TATA TECHNOLOGIES

Static Implicit Stress Simulation of Ball Bearings

Ankur Kumar, TATA Technologies Inc

/// Engineering a better world

Static Implicit Stress Simulation of Ball Bearings

Abstract

Ball Bearings are used in multiple industry verticals as a connection to transfer loads and motion from one part of a system to the other with minimal friction resistance. In this study we demonstrate the process, the challenges, and the advantages of simulating a simplified version of a shaft bearing model in Abaqus. The static implicit technique to perform ball bearing stress analysis is unique where Abaqus can outshine its competitors if the model is set up correctly.

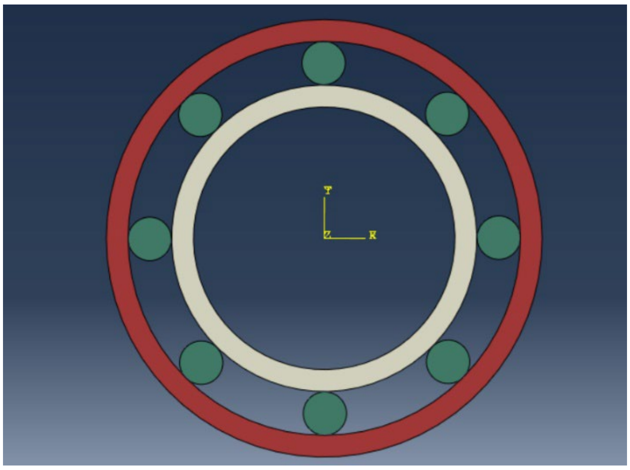
Introduction

There are primarily four types of ball bearings: Rolling Bearing, Plain Bearing, Thrust Bearing and Sleeve Bearing. They are differentiated based on their loads carrying capacity in axial and radial directions. While some bearings can take only radial loads, others can take both axial and radial loads at the same time. The loads include both forces as well as torques.

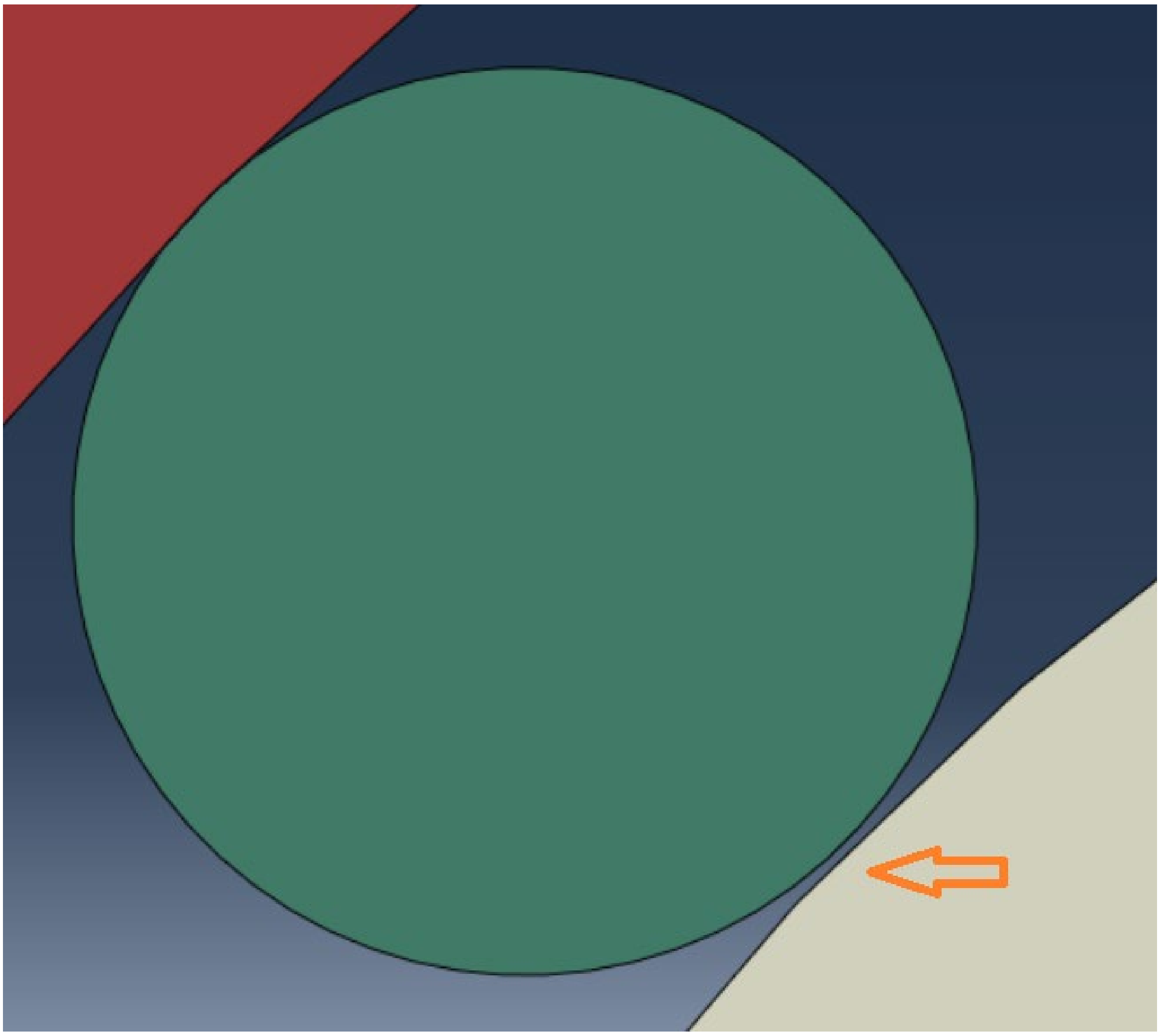
Irrespective of the type of bearings, they have some common features. First, they have similar geometry which is of cyclic symmetry in nature. The rollers may be cylindrical or barrel in shape. Second, they are made of only metals and alloys which makes it easier to define a material model for simulation. Third, they have high stiffness so they can withstand very large loads without permanent deformation.

