A Static Structural, Modal and Thermal Analysis of a Cyclic Symmetry of a Rotor - Brake Assembly

C. NAGARAJA[1], M. GOVARDHAN GOPI[2], S. VISHNU BHARADWAJA REDDY[3], S. ZAHEER AHAMMAD[4], S. IRFAN AHAMED[5], O. SHARIEF[6], K. MADDILETI REDDY[7]

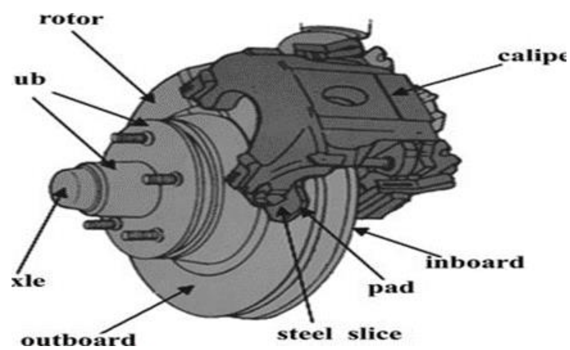
[1] C. Nagaraja, Assistant Professor, Department of ME, KSRMCE (Autonomous), Kadapa, AP, India.
[2, 3, 4, 5, 6, 7] Department of ME, KSRMCE (Autonomous), Kadapa, AP, India.

Abstract- The brake is an instrument that prevents movements. The opposite part of this device is a clutch. Many brakes utilize friction to transfer kinetic energy into heat energy. The procedure to transfer kinetic into potential energy is saved in the form of pressurized air or oil. Therefore, braking procedures convert kinetic energy into many forms. Brakes are mainly used to move axles or wheels and yet include many forms like the moving fluid surface. The vehicles employ braking mechanisms like Formula racing cars with the help of wheel brakes, flights, parachute, and helicopter and drag flaps. Automobile's friction brakes save braking heat in disc brake during braking and carry to the air slowly. The brakes prevent the slowing down of the vehicle within the smallest distance in the emergency situation. When moving down a hill, they manage the vehicle to absorb. The braking torques brakes the drums which spend large amount of heat without increase in temperature. There are three kinds of braking system. They are Mechanical Braking System, Engine Braking, and Exhaust Braking.

Indexed Terms- Analysis on Steady State Transient Thermal, Ansys-19.2 Software, A Nonlinear Static Structural and Behavior Modal Analysis of a Rotor Brake Assembly.

I. INTRODUCTION

The 1800's, the first mechanisms to slow a vehicles momentum and prevent motion were tested. Today, over 100 years later, the braking system has evolved into a complex device designed to adapt to different road conditions. From the early drum brakes to modern day discs, brake system evolution has improved safety and reduced the risk of car crashes across the globe. With so many forms of brakes that have existed over the century, it is hard to pinpoint the inventor of the original brake system; however, those who designed these systems had a common goal: to make it possible for humans to control a motor vehicle.



II. WORKING PRINCIPLE

A common misconception about brakes is that brakes squeeze against a drum or disc, and the pressure of the squeezing action slows the vehicle down. This is in fact a part of the reason for slowing down a vehicle. Actually, brakes use friction of brake shoes and drums to convert kinetic energy developed by the vehicle into heat energy. When brakes are applied, the pads or shoes that press against the brake drums or rotor convert kinetic energy into thermal energy via friction. Thus, brakes are essentially a mechanism to change energy types.

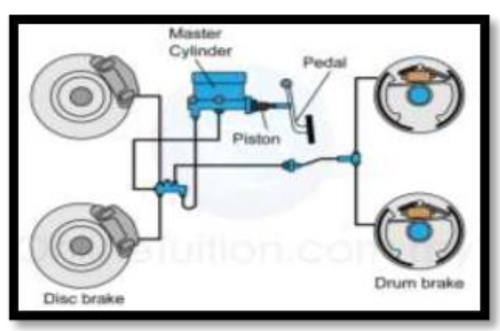


Fig: Working of braking system in automobile

III. TYPES OF BRAKE

Basically, there are three types of brakes which are used in automobile sectors.

