

Modeling and Structural Analysis of Journal Bearing

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Abstract:

A journal bearing is the most straightforward sort of bearing, involving only a bearing surface and no moving components. Journal bearings, when all is said in done, are the minimum costly sort of bearing. They are likewise minimized and lightweight, and they have a high load-conveying capacity. Here the issue discover that journal bearing exposed to huge amount of loading capacity. Appropriate material is must utilized for that journal bearing If not that journal bearing part yields break. For that one, light weight and high quality material is required for assembling of journal bearing.

The point of the project is to make 3D model of Journal bearing. 3d model of journal bearing is examined for basic investigation. 3D model of Journal bearing is made utilizing Unigraphics NX-CAD. Auxiliary investigation is finished by utilizing ANSYS software. In this project auxiliary investigation of journal bearing donefor steel and Aluminum materials. From investigation results best material proposed for journal bearing.

Keywords: Journal bearing, CAD, Unigraphic , Finite Element Modelling

INTRODUCTION

1.1 JOURNAL BEARING

Bearings are utilized to avert friction between parts amid relative movement. In hardware they fall into two essential

classifications: hostile to friction or moving component bearings and hydrodynamic journal bearings. The essential capacity of a bearing is to convey stack between a rotor and the case with as meager wear as could reasonably be expected. This bearing capacity exists in pretty much every event of day by day life from the watch on your wrist to the automobile you drive to the circle drive in your PC. In industry, the utilization of journal bearings is particular for turning hardware both low and rapid. This paper will exhibit a prologue to journal bearings and oil. Grease technology runs as one with understanding journal bearings and is indispensable to bearing structure and application. Since they have noteworthy damping liquid film journal bearings strongly affect the vibration qualities of apparatus. The sorts of hardware we are worried about range from little rapid shafts to engines, blowers, blowers, fans, and siphons to expansive turbines and generators to some paper process rolls and other huge moderate speed rotors. Not secured here is the point of bearings for responding apparatus. While a portion of similar principals apply, motor bearings have uncommon needs and plan contemplations and merit a more entire investigation. Responding hardware bearings will in general be more straightforward in geometry and substantially more entangled in application than turbo apparatus bearings. For instance, the commonplace turbo apparatus journal bearing comprises of a thin layer of babbitt

on steel while an associating pole bearing may have various diverse layers of copper, steel, nickel, or different metals with a thin layer of babbitt to finish everything. This layering is improved the situation weakness protection from the beating loads experienced in such hardware. Motor bearings are often required to withstand crest particular loads more than 3,000 PSI or around multiple times an ordinary engine or turbine bearing. Responding machines depend fundamentally on the pressing of the oil film for load bolster.

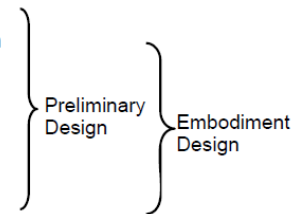
A journal bearing is the easiest kind of bearing, including only a bearing surface and no moving components. Journal bearings, all in all, are the slightest costly kind of bearing. They are likewise smaller and lightweight, and they have a high load-conveying capacity.

COMPUTER AIDED DESIGN (CAD):

Computer aided design is a critical mechanical craftsmanship broadly utilized in numerous applications, including car, shipbuilding, and aviation ventures, modern and engineering structure, prosthetics, and some more. Computer aided design is likewise generally used to create PC movement for enhancements in films, promoting and specialized manuals. The cutting edge pervasiveness and intensity of PCs implies that even fragrance containers and cleanser allocators are planned utilizing strategies incredible by designers of the 1960s. As a result of its tremendous monetary significance, CAD has been a noteworthy main thrust for research in computational geometry, PC illustrations (both equipment and software), and discrete differential geometry.

Current PC supported plan software bundles run from 2D vector-based drafting frameworks to 3D strong and surface modelers. Present day CAD bundles can

likewise much of the time permit revolutions in three measurements, permitting review of a structured question from any coveted edge, even from within watching out. Some CAD software is equipped for dynamic numerical modeling, in which case it might be advertised as CADD.

- Product need or requirement
 - Design Specifications
 - Conceptual Design
 - Configuration Design
 - Parametric Design
 - Formulate
 - Alternated Design
 - Analytical Design
 - Evaluate
 - Refine Optimize
 - Detailed Design
 - Manufacturing
 - Marketing
- 

Computer aided design is utilized in the plan of instruments and apparatus and in the drafting and plan of a wide range of structures, from little private sorts (houses) to the biggest business and modern structures (doctor's facilities and manufacturing plants). Computer aided design is for the most part utilized for itemized building of 3D models and additionally 2D illustrations of physical segments, however it is likewise utilized all through the building procedure from reasonable structure and format of items, through quality and dynamic examination of congregations to meaning of assembling strategies for segments. It can likewise be utilized to configuration objects.

Computer aided design has additionally been turned out to be helpful to engineers too. Utilizing four properties which are history, highlights, parameterization, and abnormal state imperatives (Zhang). The development history can be utilized to think once again into the model's close to home highlights and work on the single territory instead of the entire model . Parameters and requirements can be utilized to decide the

size, shape, and the distinctive modeling components. The highlights in the CAD framework can be utilized for the assortment of devices for estimation, for example, elasticity and yield quality, additionally its anxiety and how the component gets influenced in specific temperatures.

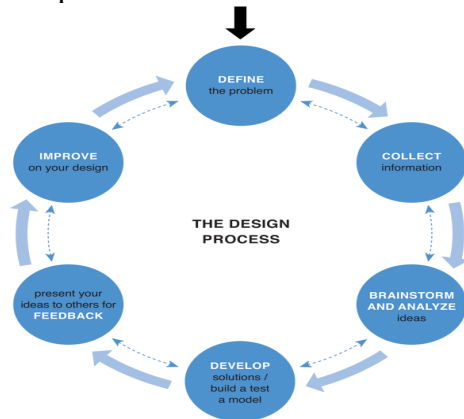


fig shows design process of product
UNIGRAPHICS INTRODUCTION:

NX, otherwise called NX unigraphics or typically just u-g, is a propelled CAD/CAM/CAE software bundle created by Siemens PLM software. It is utilized, among different assignments, for:

- Design (parametric and coordinate strong/surface modeling)
- Engineering examination (static, dynamic, electro-magnetic, warm, utilizing the limited component technique, and liquid utilizing the limited volume strategy).
- Manufacturing completed design by utilizing included machining modules

First arrival of the new "people to come" rendition of unigraphics and I-deas, called NX. this will in the end bring the usefulness and capacities of both unigraphics and I-deas together into a solitary merged item.