In this study, we are modeling a rolling bearing with cylindrical rollers. This allows us to use a 2D plane stress approach instead of a full 3D model. The scope of study includes the methods used to fix convergence as well as effects of roller stiffness on the overall stress response of the bearings.

Problem Description



The geometry is simple. The bearing has an inner ring diameter of 30 mm, the outer ring diameter of 45 mm. There are eight cylindrical rollers with a diameter of 5mm. No friction is assumed in the model.
An interesting part of the model is a clearance of 0.04 mm between the roller and the inner ring. The inner ring is fixed at inner edge, and the outer ring is subjected to a vertical downward distributed load of 20kN.



The model can be assumed as plane stress with out-of-plane thickness of 5mm. All parts are made of steel with E value of 2e5 MPa, poisson ratio of 0.3 and Yield stress of 1900 MPa.

The model should not be subjected to any initialization which means the clearance between the rollers and the inner ring should exist at the beginning of simulation. Any efforts to remove the clearance may result in rollers touching the ring at a different location than expected. The precise estimation of contact points is one of the objectives of this work.
Further, the model should be solved using static implicit scheme. This model is not subject to any impact or collision, so the dynamic effects are not the focus. The stress response is required once the structure fully stabilizes in response to the load and the contacts are well established. The explicit scheme is the last resort if the static implicit fails to solve the problem. In-spite of simple geometry and loading scenario, this problem is non-trivial because of potential numerical issues the implicit solver may experience due to clearance.