They are as follows:

1. Mechanical brakes: These are of two types:
   a. Drum brakes
   b. Disc brakes
2. Hydraulic brakes:
   a. On the basis of frictional contact mechanism:
   b. On the basis of brake force distribution:
3. Power brakes

These are classified in four types:
   a. Air brakes
   b. Air hydraulic brakes
   c. Vacuum brakes
   d. Electric brakes
4. Emergency Brakes
5. Anti-Lock Brakes
6. Parking Brakes

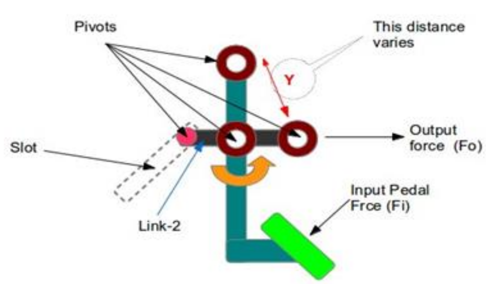
IV. COMPONENTS OF BRAKING SYSTEM

The main parts of automobile braking system include the pedal, drum and disc brakes, a brake booster and push rod, the master cylinder, valves and lines, and the emergency and anti-lock brakes.

.

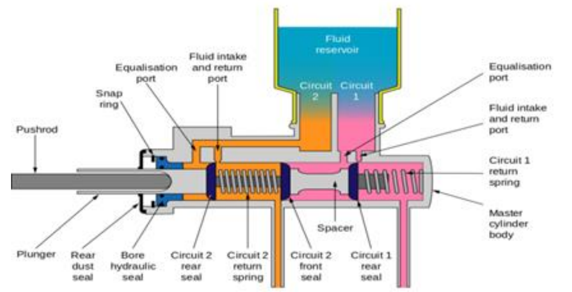
- Brake Pedal:

The pedal is what that is pushed by foot to activate the brakes. It causes brake fluid to flow through the system to put pressure on the brake pads.



- Brake Master Cylinder

The master cylinder is basically a plunger that is activated by the brake pedal. It is what holds the brake fluid and forces it through the brake lines when activated. The master cylinder is part that converts the applied force from the pedal to hydraulic pressure. The function of the part is to develop pressure, equalize the required pressure for braking, and also prevent contaminants such as water and air. Components of the master cylinder include housing, reservoir, piston, rubber cup, pressure, check valve, etc.



- Brake Lines:

Generally made of steel, brake lines are what carry the brake fluid from the master cylinder reservoir to the wheels where pressure is applied to stop the car.

- Drum Brakes

Drum brakes are located on the rear wheels. When the brakes are applied, the pressurized fluid forces its way into the wheel cylinder of the drum brakes. This pushes the brake shoes into contact with the inside of the brake drum and slows down the vehicle. A pushrod transfers motion from one shoe to the other. A drum brake is a brake that uses friction caused by a set of shoes or pads that press outward against a rotating cylinder-shaped part called a brake drum. The term drum brake usually means a brake in which shoes press on the inner surface of the drum. When shoes press on the outside of the drum, it is usually called a *clasp brake*. Where the drum is pinched between two shoes, similar to a conventional disc brake, it is sometimes called a *pinch drum brake*, though such brakes are relatively rare. A related type called a band brake uses a flexible belt or "band" wrapping around the outside of a drum.

- Disc Brakes

Most vehicles have disc brakes only on the front wheels, though newer vehicles may have disc brakes on all four wheels. With disc brakes, the fluid from the master cylinder forces into a caliper where it presses against a piston. The piston squeezes two brake pads on a disc rotor attached to the wheel. This forces the wheel to slow down and stop.

A disc brake is a type of brake that uses the calipers to squeeze pairs of pads against a disc or a "rotor"[1] to create friction.[2] This action slows the rotation of a shaft, such as a vehicle axle, either to reduce its rotational speed or to hold it stationary. The energy of motion is converted into waste heat which must be dispersed.

Hydraulically actuated disc brakes are the most commonly used form of brake for motor vehicles, but the principles of a disc brake are applicable to almost any rotating shaft. The component includes the disc, master cylinder, caliper (which contains cylinder and two brake pads) on both side of the disc.

- Wheel Cylinders:

It is very similar to a slave cylinder and functions in much the same way, internally consisting of only a simple plunger. On older vehicles, these may begin to leak and hinder the performance of the brakes. They are, however, normally inexpensive and relatively easy to replace.

The wheel cylinder consists of a cylinder that has two pistons, one on each side. Each piston has a rubber seal and a shaft that connects the piston with a brake shoe. When brake pressure is applied, the pistons are forced out pushing the shoes into contact with the drum. Some designs use two single piston wheel cylinders, one at the top of the drum and one at the bottom, each connected to one brake shoe.

Wheel cylinders must be rebuilt or replaced if they show signs of leaking. Wheel cylinders used to be made of cast iron. However, they were more prone to rusting and aluminum is now the preferred material.
It has a cylinder, two pistons, two rubber cups and a spring. The fluid presses against the pistons that move outward in the cylinder. When the pistons come closer, the liquid is forced into the master cylinder; the spring between the two pistons holds the rubber cups in positions.