Expanding multifaceted nature of items, advancement processes and design groups is testing organizations to discover new devices and techniques to convey more prominent development and higher quality at lower cost. driving edge technology from Siemens PLM software conveys more noteworthy power for the present design test. from imaginative synchronous technology that joins parametric and sans history modeling, to NX dynamic mockup for multi-scoundrel gathering design, NX conveys achievement technology that sets new norms for speed, execution, and usability.

NX robotizes and streamlines design by utilizing the item and process learning that organizations gain as a matter of fact and from industry best practices. it incorporates apparatuses that designers can use to catch information to mechanized monotonous undertakings. the outcome is diminished expense and process duration and enhanced quality.

Review of Solid Modeling:

The Unigraphics NX Modeling application gives a strong modeling framework to empower quick theoretical design. Architects can join their prerequisites and design confinements by characterizing numerical connections between various parts of the design.

Design architects can rapidly perform calculated and point by point designs utilizing the Modeling highlight and limitation based strong modeler. They can make and alter mind boggling, reasonable, strong models intuitively, and with far less exertion than more customary wire edge and strong based frameworks. Highlight Based strong modeling and altering abilities enable designers to change and refresh strong

bodies by specifically altering the components of a strong element as well as by utilizing other geometric altering and development methods.

Points of interest of Solid Modeling:

Strong Modeling raises the dimension of articulation with the goal that designs can be characterized as far as building highlights, instead of lower-level CAD geometry. Highlights are parametrically characterized for measurement driven altering dependent on size and position.

Highlights:

Powerful inherent building focused frame highlights spaces, openings, cushions, supervisors, pockets-catch design plan and increment profitability

Patterns of highlight occasions rectangular and round exhibits with uprooting of individual highlights; all highlights in the example are related with the ace component.

Mixing and Chamfering:

- zero sweep
- Ability to chamfer any edge
- Cliff-edge mixes for designs that can't oblige finish mix range yet at the same time require mixes

Propelled Modeling Operations:

- Profiles can be cleared, expelled or spun to frame solids
- Extremely incredible empty body order transforms solids into thin-walled designs like a flash; inward divider topology will contrast from the external divider, if vital
- Fixed and variable range mixes may cover encompassing countenances and

stretch out to a Tapering for modeling produced close net shape parts

- User-characterized highlights for basic design components (Unigraphics NX/User-Defined Features is required to characterize them ahead of time

General Operation:

Begin with a Sketch

Utilize the Sketcher to freehand a portray, and measurement a "plot" of Curves. You would then be able to clear the portray utilizing Extruded Body or Revolved Body to make a strong or sheet body. You can later refine the portray to correctly speak to the question of enthusiasm by altering the measurements and by making connections between geometric articles. Altering an element of the portray alters the geometry of the outline, as well as the body made from the draw.

Making and Editing Features:

Highlight Modeling gives you a chance to make highlights, for example, gaps, spaces and notches on a model. You can then specifically alter the elements of the component and find the element by measurements. For instance, a Hole is characterized by its width and length. You can straightforwardly alter these parameters by entering new qualities. You can make strong groups of any coveted design that can later be characterized as a frame highlight utilizing User Defined Features. This gives you a chance to make your very own custom library of shape highlights.

Associativity:

Cooperatively is a term that is utilized to show geometric connections between individual segments of a model. These connections are built up as the designer

utilizes different capacities for model creation. In an affiliated model, imperatives and connections are caught consequently as the model is created. For instance, in an affiliated model, a through opening is related with the appearances that the gap enters. In the event that the model is later changed so either of those faces moves, the opening updates consequently because of its relationship with the countenances. See Introduction to Feature Modeling for extra points of interest.

Situating a Feature:

Inside Modeling, you can position a component with respect to the geometry on your model utilizing Positioning Methods, where you position measurements. The component is then connected with that geometry and will keep up those affiliations at whatever point you alter the model. You can likewise alter the situation of the element by changing the estimations of the situating measurements.

Reference Features:

You can make reference highlights, for example, Datum Planes, Datum Axes and Datum CSYS, which you can use as reference geometry when required, or as development gadgets for different highlights. Any element made utilizing a reference highlight is related to that reference include and holds that relationship amid alters to the model. You can utilize a datum plane as a source of perspective plane in building draws, making highlights, and situating highlights. You can utilize a datum pivot to make datum planes, to put things concentrically, or to make spiral examples.

Articulations:

The Expressions device gives you a chance to consolidate your prerequisites and design

limitations by characterizing numerical connections between various parts of the design. For instance, you can characterize the stature of a manager as multiple times its breadth, with the goal that when the distance across changes, the tallness changes too.

Boolean Operations:

Modeling gives the accompanying Boolean Operations: Unite, Subtract, and Intersect. Join consolidates bodies, for instance, joining two rectangular squares to frame a T-formed strong body. Subtract expels one body from another, for instance, expelling a barrel from a square to shape a gap. Cross makes a strong body from material shared by two strong bodies. These activities can likewise be utilized with free frame highlights called sheets.

Fix:

You can restore a design to a past express any number of times utilizing the Undo work. You don't need to take a lot of time ensuring every task is totally right, in light of the fact that a mix-up can be effortlessly fixed. This opportunity to effortlessly change the model gives you a chance to stop agonizing over failing to understand the situation, and liberates you to investigate more potential outcomes to hit the nail on the head.

Extra Capabilities:

Different Unigraphics NX applications can work straightforwardly on strong articles made inside Modeling with no interpretation of the strong body. For instance, you can perform drafting, designing examination, and NC machining capacities by getting to the fitting application. Utilizing Modeling, you can design a total, unambiguous, three dimensional model to depict a question.

You can separate an extensive variety of physical properties from the strong bodies, including mass properties. Shading and shrouded line abilities enable you to picture complex congregations. You can recognize obstructions naturally, dispensing with the need to endeavor to do as such physically. Concealed edge perspectives can later be produced and set on illustrations. Completely acquainted dimensioned illustrations can be made from strong models utilizing the proper alternatives of the Drafting application. On the off chance that the strong model is altered later, the illustration and measurements are refreshed naturally.