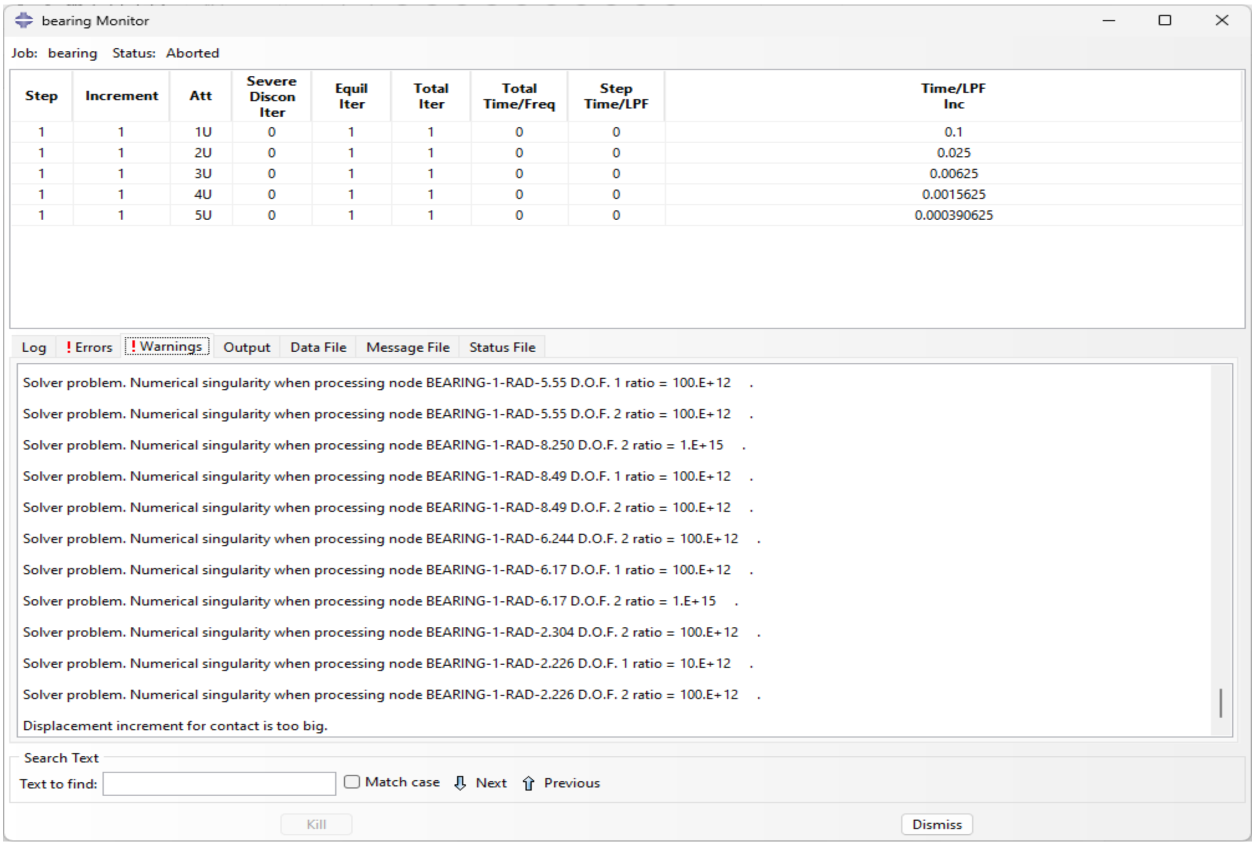
The Abaqus Implicit Scheme:
The implicit numerical scheme is unconditionally stable in nature. Irrespective of the load increment size, it converges to a unique solution if modeled correctly. An initial large increment size may result in cutbacks, but the solver eventually stabilizes and marches forward incrementally until a solution is achieved.
Abaqus uses the Newton Raphson implicit numerical scheme for static linear and Non-Linear class of problems. The scheme works with efficiency and accuracy in situations of L0 and L1 continuity i.e., the first derivative of the function exists. A function f(x)=0 can be discretized using a Newton Raphson method as follows:

$$x_{n+1} = x_n - \frac{f(x_n)}{f'(x_n)}$$

Where $f'(x_n) = [f(x_n) - f(x_{n-1})]/(x_n - x_{n-1})$

The equation above constitutes a system of equation with n as a counter from zero at initial state to N at final state. The solution of this system of equations would require a matrix inversion which may exhibit singularity if the model is under constrained or zero pivots if model is over constrained.
The given model is under constrained because of the clearance between rollers and inner ring. The load transmitted to rollers is not counter balanced due to initial clearance. As a result, the model struggles with equilibrium as soon as the first load increment is applied. In numerical terms, the stiffness matrix becomes singular, the first derivative in Newton Raphson method approaches zero and the iterative scheme fails to converge.

The Modeling Approach:


bearing Monitor

Job: bearing Status: Aborted

Step	Increment	Att	Severe Discon Iter	Equil Iter	Total Iter	Total Time/Freq	Step Time/LPF	Time/LPF Inc
1	1	1U	0	1	1	0	0	0.1
1	1	2U	0	1	1	0	0	0.025
1	1	3U	0	1	1	0	0	0.00625
1	1	4U	0	1	1	0	0	0.0015625
1	1	5U	0	1	1	0	0	0.000390625

Log | ! Errors | ! Warnings | Output | Data File | Message File | Status File

Solver problem. Numerical singularity when processing node BEARING-1-RAD-5.55 D.O.F. 1 ratio = 100.E+12 .
Solver problem. Numerical singularity when processing node BEARING-1-RAD-5.55 D.O.F. 2 ratio = 100.E+12 .
Solver problem. Numerical singularity when processing node BEARING-1-RAD-8.250 D.O.F. 2 ratio = 1.E+15 .
Solver problem. Numerical singularity when processing node BEARING-1-RAD-8.49 D.O.F. 1 ratio = 100.E+12 .
Solver problem. Numerical singularity when processing node BEARING-1-RAD-8.49 D.O.F. 2 ratio = 100.E+12 .
Solver problem. Numerical singularity when processing node BEARING-1-RAD-6.244 D.O.F. 2 ratio = 100.E+12 .
Solver problem. Numerical singularity when processing node BEARING-1-RAD-6.17 D.O.F. 1 ratio = 100.E+12 .
Solver problem. Numerical singularity when processing node BEARING-1-RAD-6.17 D.O.F. 2 ratio = 1.E+15 .
Solver problem. Numerical singularity when processing node BEARING-1-RAD-2.304 D.O.F. 2 ratio = 100.E+12 .
Solver problem. Numerical singularity when processing node BEARING-1-RAD-2.226 D.O.F. 1 ratio = 10.E+12 .
Solver problem. Numerical singularity when processing node BEARING-1-RAD-2.226 D.O.F. 2 ratio = 100.E+12 .
Displacement increment for contact is too big.