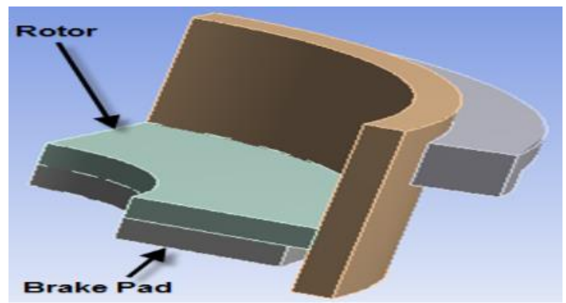
- Brake Pads:

The brake pads are what actually rub against the drums or rotors. They are made of composite materials and designed to last for many, many thousands of miles Brake pads convert the kinetic energy of the vehicle to thermal energy through friction. Two brake pads are contained in the brake with their friction surfaces facing the rotor.[1] When the brakes are hydraulically applied, the caliper clamps or squeezes the two pads together onto the spinning rotor to slow and stop the vehicle. When a brake pad heats up due to contact with the rotor, it transfers small amounts of its friction material onto the disc, leaving a dull grey coating on it. The brake pad and disc (both now having the friction material), then "stick" to each other, providing the friction that stops the vehicle.

In disc brakes, there are usually two brake pads per disc rotor. These are held in place and actuated by a caliper affixed to the wheel hub or suspension upright. Racing calipers, however, can utilize up to six pads, with varying frictional properties in a staggered pattern for optimum performance. Depending on the properties of the material, the weight of the vehicle and the speeds it is driven at, disc wear rates may vary.

- Program Description:

This tutorial demonstrates the use of cyclic symmetry analysis features in the Mechanical Application to study a sector model consisting of a rotor and brake assembly in frictional contact. With increased loading of the brake, the contact status between the pad and the rotor changes from “near”, to “sliding”, to “sticking”. Each of these contact states affects the natural frequencies and resulting mode shapes of the assembly. Three pre-stress modal analyses are used to verify this phenomenon.



Features Demonstrated

- Cyclic Regions
- Named Selections based on Criteria
- Thermal Steady-State Analysis with Cyclic Symmetry
- Static Structural Analysis with Cyclic Symmetry
- Modal Analysis with Cyclic Symmetry
- Generation of Restart Points
- Modal Analysis with Nonlinear Pressers (Linear Perturbation)

Note

The procedural steps in this tutorial assume that you are familiar with basic navigation techniques within the Mechanical application. If you are new to using the application, consider running the tutorial: “Steady-State and Transient Thermal Analysis of a Circuit Board” before attempting to run this tutorial.

Assign materials.

Accept Structural Steel (typically the default material) for the model.

a. In the Steady-State Thermal schematic, right-click the Engineering Data cell and choose Edit.... The Engineering Data tab opens and displays Structural Steel as the default material.
b. Click the Return to Project toolbar button.

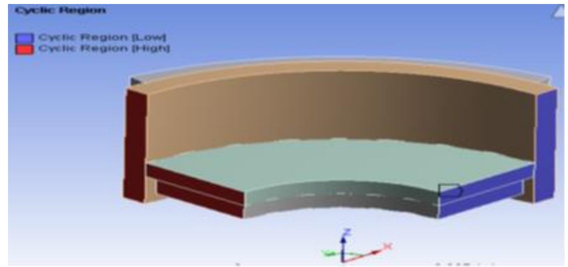
3. Attach geometry.
a. In the Steady-State Thermal schematic, right-click the Geometry cell, and then choose Import Geometry.
b. Browse to open the file “Rotor Brake agdb.” Define the Cyclic Symmetry Model

We now specify the cyclic symmetry for our quarter sector model (N = 4, 90 degrees) and prepare other general aspects of modeling in the Mechanical application. To setup a cyclic symmetry analysis, Mechanical uses a Cyclic Region object. This object requires selection of the sector boundaries, together with a cylindrical coordinate system whose Z axis is colinear with the axis of symmetry, and whose Y axis distinguishes the low and high boundaries.

1. Enter the Mechanical Application and set unit systems.
a. In the Steady-State Thermal schematic, right-click the Model cell, and then choose Edit.... The Mechanical Application opens and displays the model.
b. From the Menu bar, choose Units> Metric (mm, kg, N, s, mV, mA).

2. Define the Coordinate System to specify the axis of symmetry.
a. Right-click Coordinate Systems in the tree and choose Insert> Coordinate System.
b. In the Details view of the newly-created Coordinate System, set Type to Cylindrical and Define by to Global Coordinates.