Parent/Child Relationships:

If a feature depends on another object for its existence, it is a child or dependent of that object. The object, in turn, is a parent of its child feature. For example, if a HOLLOW (1) is created in a BLOCK (0), the block is the parent and the hollow is its child. A parent can have more than one child, and a child can have more than one parent. A feature that is a child can also be a parent of other features. To see all of the parent-child relationships between the features in your work part, open the Part Navigator.

Creating a Solid Model:

Modeling provides the design engineer with intuitive and comfortable modeling techniques such as sketching, feature based modeling, and dimension driven editing. An excellent way to begin a design concept is with a sketch. When you use a sketch, a rough idea of the part becomes represented and constrained, based on the fit and function requirements of your design. In this way, your design intent is captured. This ensures that when the design is passed down to the next level of engineering, the basic requirements are not lost when the design is edited.

The strategy you use to create and edit your model to form the desired object

depends on the form and complexity of the object. You will likely use several different methods during a work session. The next several figures illustrate one example of the design process, starting with a sketch and ending with a finished model. First, you can create a sketch "outline" of curves. Then you can sweep or rotate these curves to create a complex portion of your design.

Introduction to Drafting:

The Drafting application is designed to allow you to create and maintain a variety of drawings made from models generated from within the Modeling application. Drawings created in the Drafting application are fully associative to the model. Any changes made to the model are automatically reflected in the drawing. This Associativity allows you to make as many model changes as you wish. Besides the powerful Associativity functionality, Drafting contains many other useful features including the following:

- An intuitive, easy to use, graphical user interface. This allows you to create drawings quickly and easily.
- A drawing board paradigm in which you work "on a drawing." This approach is similar to the way a drafter would work on a drawing board. This method greatly increases productivity.
- Support of new assembly architecture and concurrent engineering. This allows the drafter to make drawings at the same time as the designer works on the model.
- The capability to create fully associative cross-sectional views with automatic hidden line rendering and crosshatching.
- Automatic orthographic view alignment. This allows you to quickly place views on a drawing, without having to consider their alignment.
- Automatic hidden line rendering of drawing views.
- The ability to edit most drafting objects (e.g., dimensions, symbols, etc.) from the graphics window. This allows you to

create drafting objects and make changes to them immediately.

- On-screen feedback during the drafting process to reduce rework and editing.
- User controls for drawing updates, which enhance user productivity.

Finally, you can add form features, such as chamfers, holes, slots, or even user defined features to complete the object.

Updating Models:

A model can be updated either automatically or manually. Automatic updates are performed only on those features affected by an appropriate change (an edit operation or the creation of certain types of features). If you wish, you can delay the automatic update for edit operations by using the Delayed Update option. You can manually trigger an update of the entire model. You might, for example, want to use a net null update to check whether an existing model will successfully update in a new version of Unigraphics NX before you put a lot of additional work into modifying the model. (A net null update mechanism forces a complete update of a model, without changing it.)

The manual methods include:

- The Unigraphics NX Open C and C++ Runtime function, UF_MODEL. Update all features, which logs all the features in the current work part to the Unigraphics NX update list, and then performs an update. See the Unigraphics NX Open C and C++ Runtime Reference Help for more information.
- The Playback option on the Edit Feature dialog, which recreates the model, starting at its first feature. You can step through the model as it is created one feature at a time, move forward or backward to any feature, or trigger an update that continues until a failure occurs or the model is complete.

The Edit during Update dialog, which appears when you choose Playback, also includes options for analyzing and editing features of the model as it is recreated (especially useful for fixing problems that caused update failures). Methods that users have tried in the past that has led to some problems or is tricky to use:

- One method uses the Edit Feature dialog to change the value of a parameter in each root feature of a part, and then change it back before leaving the Edit Feature dialog. This method produces a genuine net null update if used correctly, but you should ensure that you changed a parameter in every root feature (and that you returned all the parameters to their original values) before you trigger the update.
- Another method, attempting to suppress all of the features in a part and then unsuppressed them, can cause updates that are not net null and that will fail. The failures occur because not all features are suppressible; they are left in the model when you try to suppress all features. As the update advances, when it reaches the point where most features were suppressed, it will try to update the features that remain (this is like updating a modified version of the model). Some of the "modifications" may cause the remaining features to fail. For these reasons, we highly recommend that you do not attempt to update models by suppressing all or unsurprising all features. Use the other options described here, instead.

Assemblies concepts:

Components:

Assembly part files point to geometry and features in the subordinate parts rather than creating duplicate copies of those objects at each level in the assembly. This technique not only minimizes the size of assembly parts files, but also provides

high levels of Associativity. For example, modifying the geometry of one component causes all assemblies that use that component in the session to automatically reflect that change. Some properties, such as translucency and partial shading (on the Edit Object Display dialog), can be changed directly on a selected component. Other properties are changed on selected solids or geometry within a component. Within an assembly, a particular part may be used in many places. Each usage is referred to as a component and the file containing the actual geometry for the component is called the component part.

Top-down or Bottom-up Modeling:

You are not limited to any one particular approach to building the assembly. You can create individual models in isolation, and then later add them to assemblies (bottom-up), or you can create them directly at the assembly level (top-down). For example, you can initially work in a top-down fashion, and then switch back and forth between bottom-up and top-down modeling.

Design in Context:

When the displayed part is an assembly, it is possible to change the work part to any of the components within that assembly (except for unloaded parts and parts of different units). Geometry features, and components can then be added to or edited within the work part. Geometry outside of the work part can be referenced in many modeling operations. For example, control points on geometry outside of the work part can be used to position a feature within the work part. When an object is designed in context, it is added to the reference set used to represent the work part.

Associativity Maintained:

Geometric changes made at any level within an assembly result in the update of associated data at all other levels of affected assemblies. An edit to an individual piece part causes all assembly drawings that use that part to be updated appropriately. Conversely, an edit made to a component in the context of an assembly results in the update of drawings and other associated objects (such as tool paths) within the component part. See the next two figures for examples of top-down and bottom-up updates.