Search Text

Text to find: Match case Next Previous

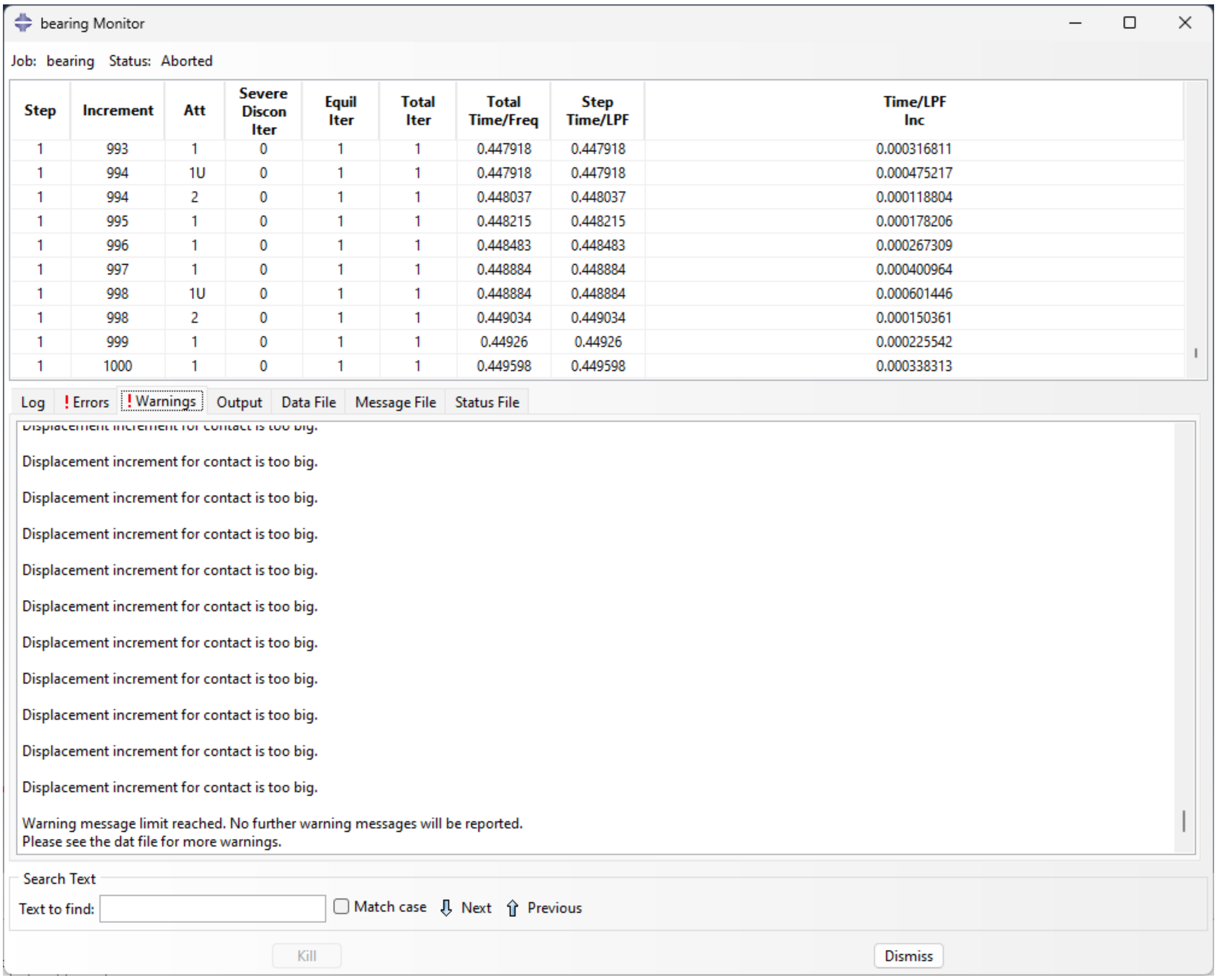
Kill Dismiss

The model is simple enough to build from scratch in any pre-processor. It was built in Abaqus CAE. The meshing, section properties definition and assignments as well as loads and boundary conditions are applied as usual. General contact without friction is used to define interactions. Default penalty stiffness is used for contacts in the normal direction. A coupling is defined for the outer ring and its 1 and 6 degrees of freedom are fixed using boundary conditions so the outer ring can only move in the downward Y direction.