3. Define the Cyclic Region object.
a. Right-click Model in the tree and choose Insert> Symmetry.
b. Right-click Symmetry and choose Insert> Cyclic Region. The direction of the Y-axis should be compatible with the selection of low and high boundaries. The low boundary is designated as the one with a lower value of Y or azimuth.
c. Select the three faces that have lower azimuth for the low boundary. These faces are highlighted in blue in the figure below



d. Select the three matching faces on the opposite end of the sector for the high boundary. These faces are highlighted in red in the figure below

4. Define Connections. Frictional contact exists between the rotor and brake pad, whereas bonded contact exists between the wall and the rotor.
a. Expand the Connections folder in the tree, and then expand the Contacts folder. Within the Contacts
b. folder, two contact regions were detected automatically and displayed as Contact Region and Contact Region 2.
c. Right-click the Contacts folder and choose Renamed Based on Definition. The contact region names automatically change to Bonded - Pad to Rotor and Bonded - Blade to Wall respectively.
d. Highlight Bonded - Pad to Rotor and in the Details view, set Type to Frictional. Note that the name of the object changes accordingly.
e. In the Friction Coefficient field, type 0.2 and press Enter.

Note
For higher values of contact friction coefficient, a damped modal analysis would be needed. At a level of 0.2 damping effects are being neglected.

- Generate the Mesh

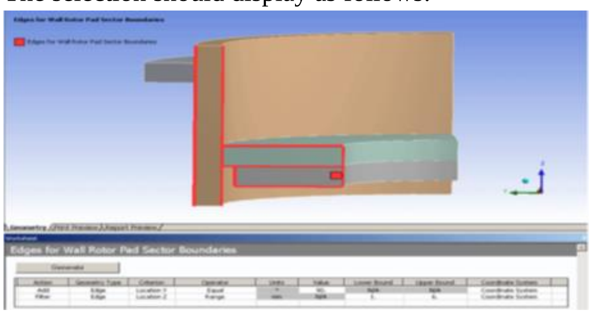
In the following section we’ll use mesh controls to obtain a mesh of regular hexahedral elements. The Cyclic Region object will guarantee that matching meshes are generated on the low and high boundaries of the cyclic sector. Taking advantage of the shape and dimensions of the model, Named Selections will be used to choose the edge selections for each mesh control.

- Mesh control: Element Size on Pad-Wall-Rotor:

1. Create a Named Selection for this Mesh Control.
a. Right-click on Model and choose Insert> Named Selection.

b. Highlight the Selection object, and set Scoping Method to Worksheet.
c. Program the Worksheet, as shown below, to select the edges at 90 degrees of azimuth in the cylindrical coordinate system, keeping those in the z-axis range [1mm, 6 mm] (to remove the thickness of the wall). To add rows to the Worksheet, right-click in the table and select the option from the flyout menus.
d. Click the Generate button. You should see 11 edges.
e. Rename the object to Edges for Wall Rotor Pad Sector Boundary.

The selection should display as follows:

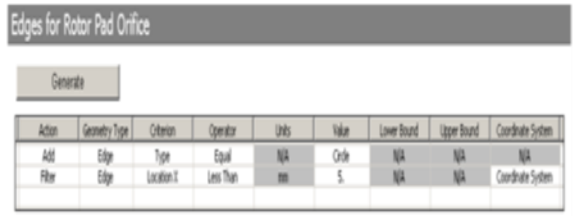


Note
It may be useful to undock the Worksheet window and tile it with the Geometry view as shown.

2. Insert a Mesh Sizing control.
a. Right-click on Mesh and choose Insert> Sizing.
b. Set Scoping Method to Named Selection.
c. Choose the named selection defined in the previous step.
d. Set its Element Size to 0.5 mm.
e. Set Behavior to Soft.

- Mesh control: Number of Divisions on Pad-Rotor:

1. Create a Named Selection to pick the circular edges in the orifice of the pad and rotor. This Named Selection will pick the circular edges in the orifice of the pad and rotor, which is within a radius of 5 mm.
a. Right-click on Model and choose Insert> Named Selection.
b. Highlight the Selection object, and set Scoping Method to Worksheet.
c. Rename the object to Edges for Rotor Pad Orifice.
d. Program the Worksheet, as shown below.



Edges for Rotor Pad Orifice

Generate

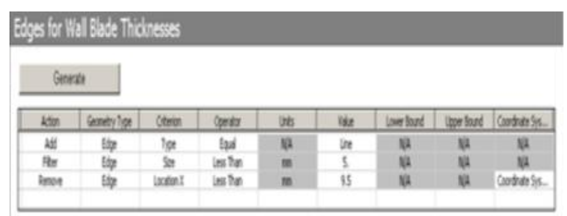
Action	Geometry Type	Criterion	Operator	Units	Value	Lower Bound	Upper Bound	Coordinate System
Add	Edge	Type	Equal	N/A	Circle	N/A	N/A	N/A
Filter	Edge	Location X	Less Than	mm	5.	N/A	N/A	Coordinate System

e. Click the Generate button. You should see 4 edges.