Mating Conditions:

Mating conditions let you position components in an assembly. This mating is accomplished by specifying constraint relationships between two components in the assembly. For example, you can specify that a cylindrical face on one component is to be coaxial with a conical face on another component. You can use combinations of different constraints to completely specify a component's position in the assembly. The system considers one of the components as fixed in a constant location, then calculates a position for the other component which satisfies the specified constraints. The relationship between the two components is associative. If you move the fixed component's location, the component that is mated to it also moves when you update. For example, if you mate a bolt to a hole, if the hole is moved, the bolt moves with it.

Using Reference Sets to Reduce the Graphic Display:

Large, complex assemblies can be simplified graphically by filtering the amount of data that is used to represent a given component or subassembly by using reference sets. Reference sets can be used to drastically reduce (or even totally eliminate) the graphical representation of portions of the assembly without modifying the actual assembly structure or underlying geometric models. Each component can use a different

reference set, thus allowing different representations of the same part within a single assembly. The figure below shows an example of a bushing component used twice in an assembly, each displayed with a different reference set. When you open an assembly, it is automatically updated to reflect the latest versions of all components it uses. Load Options lets you control the extent to which changes made by other users affect your assemblies. Drawings of assemblies are created in much the same way as piece part drawings. You can attach dimensions, ID symbols and other drafting objects to component geometry. A parts list is a table summarizing the quantities and attributes of components used in the current assembly. You can add a parts list to the assembly drawing along with associated callout symbols, all of which are updated as the assembly structure is modified. See the following figure.

Machining of Assemblies:

Assembly parts may be machined using the Manufacturing applications. An assembly can be created containing all of the setup, such as fixtures, necessary to machine a particular part. This approach has several advantages over traditional methods:

- It avoids having to merge the fixture geometry into the part to be machined.
- It lets the NC programmer generate fully associative tool paths for models for which the programmer may not have write access privilege.

It enables multiple NC programmers to develop NC data in separate files simultaneously.

LITERATURE SURVEY

PRASANNA

SUBBARAO

BHAMIDIPATI: has written a title on " FEA ANALYSIS OF NOVEL DESIGN OF CYLINDRICAL ROLLER BEARING "

and explained that When a bearing is properly designed, manufactured, installed, and maintained, then the natural cause of bearing failure is typically the fatigue life of its rolling elements and races. The environment within which the bearing operates also determines the bearing life. The contact stresses developed in the rolling elements and races of a typical bearing is cyclic in nature. This in turn will result in a potential fatigue failure for these elements. The fatigue life a bearing is influenced by the operating speed, load conditions, bearing material, clearance of the mating parts, contact surface geometry, and the environment in which the bearing operates.

R. J. Kleckner J. Pirvics: have published a paper on " HI GH SPEED CYLINDRICAL RO LLER BEARING ANALYS IS". This paper presents that Engine shaft speeds will be increased ,to derive greater compressor efficiency. Increased speeds, lj.owever, accentuate centrifugal effects. Dominance in raceway loading is transformed from inner to outer rings. Failure to maintain loaded contact on the inner ring across thus spectrum of operating speeds results in the increased hazard of roller skidding, unnecessary heat generation and unstable performance. Combined with increased shaft diameters, required by system stiffnElss, increased speed raises questions beyond the limits of currently available bearing design analysis.

B. Ramu, V. V. R. Murthy: have published a project entitled " Contact Analysis of Cylindrical Roller Bearing Using Different Roller Profiles". According to this, Cylindrical roller bearings are designed to carry heavy radial loads, but due to misalignment and edge loading it is affecting the life of the bearing. So in the design of cylindrical roller bearings the profile of the roller plays important roll. Stress analysis is done based on two dimensional models of the roller and raceways. The roller profiles analyzed are

flat, circular and logarithmic, which is loaded against two flat raceways.

Goutam Mukhopadhyay, S. Bhattacharya: have published a journal on " Failure Analysis of a Cylindrical Roller Bearing from a Rolling Mill". According to this project, Premature failure of a cylindrical roller bearing of a gear box input shaft from a hot strip mill has been investigated. The pins of the cylindrical rollers of the bearing broke from the welded joints at their ends on the cage ring. Investigations were carried out on the failed roller pin and the welded joint. The investigation consists of visual observation, chemical analysis, characterization of macro- and microstructures, measurement of hardness profile, fractography, and energy dispersive spectroscopy (EDS). The fracture surface of the roller pins exhibits beach marks. Scanning electron microscopy (SEM) of the fracture surfaces reveals striations suggesting fatigue failure.

Mr.B.J.Jugale, Mr.M.V.Kharade: have written a paper on "A Review Paper on Failure Analysis of Cylindrical Roller Bearing". This paper represents that Cylindrical roller bearings are commonly used in aircraft power transmission, machine tools, steel industries and others due to their precision and high load carrying capacity. In present study, an experimental test rig has been developed to investigate the wear in roller bearing. The changes in micro-geometry of the bearing components have been observed through scanning electron microscope. Some major cracks have been found over the inner race surface. Solid debris particles, removed from the inner race surface, further damage the other bearing surfaces. The lubricating grease has been also failed to provide elasto hydrodynamic lubrication between the inner race and rollers.

PROBLEM DEFINITION AND METHODOLOGY

PROBLEM DEFINITION:

Journal bearing carry heavy radial loads. These are suited for low coefficient of friction and less frictional loss in high speed application. Due to this reason 1 bearing generates more stresses on component. Optimization of stress depend on materials suited for bearing, accurate design of journal bearing and radial force acting on it.

METHODOLOGY:

1. Modelling of journal bearing will pass through NX CAD software based.
2. Designed journal bearing(.prt format) was exported in parasolid format (.xt format). imported in Ansys11.0 software.
3. Parasolid format (.xt format) imported in Ansys software.
4. Static analysis of journal bearing was done under maximum radial load act on it.
5. Stresses are estimated from static analysis.
6. Static analysis of journal bearing done for steel and Aluminium materials.
7. Results obtained from the analysis are compared and the best material is proposed based on the ratio of resultant (von mises) strength too yield strength(FOS).

3D MODELLING OF JOURNAL BEARING

STEPS INVOLVED IN 3d MODELING:

A 2D drawing is used to design a 3D model for our component using Unigraphics NX 7.5 CAD software.

Below image shows the 2D drawings of the journal bearing with all the required dimensions and GD&T representations the suits the best for manufacturing the component without any errors.