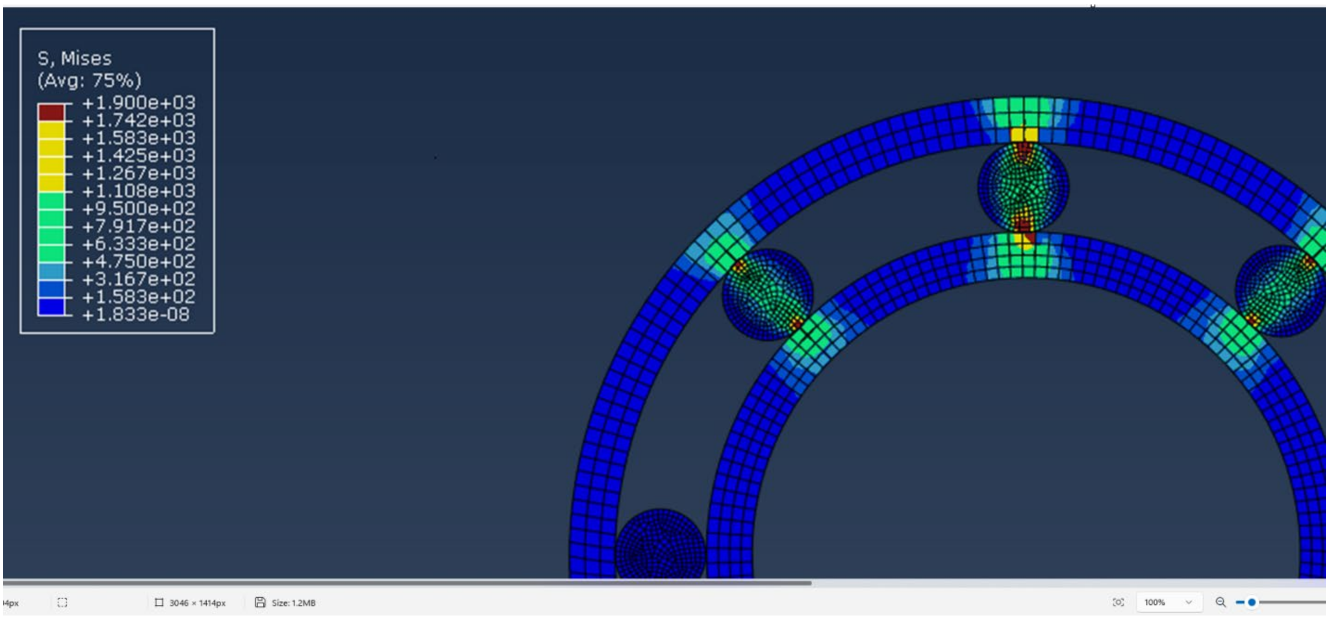
The acceptable solution is achieved in couple of stages to overcome singularity issues as we know that any change in position of parts either in CAD or through initialization is not permitted.

In stage 1 the job is executed as such without any special consideration for numerical singularities. The job went through multiple cutbacks in the increment size and excited with errors. Multiple singularity warning messages are printed in diagnostics as seen below. These warnings, when studied, refer to the numerical singularities of 1,2 and 6 degrees of freedoms at most of the roller nodes. The BEARING-1-RAD-xx refers to the roller instances with xx ranging from 1 to 8. DOF 1, DOF 2 and DOF6 refer to the global Tx, Ty, and Rz directions of the model. The model cannot be constrained further as per rules to fix these singularities.