2. Insert a Mesh Sizing Control as before to select this Named Selection.
a. Right-click on Mesh and choose Insert> Sizing.
b. Set Scoping Method to Named Selection.
c. Choose the named selection defined in the previous step.
d. Set its Type to Number of Divisions and specify 9.
e. Set Behavior to Hard.

- Mesh control: Element Size on Wall-Blade

1. Create a Named Selection object to pick the thicknesses of the Wall and Blade.
a. Right-click on Model and choose Insert> Named Selection.
b. Highlight the Selection object, and set Scoping Method to Worksheet.
c. Rename the object to Edges for Wall Blade Thicknesses.
d. Program the Worksheet as shown below.



Edges for Wall Blade Thicknesses

Generate

Action	Geometry Type	Criterion	Operator	Units	Value	Lower Bound	Upper Bound	Coordinate Sys...
Add	Edge	Type	Equal	N/A	Line	N/A	N/A	N/A
Filter	Edge	Size	Less Than	mm	5.	N/A	N/A	N/A
Remove	Edge	Location X	Less Than	mm	9.5	N/A	N/A	Coordinate Sys...

e. Click the Generate button. You should see 16 edges.

2. Insert a Mesh Sizing Control as before to select this Named Selection.
a. Right-click on Mesh and choose Insert> Sizing.
b. Set Scoping Method to Named Selection.
c. Choose the named selection defined in the previous step.
d. Set its Element Size to 1 mm.
e. Set Behavior to Hard.

- Mesh Control: Number of Divisions on Blade - Longer Edges

1. Create a Named Selection object to pick the longer edges of the Blade.
a. Right-click on Model and choose Insert> Named Selection.

b. Highlight the Selection object, and set Scoping Method to Worksheet.
c. Rename the object to Edges for Blade.
d. Program the Worksheet as shown below.

Edges for Blade

Generate

Action	Geometry Type	Criterion	Operator	Units	Value	Lower Bound	Upper Bound	Coordinate Syst...
Add	Edge	Type	Equal	N/A	Circle	N/A	N/A	N/A
Filter	Edge	Location Z	Range	mm	N/A	6.5	7.5	Coordinate Syst...

e. Click the Generate button. You should see 2 edges.
2. Insert a Mesh Sizing Control as before to select this Named Selection.
a. Right-click on Mesh and choose Insert> Sizing.
b. Set Scoping Method to Named Selection.
c. Choose the named selection defined in the previous step.
d. Set its Type to Number of Divisions and specify 14.
e. Set Behavior to Hard.

Mesh Control: Number of Divisions on Blade - Shorter Edges
1. Create a Named Selection object to pick the shorter edges of the Blade.
a. Right-click on Model and choose Insert> Named Selection.
b. Highlight the Selection object, and set Scoping Method to Worksheet.
c. Rename the object to Edges for Blade 2.
d. Program the Worksheet as shown below.

Edges for Blade 2

Generate

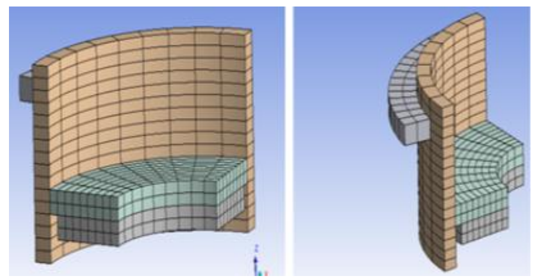
Action	Geometry Type	Criterion	Operator	Units	Value	Lower Bound	Upper Bound	Coordinate Syst...
Add	Edge	Type	Equal	N/A	Circle	N/A	N/A	N/A
Filter	Edge	Size	Less Than	mm	2.	N/A	N/A	N/A

e. Click the Generate button. You should see 2 edges.
2. Insert a Mesh Sizing Control as before to select this Named Selection.
a. Right-click on Mesh and choose Insert> Sizing.
b. Set Scoping Method to Named Selection.
c. Choose the named selection defined in the previous step.
d. Set its Type to Number of Divisions and specify 1.
e. Set Behavior to Hard.

Mesh Control: Method on Pad-Rotor-Wall-Blade
1. Insert a Sweep Method control.
a. Right-click Mesh in the tree and choose Insert> Method.
b. Select all the bodies by choosing Edit> Select All from the toolbar, then click the Apply button.
c. In the Details view, set Method to Sweep.
d. Set Free Face Mesh Type to All Quad.