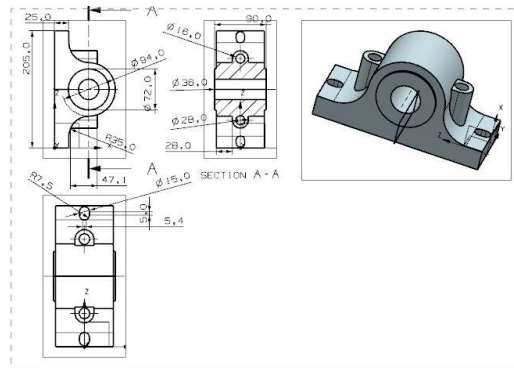


Fig4.1: 2D drawing of journal bearing
**DEVELOPMENT OF 3D MODELING:
Sketching**

Below is the sketch required to obtain the 3D model of the journal bearing from the above 2D drawing input.

Below image shows 2D sketch of journal bearing

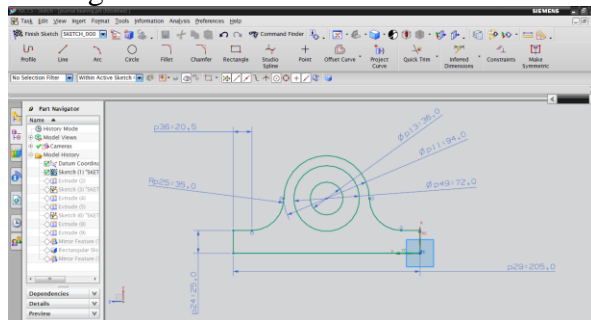


Fig4.2: 2D sketch of journal bearing
Procedure to draw the above sketch
Insert → sketch in task environment
select plane ok.
insert → curve → profile.

Below image shows extrude of sketch of journal bearing

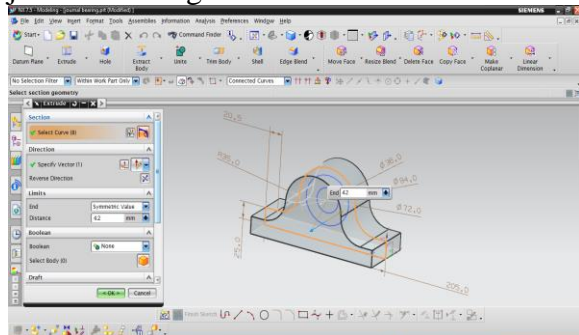


Fig4.3: Extrude of sketch of journal bearing
EXTRUDE

➤ Extrude command is used to create a body by sweeping a 2D or 3D section of curves, Edges, sketches in a specified Direction.

Insert → design features → extrude.
Select curve → specify vector → Boolean operation (none) ok.

Below image shows creating of hole in journal bearing

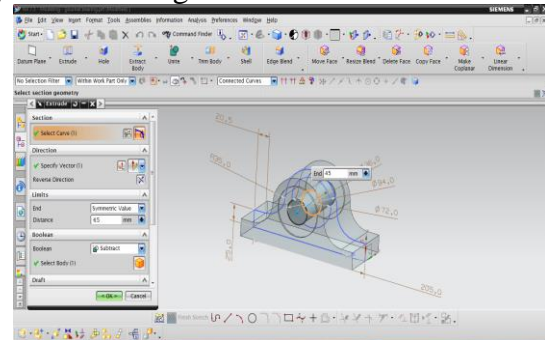


Fig4.4: Creating of hole in journal bearing
EXTRUDE

➤ Extrude command is used to create a body by sweeping a 2D or 3D section of curves, Edges, sketches in a specified Direction.

Insert → design features → extrude.
Select curve → specify vector → Boolean operation (subtract) ok

Below image shows sketch on face of journal bearing

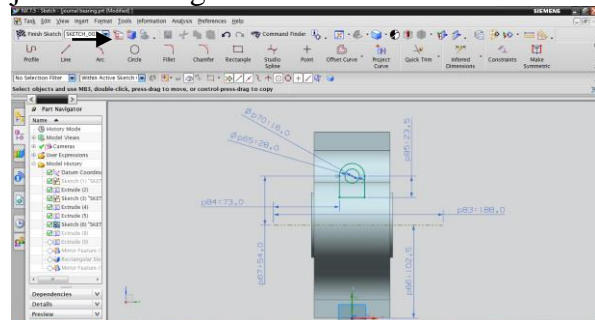


Fig4.5: Sketch on face of journal bearing
Procedure to draw the above sketch
Insert → sketch in task environment
select plane ok.
insert → curve → profile.
Below image shows extrude of above sketch on journal bearing

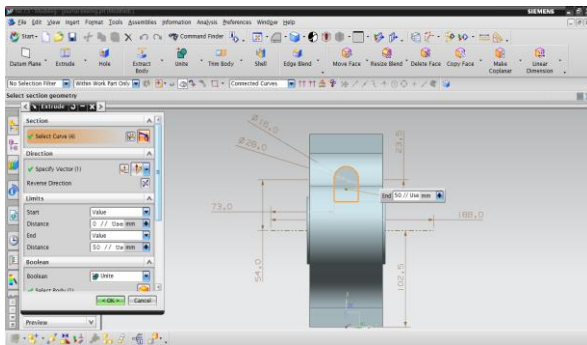


Fig4.6: Extrude of sketch on face of journal bearing

EXTRUDE

➤ Extrude command is used to create a body by sweeping a 2D or 3D section of curves, Edges, sketches in a specified Direction.

Insert → design features → extrude.

Select curve → specify vector → Boolean operation (unite) ok.