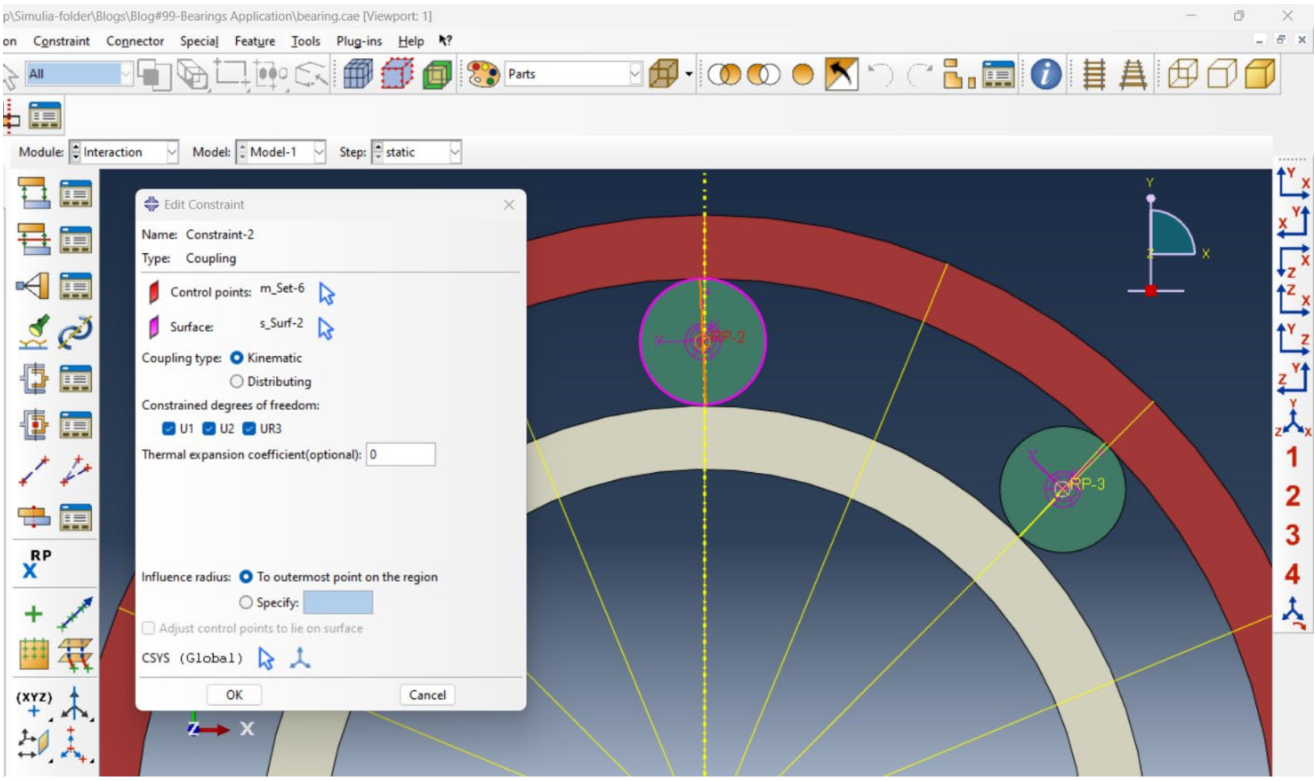
In stage 2, a damping factor of 0.004 is applied throughout the model. The damping eliminates the singularity, and simulation eventually kicks off in steps and gradually converges. The adaptive time increment ranges between 1e-3 and 6e-3. As time increment exceeds 6e-3, Abaqus encounters a cut-back. This increment size is good enough for moderate size jobs executed on 8 to 16-cores desktops. The job diagnostic and stress responses are below. The stress hot spots are in the rollers instead of the rings, which is an interesting observation. The rollers reach the Yield point stress at the point of contact and accordingly stresses are 1900 MPa.