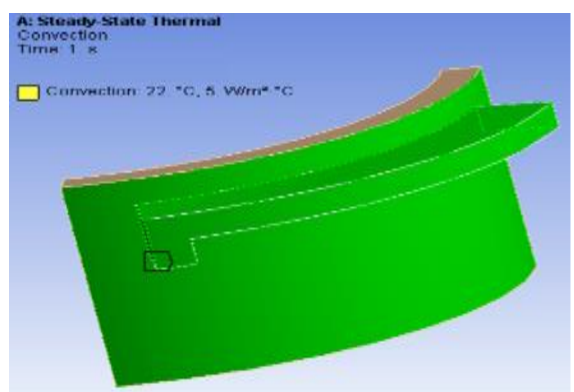
Generate the Mesh

• For convenience, select all 6 mesh controls defined, right-click and choose Rename Based on Definition.
• Right-click Mesh in the tree and choose Generate Mesh. The mesh should appear as shown below:



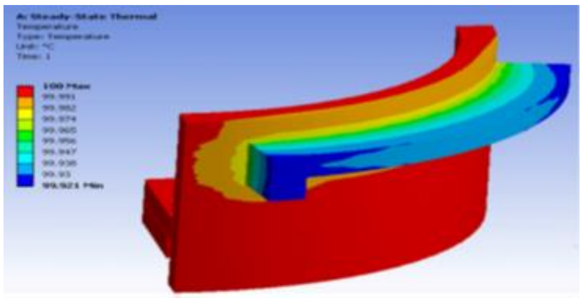
Steady-State Thermal Analysis
We now proceed to define the boundary conditions for a thermal analysis featuring cyclic symmetry.

Thermal boundary conditions are prescribed throughout the model while steering clear of the faces comprising the sector boundaries since temperature constraints are already implied there.
1. Define a convection interface.
a. Right-click Steady-State Thermal in the tree and choose Insert> Convection.
b. Select the outer faces of the Wall and the Blade as shown in the figure (8 faces).



c. Specify a Film Coefficient of air by right-clicking on the property and choosing Import Temperature Dependent upon which you choose Stagnant Air - Simplified Case.
2. Insulate the upper and lower faces of the Wall.
• Select the upper and lower faces of the Wall, then right-click and choose Insert> Perfectly Insulated.

3. Apply a temperature load to the Pad and Rotor.
a. Select the remaining faces on the assembly on the Pad and the Rotor, then right-click and choose Insert>Temperature. Exclude any faces on the sector boundaries or in the frictional contact.
b. Type 100°C as the Magnitude and press Enter.
4. Solve and review the temperature distribution.
a. Right-click Solution under Steady-State Thermal and choose Insert> Thermal> Temperature.
b. Solve the steady-state thermal analysis.
c. Review the temperature result by highlighting the Temperature result object.

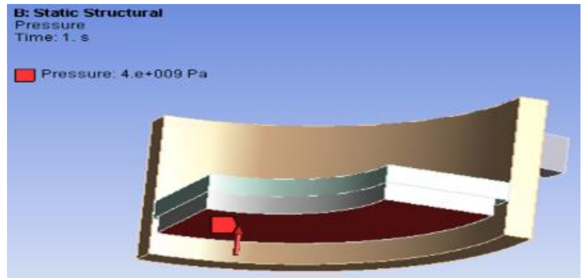


Note
Although insignificant in this model, temperature variations and their effect on the structural material properties are generally important to the formulation of physically accurate models.

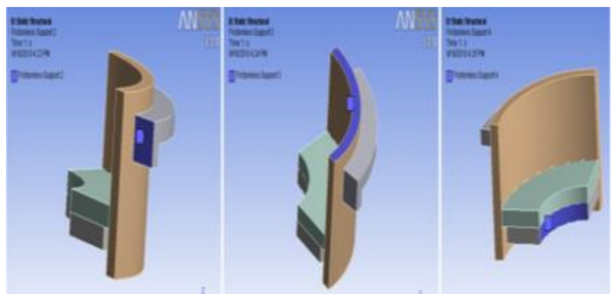
V. STATIC STRUCTURAL ANALYSIS

In this analysis, the brake is loaded onto the rotor in a single load step. The contact status is monitored at various stages of loading and three points are selected as pre-stress conditions for subsequent modal analyses. Because both contact and geometric nonlinearities are present, each pre-stress condition will present a different effective stiffness matrix to its corresponding modal analysis. The solver uses restart points, generated in the static analysis, to record the snapshot of the nonlinear tangent stiffness matrices and transfers them into the subsequent linear systems. This technique is referred to as Linear Perturbation.