Below image shows creating rectangular slot on journal bearing

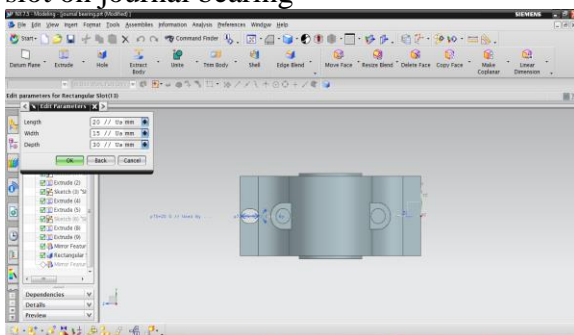


Fig4.7: Creating rectangular slot on journal bearing

Below image shows final 3D model of journal bearing

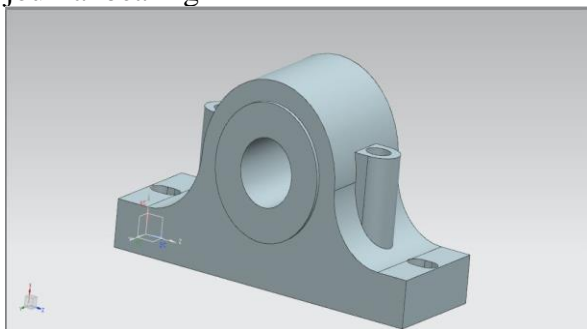


Fig4.8: 3D model of journal bearing

FINITE ELEMENT ANALYSIS OF JOURNAL BEARING

STRUCTURAL ANALYSIS OF JOURNAL BEARING

Finite Element Modelling (FEM) and Finite Element Analysis (FEA) are two most popular mechanical engineering applications offered by existing CAE systems. This is attributed to the fact that the FEM is perhaps the most popular numerical technique for solving engineering problems. The method is general enough to handle any complex shape of geometry (problem domain), any material properties, any boundary conditions and any loading conditions. The generality of the FEM fits the analysis requirements of today's complex engineering systems and designs where closed form solutions are governing equilibrium equations are not available. In addition it is an efficient design tool by which designers can perform parametric design studying various cases (different shapes, material loads etc.) analyzing them and choosing the optimum design.

FINITE ELEMENT METHOD

The FEM is numerical analysis technique for obtaining approximate solutions to wide variety of engineering problems. The method originated in the aerospace industry as a tool to study stresses in complicated airframe structures. It grew out of what was called the matrix analysis method used in aircraft design. The method has gained popularity among both researchers and practitioners and after so many developments codes are developed for wide variety of problems.

DESCRIPTION:

ANSYS is a Finite Element Analysis (FEA) code widely used in the Computer Aided Engineering (CAE) field. ANSYS software allow to construct computer models of structures, machine components or systems, apply operating loads and other design criteria and study physical responses, such as stress levels, temperature distributions, pressure, etc. The ANSYS program has a variety of design analysis

applications, ranging from automobiles to such highly sophisticated systems as aircraft, nuclear reactor containment buildings and bridges. There are 250+ elements derived for various applications in ANSYS. In the present application shell, beam and mass elements that have structural static and dynamic analysis capabilities were considered.

FINITE ELEMENT MODELING:

3D model of the journal bearing was developed in NX-CAD. The model was then converted into a parasolid to import into ANSYS. A Finite Element model was developed with solid elements. The elements that are used for idealizing the journal bearing are described below. A detailed Finite Element model was built with solid elements to idealize all the components of the journal bearing. Pre stressed modal analysis was carried out to find the natural frequencies and their mode shapes. The elements that are used for idealizing the journal bearing are solid 92.

6.1 STATIC ANALYSIS OF JOURNAL BEARING USING STEEL:

Material properties of steel:

- Young modulus(E) = **200GPa**
- Density (ρ) = **7850 kg/m³**
- Yield strength = **275MPa**
- Poison's ratio = **0.25**

Boundary (Loading) conditions:

- Bottom area of journal bearing is arrested in all DOF.
- Maximum load 2000N which is observed from shaft standard values is applied at center hole of journal bearing.

