to create loading conditions at the contact interface

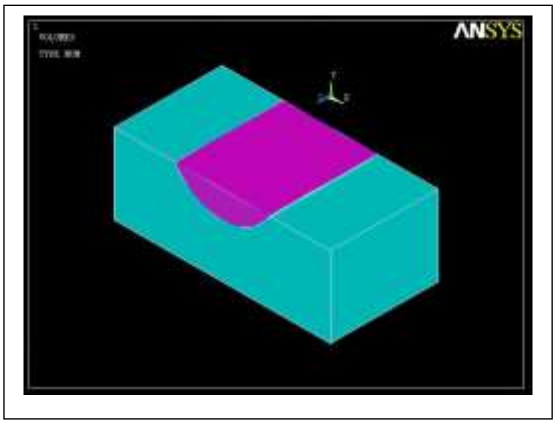


Fig (6)

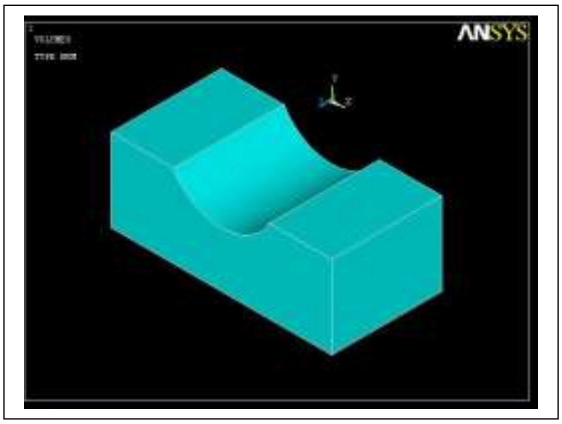


Fig (7)

2-6 Meshing

In order to reduce computing time, on one hand, the model of half roller at the bottom of the bearing that bears the maximum load is created; on the other hand, the chamfers of the inner and outer rings and the effect of radial clearance are ignored. The Solid 45 is adopted to mesh the model. The meshes of the contact part of the roller and the inner and outer rings are refined to enhance the computing accuracy. So we used smart size meshing to refine the element in the contact surface and crossing the element in the other parts,

2-7 Creation of contact pair

The creation of the contact pair and the setting of the contact parameters are crucial issues on contact analysis. Considering the inner and outer rings whose surfaces are bigger and stiffness is higher than the roller, the inner and outer rings are set as target surfaces and the roller is set as a contact surface. Then, the contact pairs are created respectively. Both the contact stiffness coefficient and the tolerance of penetration are key contact parameters. The smaller contact stiffness coefficient is favorable to be convergent. However, the bigger contact stiffness coefficient is favorable to improve the precision. By many times of calculation, the contact stiffness coefficient is set to 1.5 and the tolerance of penetration is set by a default value .The contact wizard frame show a contact pair consists of a target surface (block) and contact surface (cylinder) ,Where the target surface is represented body(volume) and flexible ,and the contact surface is represented body (volume) and element type (surface to surface) , the created contact pair is shown in figure (8) .Several setting should be doing to complete the process Include initial penetration ,coefficient of friction and stiffness matrix (unsymmetrical).

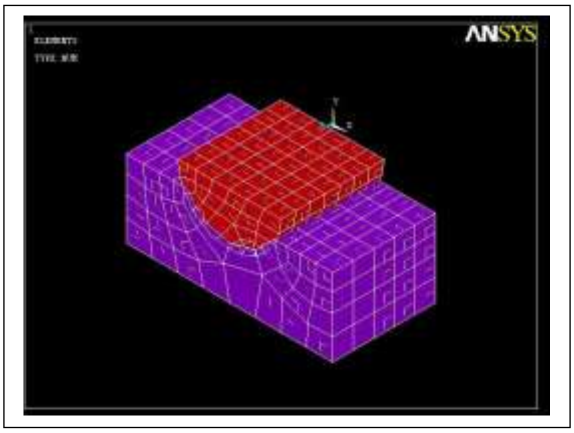


Fig (8) Contact pair created by contact wizard

2-8 SOLUTION

Before the solution process can be attempted, solution criteria have to be specified.
As a first step in that process, symmetry constraints are applied on the half-symmetry model.
From ANSYS Main Menu select : Solution→ Define Loads→ Apply→ Structural → Displacement → Symmetry BC → On Areas.
The next step is to apply constraints on the bottom surface of the block.
Form ANSYS Main Menu select : Solution→ Define Loads →Apply → Structural →Displacement→ On Areas. and selected: DOFs to be constrained=All DOF and Displacement value=0. Because the cylinder has been moved toward the block by 0.05 cm, in order to create interference load, the analysis involves a large displacement effects.

2-9 RESULTS

Through modeling, meshing, constraining, loading, and solving, the results of the contact stress of the roller are obtained by finite element analysis. The normal direction contact stress produced by the roller and the inner ring . Solution results can be displayed in a variety of forms using post processing facility. For the results to be viewed for the full model, the half-symmetry model used for analysis has to be expanded.
From Utility Menu select: Plot Ctrls→ Style → Symmetry Expansion → Periodic /Cyclic Symmetry. An image of full model, as shown in Figure (9), is

produced. The objective of the analysis presented here was to observe stresses in the cylinder produced by the reduction of the initial gap between two blocks by 0.05 cm (an interference fit). From ANSYS Main Menu select General Postproc→ Plot Results→ Contour Plot→ Nodal Solu. In the selections of Stress and to be contoured =von Mises (SEQV). Contour plot of von Mises stress for the whole assembly (nodal solution) is shown in Figure (10). If one is interested in observing contact pressure on the cylinder alone then a different presentation of solution results is required.