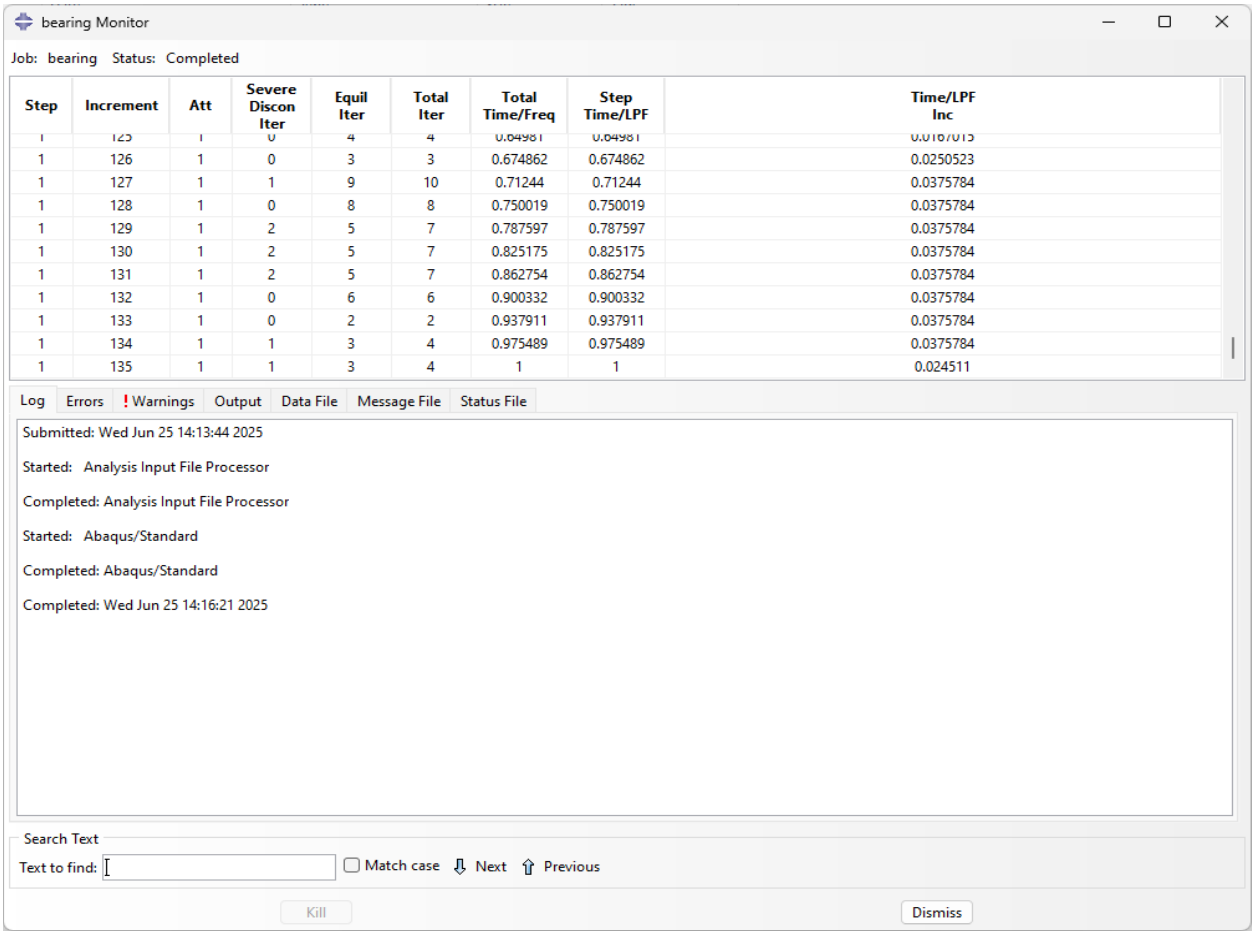
Step	Increment	Att	Severe Discon Iter	Equil Iter	Total Iter	Total Time/Freq	Step Time/LPF	Time/LPF Inc
1	993	1	0	1	1	0.447918	0.447918	0.000316811
1	994	1U	0	1	1	0.447918	0.447918	0.000475217
1	994	2	0	1	1	0.448037	0.448037	0.000118804
1	995	1	0	1	1	0.448215	0.448215	0.000178206
1	996	1	0	1	1	0.448483	0.448483	0.000267309
1	997	1	0	1	1	0.448884	0.448884	0.000400964
1	998	1U	0	1	1	0.448884	0.448884	0.000601446
1	998	2	0	1	1	0.449034	0.449034	0.000150361
1	999	1	0	1	1	0.44926	0.44926	0.000225542
1	1000	1	0	1	1	0.449598	0.449598	0.000338313