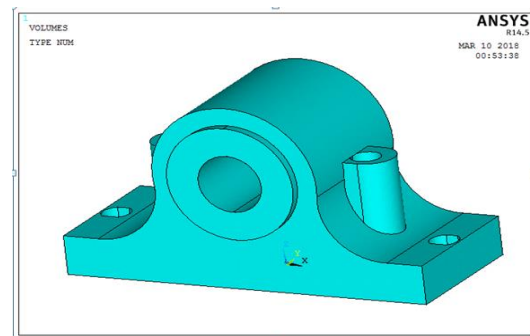


Fig6.1: shows the imported model of journal bearing

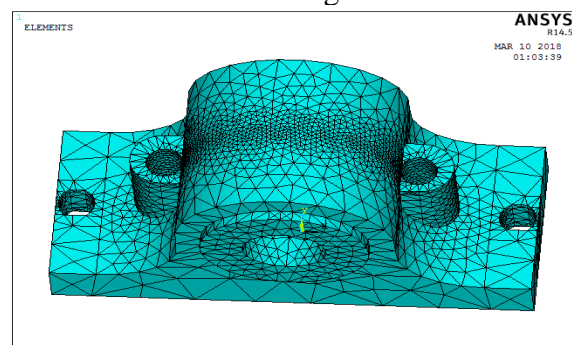


Fig 6.2: shows the imported model of journal bearing

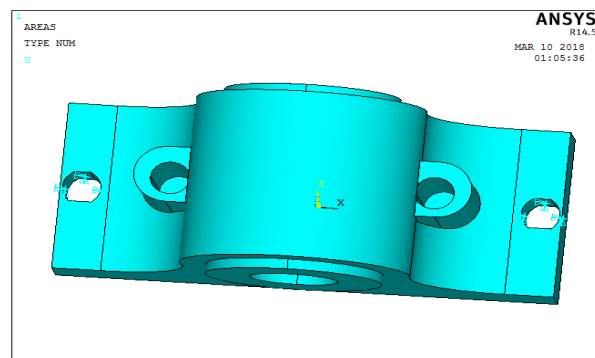


Fig 6.3: shows the fixing of journal bearing

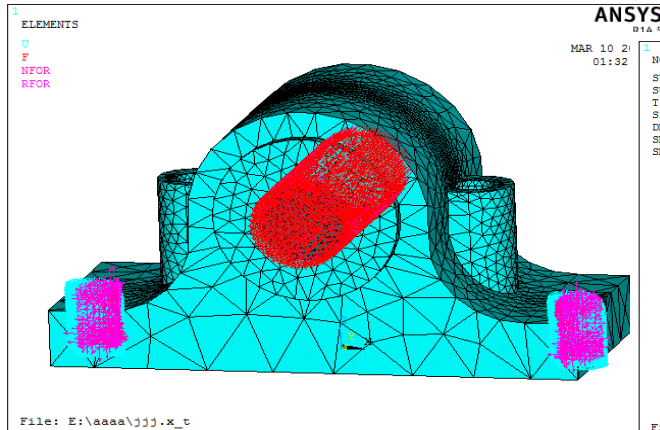


Fig 6.4: shows the loading of journal bearing

Fig 6.6: shows the 1st principal stress of journal bearing

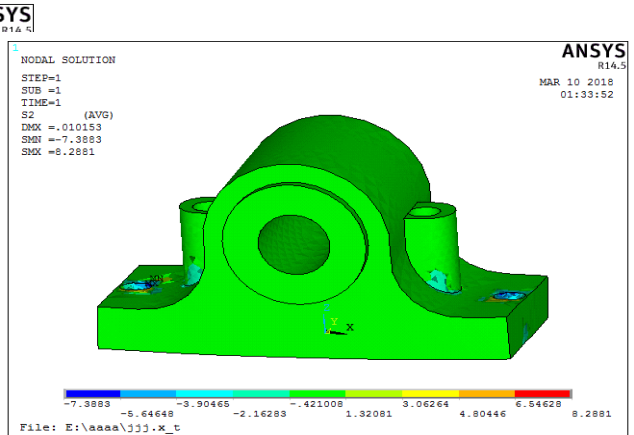


Fig 6.7: shows the 2nd principal stress of journal bearing

RESULTS

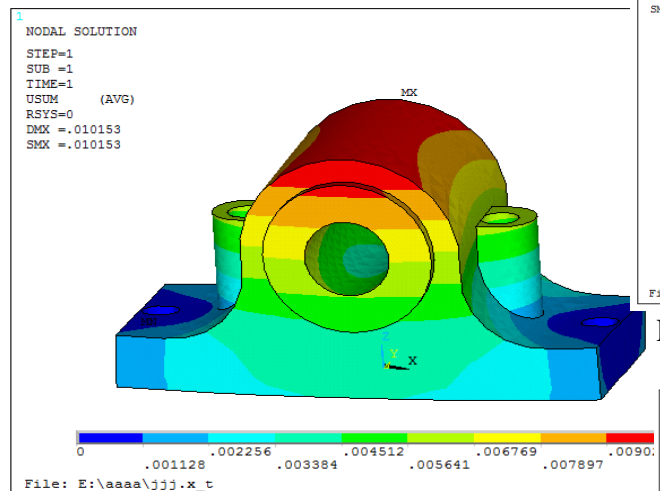


Fig 6.5: shows the displacement of journal bearing

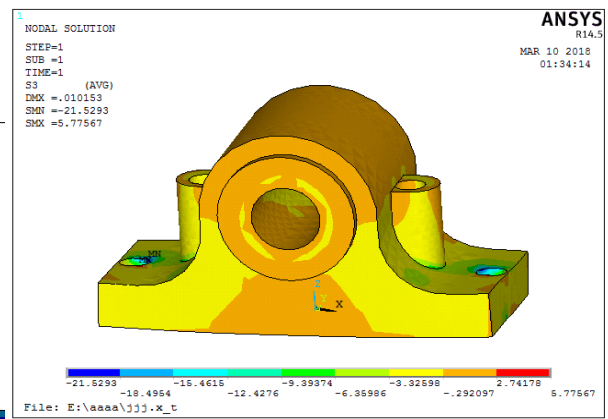


Fig 6.8: shows the 3rd principal stress of journal bearing

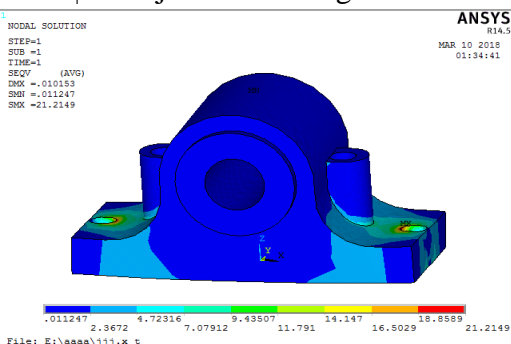
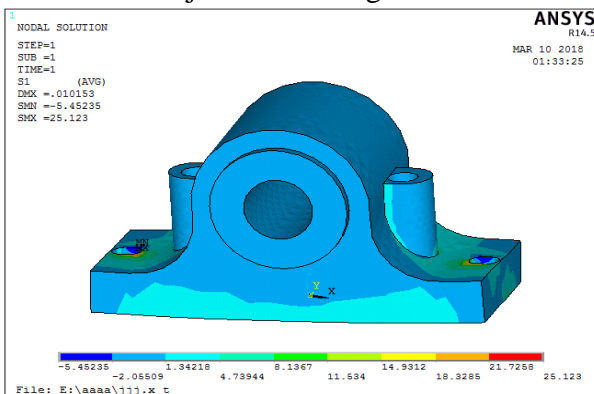


Fig 6.9: shows the vonmises stress of journal bearing

6.2 STATIC ANALYSIS OF JOURNAL BEARING USING ALUMINIUM

Material properties of Aluminium:

➤ Young modulus(E) = 71GPa

- Density (ρ) = 2650 kg/m³
- Yield strength = 180MPa
- Poisson's ratio = 0.25

Boundary conditions:

- Bottom area of journal bearing is arrested in all DOF.
- Maximum load 2000N which is observed from shaft standard values is applied at center hole of journal bearing.