1. Apply the pressure and boundary conditions to engage the brake pad into the rotor.
a. Select the bottom face of the Pad as shown below. Right-click the Static Structural object in the tree and choose Insert> Pressure.
b. In the Details view, click the Magnitude flyout menu, choose Function, and specify: =time*time*4000, then press Enter.
This represents a quadratic function reaching 4000 MPa by the end of the load step.



C. Set up the frictionless supports on the faces of Blade, Wall and Pad as shown below.

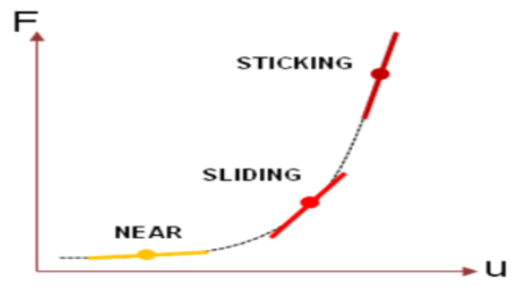


2. Configure the Analysis Settings.
a. Set Auto Time Stepping to On.
b. Set Define By to Sub steps.
c. Set Initial Sub steps to 30.
d. Set Minimum Sub steps to 10.
e. Set Maximum Sub steps to 30.
f. Set Large Deflection to On to activate geometric nonlinearities.
g. To ensure that Restart Points are generated, under Restart Controls, set Generate Restart Points to manual and request to retain all files for load steps and sub steps. Maximum points to save should also be set to all.
3. Proceed to solve the model using the standard procedure.

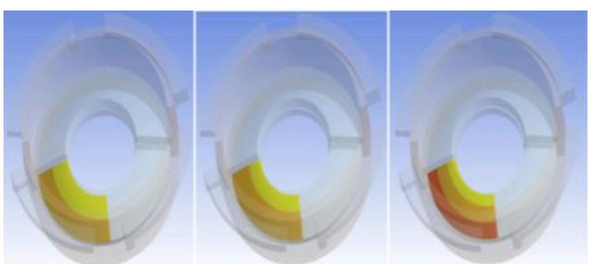
Reviewing the contact status changes during the course of the load application

The contact status will change with increasing loads from Near, to Sliding, to Sticking. A status change from Near to Sliding reflects the engagement of contact impenetrability conditions (normal direction).

A change from Sliding to Sticking, reflects additional engagement of contact friction conditions(tangential direction). This progression will generally reflect an increased effective stiffness in the tangent stiffness matrix, which can be illustrated by a Force-deflection curve:



To review the contact status, insert a Contact Tool in the Solution folder. To display only the contact results at the frictional contact, unselect Bonded - Wall to Blade in the Contact Tool Worksheet. Insert three different Contact Status results with display times at 0.03, 0.5 and 0.8 seconds, which should reveal the progression in contact status as shown below (from left to right):

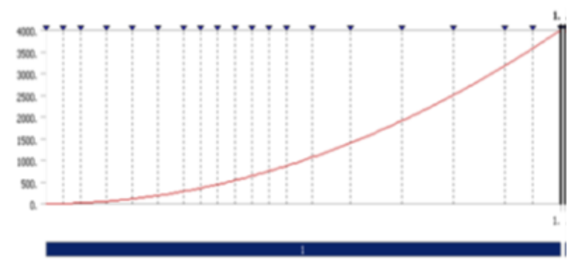


The legend for these contact status plots is as follows:
• Yellow - Near
• Light Orange - Sliding
• Dark Orange – Sticking

VI. MODAL ANALYSIS

There are three modal analyses to study the effect of contact status and stress stiffening on the free vibration response of the structure. Each of these will be based on a different restart point in the static structural analysis.

To see all available restart points, you can inspect the timeline graph displayed when the Analysis Settings object of the Static Structural analysis is selected after solving. Restart points are denoted as blue triangle marks atop the graph:



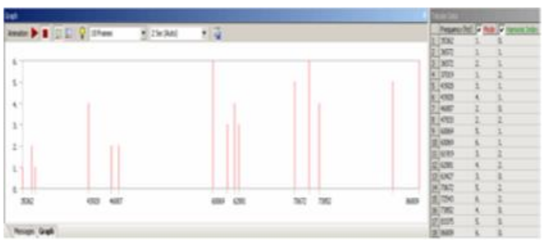
To select the restart point of interest, go to the Pre-Stress (Static Structural) object under each Modal Analysis. Make sure Pre-Stress Define by is set to Time and specify the time. The object will acknowledge the restart point in the Reported Load step, Reported Sub step and Reported Time fields.