If the rollers are deformable, then stage 2 is the optimum point to conclude the simulation. However, in many commercial applications, rollers are rigid and stress on rings and shafts are required. In such a case, there is room for stage 3. A few couplings are defined for the surface nodes of each roller with reference node at the geometric center of the roller. All DOF's are coupled as shown. Next, grounded springs with a small stiffness of 0.1 N/mm are defined in each DOF's at each reference node of the roller couplings using cylindrical coordinate system.

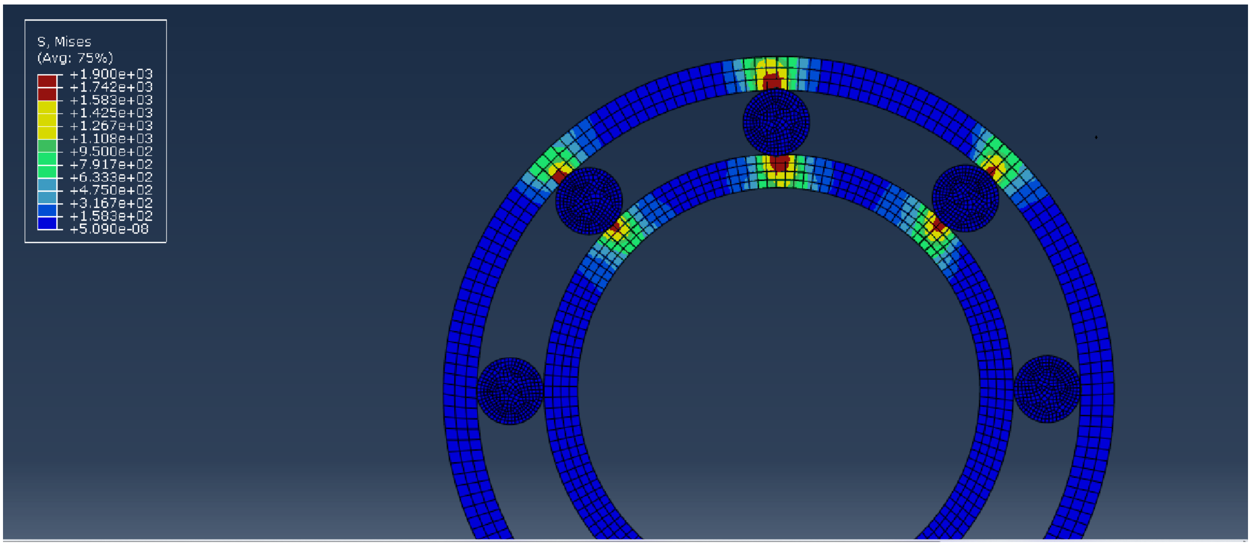


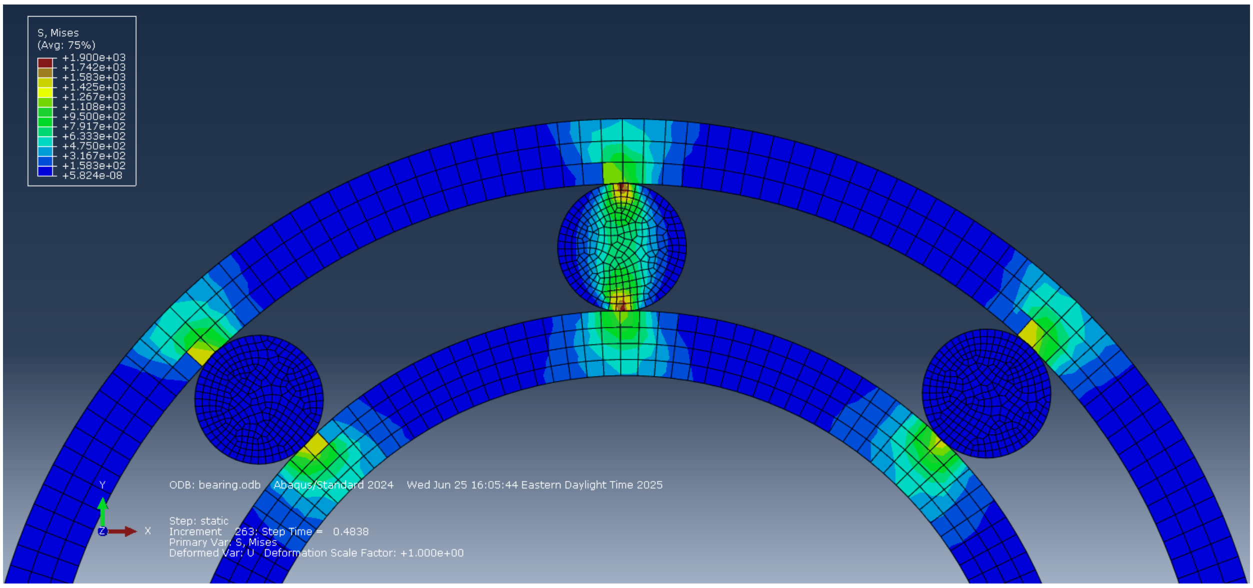
The couplings turn the rollers into rigid objects as all DOF's are coupled. The convergence behavior of this model is much better compared to the previous one. The time increment is 0.0375 and the job completes much faster.



Step	Increment	Att	Severe Discon Iter	Equil Iter	Total Iter	Total Time/Freq	Step Time/LPF	Time/LPF Inc
1	125	1	0	4	4	0.64981	0.64981	0.0167015
1	126	1	0	3	3	0.674862	0.674862	0.0250523
1	127	1	1	9	10	0.71244	0.71244	0.0375784
1	128	1	0	8	8	0.750019	0.750019	0.0375784
1	129	1	2	5	7	0.787597	0.787597	0.0375784
1	130	1	2	5	7	0.825175	0.825175	0.0375784
1	131	1	2	5	7	0.862754	0.862754	0.0375784
1	132	1	0	6	6	0.900332	0.900332	0.0375784
1	133	1	0	2	2	0.937911	0.937911	0.0375784
1	134	1	1	3	4	0.975489	0.975489	0.0375784
1	135	1	1	3	4	1	1	0.024511

The stress response shows the hot spots at outer ring instead with same Max value of 1900 MPa. The rollers behave as rigid with no stress shown. The stress response comparison between stage 2 and stage 3 results show the movement of hot spots from rollers to ring with no change in peak stress values.





Results and Conclusions

This white paper is based on a real-life commercial case. The model has been changed but the problem has been accurately replicated.

Abaqus is a proven commercial finite element code that outshines its competitors while dealing with tough to solve boundary and material non linearities. Abaqus shows robust performance in implicit problems with singularity. Many other commercial FEA solutions may not converge in such a situation even with springs. Eventually the user either changes the CAD to remove clearances or migrates to an explicit scheme with no real benefits. In this model, Abaqus solved the problem with accuracy and speed without CAD modification or explicit scheme.

References

1. A surface wear functionality using Archard wear model has been introduced in Abaqus 2025 release. It may be of interest in ball bearing design.
2. Please stay tuned for more information in our blog posts. PLM Tech Talk by TATA Technologies

TATA TECHNOLOGIES

TATA

For more information about Tata Technologies and what we can do to help create better products for your customers, visit: www.tatatechnologies.com

/TataTechnologies
@tatatechnologies
/TataTech_News
/TataTechnologies
/TataTechnologies

/// Engineering a better world