Fig 6.10: shows the imported model of journal bearing

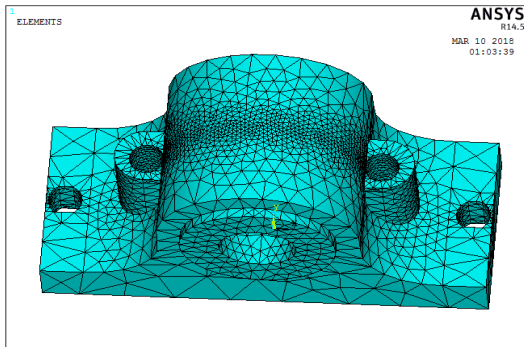


Fig 6.11: shows the imported model of journal bearing

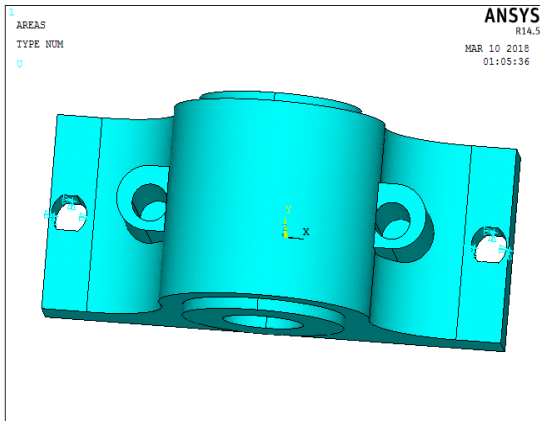


Fig 6.12: shows the fixing of journal bearing

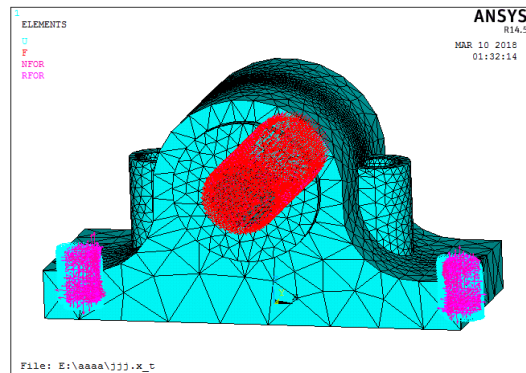
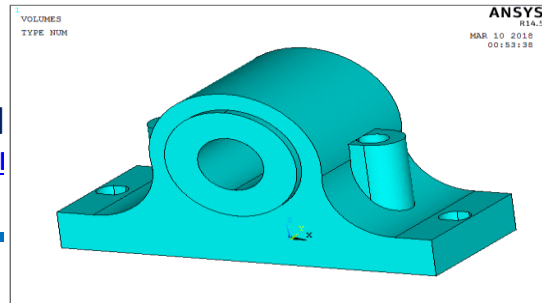


Fig 6.13: shows the loading of journal bearing

RESULTS

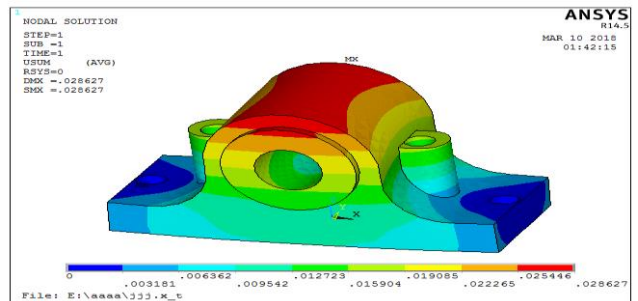


Fig 6.14: shows the displacement of journal bearing

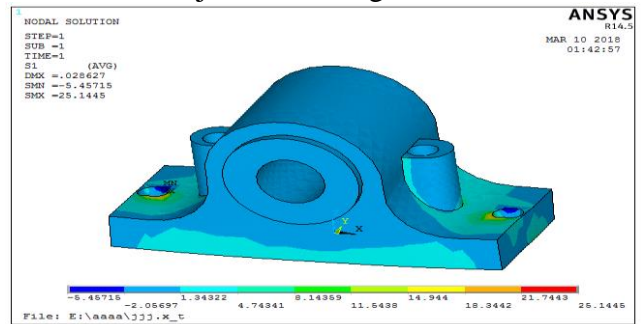


Fig 6.15: shows the 1st principal stress of journal bearing

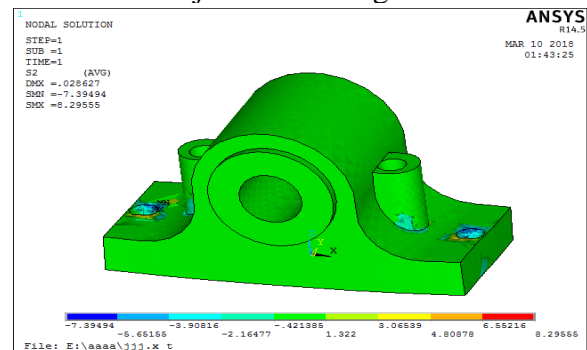


Fig 6.16: shows the 2nd principal stress of journal bearing

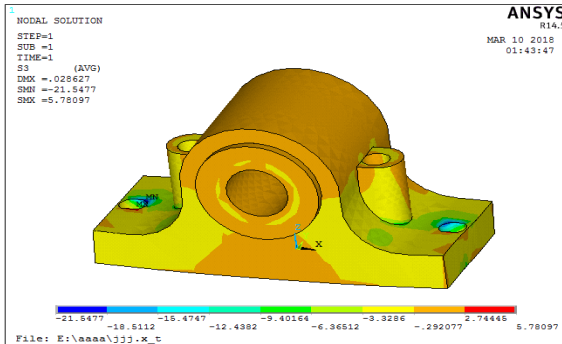


Fig 6.17: shows the 3rd principal stress of journal bearing

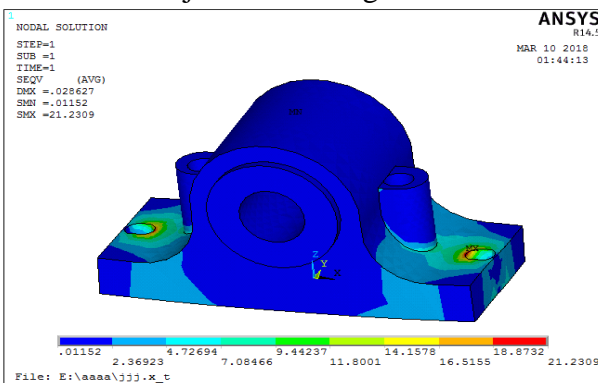


Fig 6.18: shows the vonmises stress of journal bearing

RESULTS AND CONCLUSION

Results:

In this project static analysis of journal bearing using Steel and Aluminium materials are studied. And analysis results tabulated below

RESULTS TYPE	STEE L	ALUMINIU M
Deformation(mm)	0.010	0.028
Von-mises stress(MPa)	21.21	21.23

Conclusion:

From the analysis, the minimum displacement of 0.010 mm and minimum Von Misses stress of 21.21MPa is observed on the journal bearing using steel material.

The maximum Von Misses stress of 21.21MPa is less than the yield strength of the steel material is 250 MPa. So steel journal bearing is considered as safe at loading condition.

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