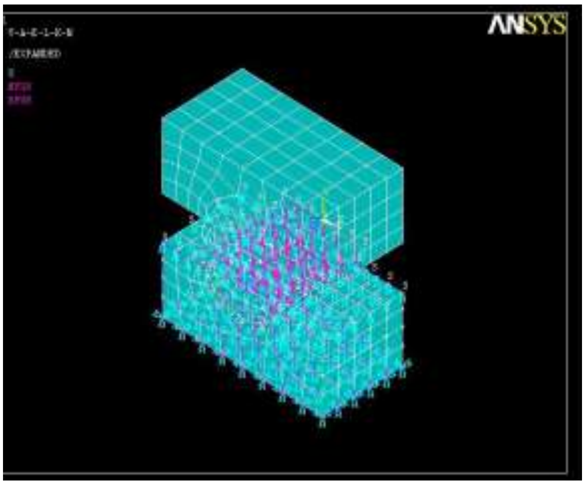


Fig (9)
Full model with mesh of elements and applied constraints

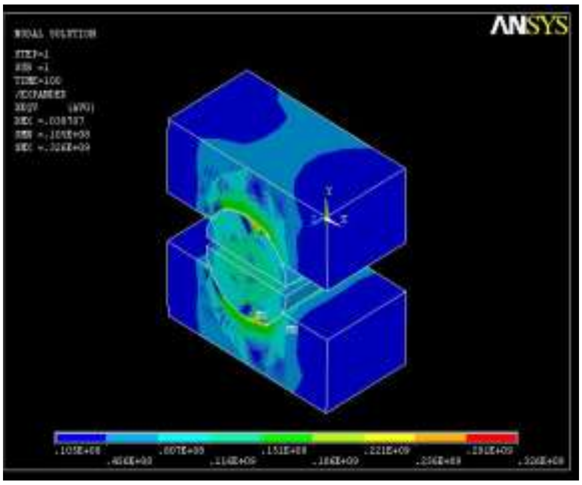


Fig (10)
Contour plot of nodal solution (vonMises stress)

the approximately uniform contact stress appears within the contact zone between the inner ring and the roller ,the uniform distribution of the contact stresses between the roller and the raceway. This can increase the service life of the cylindrical roller bearing

Conclusions

• Numerical simulation plays an indispensable role in the manufacturing process, speeding product design time while improving quality and performance .

•To improving simulation accuracy, approximations of linear behavior havebecome less acceptable, while it is possible to perform nonlinear analysis and design more often while minimizing approximations, the nonlinear analysis capabilities in general-purpose programs such as ANSYS Mechanical.

• For a reliable nonlinear analysis ,we need to used element technologies for consistent large-deformation treatment and contact interaction and assembly analysis.

• A general framework is being implemented in ANSYS for finite element treatment of multi-body, multi-filed, large deformation frictional contact problems.

• Numerical results indicate the success of ANSYS contact technologies in solving very large scale engineering problems.
• The new surface contact elements have many advantages over the node contact elements.
• Large sliding contact can generate un symmetric stiffness terms and highly curved contact and target surfaces also produce large un symmetric terms.
• The structural nonlinearities can be classified as geometric nonlinearity, material nonlinearity, and contact or boundary nonlinearity.
• There are three contact models in ANSYS: node-to-node, node-to-surface, and surface-to surface. Each of these models requires different types of contact elements.
• The surface-to-surface elements are the most widely used contact elements in ANSYS, due to the many advantages that they have over the other contact elements .
• If that the Von Mises' stress level is greater than the yield strength of the chosen material, you have three options. First, perform a non-linear analysis to determine the degree of the plastic behavior and its suitability for the application. Second, look for a better material.
Third, change the geometry to better manage the stress. Changing boundary conditions and load stiffening .
•The linear FEA does not account for any permanent set (or yielding) of the material. However, the linear analysis may be performed much quicker and can be used as a tool to check for any setup mistakes.

References

1)SaeedMoaveni , Finite Element Analysis –Theory and application with ANSYS , New Jersey , 1999 .
2) Grama R. Bhashyam1, Powerful Nonlinear Simulation Tool, ANSYS, Inc. , 2002 .
3)Rudi terHaar , Friction in Sheet Metal Forming , The influence of (local) contact conditions and deformation , Thesis , University of Twente, Netherlands , 1997
4)Jung-Ho Cheng and Noboru Kikuchi, An Analysis Of Metal Forming Processes Using Large Deformation Elastic-Plastic Formulation , The University of Michigan, . U.S.A.1984 .
5)Y. Nakasone and S. Yoshimoto , T. A. Stolarski , Engineering Analysis With ANSYS Software , Japan and UK , 2006 .
6) Mechanics Development Group , Nonlinear Contact Analysis Techniques using ANSYS .
7)ErdoganMadenci , Ibrahim Guven , The Finite Element Method and Application with In Engineering using ANSYS, The University of Arizona, 2006 .
8)Yusuf Caner Ta_an , Measturment of Deformation in Rolling and Sliding Contacts, Ph.D. Thesis , Turkey , 2005 .
9) Brian Vicich; C. Ryan; K. Meredith , Linear vs. Non-Linear Contact Analysis , Inc. 2007
10)Xintao Xia, Shichao Zhu, Study of Interval of Arc Modification Length of Cylindrical Roller Using ANSYS , International Journal of Engineering and Science , 2012, China .
11)Jeffrey A. Scarcella , Contact Stress Concentration due to Surface Irregularity in Cylindrical Rolling Element Bearings , Cornell University , 2008 .