Configure the Modal analyses as follows:
• In Modal 1 set Pre-Stress Time to 0.033 seconds.
• In Modal 2 set Pre-Stress Time to 0.5 seconds.
• In Modal 3 set Pre-Stress Time to 0.8 seconds.
Because the boundary conditions (that is, the frictionless supports) are automatically imported from the static analysis, we can proceed directly to solve. Solving and Reviewing Modal Results

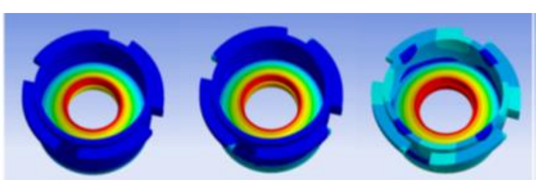
We'll monitor the lowest frequencies of vibration which belong to Harmonic Indices 0 (symmetric) and2 (anti-symmetric).
1. Right-click on the Solution folder of each Modal analysis and choose Solve.
2. When the solutions complete, go to the Tabular Data window of each modal analysis. You can inspect the listing of modes and their frequencies. Because our structure has a symmetry of N=4, there will be three solutions, namely for Harmonic Indices 0, 1 and 2.
3. In the Tabular Data window of each modal analysis, select the two rows for Harmonic Index 0 Mode 1and Harmonic Index 2 - Mode 1. Right-click and choose Create Mode Shape Results.

The image below shows this view for the first Modal analysis:

An interesting alternative to this view is to see the sorted frequency spectrum. You may review this by setting the X-Axis to Frequency on any of the Total Deformation results in each modal analysis:



At this point, each modal analysis should have two results for Total Deformation to inspect the first mode of Harmonic Indices 0 and 2. Recall the meaning of Harmonic Index solutions and how they apply to the model. Harmonic Index0 represents the constant offset in the discrete Fourier Series representation of the model and corresponds to equal values of every transformed quantity, for example, displacements in X, Y and Z directions, in consecutive sectors. Thus, deformations that are axially positive in one sector will have the same axially positive value in the next. The following picture compiles, from left to right, the mode shapes for the Near, Sliding and Sticking status at Harmonic Index 0:



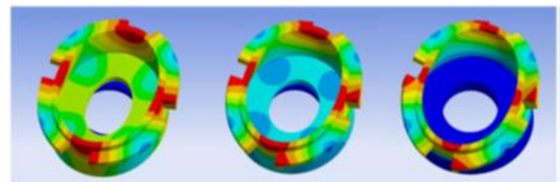
Near	Sliding	Sticking
35362 Hz	41476 Hz	49208 Hz

Notice how increased engagement of the frictional contact in the assembly has the effect of producing higher frequency vibrations. Also, the mode of vibration goes from being localized at the contact interface when the contact is Near, but is forced to distribute throughout the wall of the rotor as the contact sticks.

Note

You may need to specify Auto Scale on the Results toolbar so the mode shapes are plotted as shown.

Harmonic Index 2 solutions correspond to N/2 for our sector (90 degrees or N = 4). This Harmonic Index, sometimes called the asymmetric term in the Fourier series, represents alternation of quantities.



Near	Sliding	Sticking
37019 Hz	46014 Hz	46650 Hz

The lowest mode shows nearly independent vibration of the rotor relative to the blade. On the highest mode, sticking reduces this relative movement. For a continued discussion on post-processing for Cyclic Symmetry and especially on features for post processing degenerate Harmonic Indices (those between 0 and N/2), please see reviewing results for Cyclic Symmetry in a Modal Analysis in the Mechanical help.

REFERENCES

[1] C.H. Gao, J.M. Huang, X.Z. Lin, X.S. Tang, “stress analysis of thermal fatigue fracture of brake disks based on thermo mechanical coupling”, ASME Journal of Tribology, 129 (July 2007): 536-544.

[2] H.B. Yan, Q.C. Zhang, T.J. Lu, “Heat transfer enhancement by X-type lattice in ventilated brake disc”, International Journal of Thermal Sciences, 107 (April 2016): 39-55.

[3] Interface Dealing in Engine Block Assembly Modal Analysis by FEM by Shu Ge-Qun Yang Zhi-Yi.

[4] F.E. Analysis of Residual Stresses in an Engine Cylinder Head by M.Durando, R. Bonavolonta, D. Petronio, M. Leghissa.

[5] Vibrational Analysis of a VCR Diesel Engine by Sachinpente, Nitin Wankhade, Pankaj Bhoyar.

[6] To Study the Dynamic Behavior of Engine Assembly by Using Altair OptiStruct by Amit Joshi, Devashish Sarkar.