

A Discussion of Complex Eigenvalue Analytical Methods as They Relate to the Prediction of Brake Noise

Robert S. Ballinger
BWI Group North America

ABSTRACT

The complex eigenvalue analysis has been used by the brake research community to study friction-induced squeal in automotive disk brake assemblies. The analysis process uses a nonlinear static pre-stressed normal modes analysis simulation sequence followed by a complex eigenvalue extraction algorithm to determine the dynamic instabilities. When brake hardware exists, good correlation between analysis results and experimental data can be obtained. Consequently, complex eigenvalue analysis can be a valuable method in an effort to understand brake components that might have a propensity to influence the noise behavior of a brake system. However, when hardware does not exist and the complex eigenvalue method is asked to be predictive, it becomes a difficult, if not impossible task. This paper will focus on some of the reasons the complex eigenvalue analysis method is not a reliable predictor of friction-induced squeal in automotive disk brake assemblies. Two general areas of modelling methodology are discussed. The first area of discussion focuses on generic FEA boundary conditions that are independent of the commercial code used. Examples of generic or non-code specific boundary conditions are rotor and wheel/wheel simulator or dynamometer attachment methodology and slide pin and piston penalty contact or radial coupling definitions. The second area of discussion focuses on specific boundary conditions that are commercial code dependent. An example of a specific boundary condition is the contact stiffness scale factor specification for all prestressed contact surfaces and the resulting pre-stressed normal modes analysis results. Inappropriate choices for these generic and specific boundary conditions will render the complex eigenvalue method incapable of functioning as a predictive tool for friction-induced squeal in automotive disk brake assemblies.

CITATION: Ballinger, R., "A Discussion of Complex Eigenvalue Analytical Methods as They Relate to the Prediction of Brake Noise," *SAE Int. J. Passeng. Cars - Mech. Syst.* 9(1):2016, doi:10.4271/2016-01-1299.

INTRODUCTION

This paper discusses the use of the complex eigenvalue method as a tool to predict disk brake noise. When the real parts of complex eigenvalues are positive (negative damping) the complex eigenvalues and corresponding complex mode shapes or vectors are unstable and could result in unwanted brake noise. The complex eigenvalue method has been used in many different forms for many years to attempt to predict or to correlate vehicle or dynamometer test results for brake noise when actual brake hardware exists. When no brake hardware exists - as in a new brake design - the question is, can the complex eigenvalue method be used to predict brake noise? What assurance does the analyst have that "one" single complex eigenvalue solution is truly predictive and the complex eigenvalues will correlate to the test results when the hardware is finally obtained?

BACKGROUND

It was decided that - in the absence of hardware - if two complex eigenvalue solutions are solved with two different commercial codes Abaqus/Standard and ANSYS Structural to predict the unstable modes - and the two codes agree or correlate, then at a minimum the analyst did not make any logistical or careless mistakes and the

analysis procedures are validated with respect to correlation with each other. One scenario could be a single analysis running the same complex eigenvalue model with two codes Abaqus/Standard and ANSYS Structural. Other scenarios could be extended to multiple analysts running one commercial code or multiple analysts running two or more codes. It was concluded that the finite element model boundary conditions and assumptions in each complex eigenvalue model solution must be identical if the multiple complex eigenvalue methods Abaqus/Standard and ANSYS Structural provide equivalent results and correlate to each other. More specifically, to achieve correlated results with two codes Abaqus/Standard and ANSYS Structural, contact parameters must be set up identically. This is because the basis for the complex eigenvalue method is a prestressed normal modes analysis. The prestressed or initial preload analysis portion of the prestressed normal modes analysis is highly contact dependent. It turns out that the Abaqus/Standard pre-stressed or preload analysis solution together with the default (program determined) contact parameters for the complex eigenvalue analysis provide a robust solution, but with ANSYS Structural, the prestressed or initial preload phase of the solution requires the default contact parameters be changed to match the Abaqus/Standard default contact parameter settings in order for the two codes to correlate.

COMPLEX EIGENVALUE METHOD PROCEDURE

The complex eigenvalue procedure in both Abaqus/Standard and ANSYS is basically a pre-stressed nonlinear static simulation linearized to a steady-state condition, followed by a normal modes analysis and a complex eigenvalue extraction to determine the dynamic modally-coupled instabilities that could possibly result in unwanted noise. There are basically five steps to the complete simulation procedure as follows [1]:

- Resolve contact interference (surface-to-surface contact) between the lining retention clips and the shoe tabs and apply ten percent of the target brake pressure.
- Apply the remainder of the target brake pressure. This step initializes all of the surface-to-surface preload or pre-stress contact in the brake axial direction.
- Virtually rotate the rotor contact surfaces (*Motion, Abaqus/Standard solution and Cmrrotate, ANSYS solution) with full pressure applied to translate and rotate the brake shoes into the lining retention clips supports attached to trailing ends of the carrier bracket abutments. This step initializes all of the surface-to-surface preload or pre-stress in the brake tangential direction.
- Next, the natural frequencies or normal modes of the prestressed brake system are calculated. This involves a linearization of the nonlinear contact forces at all preloaded surface-to-surface contact interfaces in the brake assembly.
- Finally, unsymmetric frictional coupling matrices between components are used to compute the complex eigenvalues and identify potentially unstable frequencies and modes of vibration that could result in or generate brake noise. This analysis combines all five steps into one seamless analysis that contains the nonlinear effects and frictional coupling without the need for manual matrix input (DMIG - Direct Matrix Input Grid) methods with matched-nodal manually created meshes on adjacent components [2].

When the real part of a complex eigenvalue becomes positive for given input conditions, the complex mode shape corresponding to that frequency is unstable and potentially could radiate undesirable brake noise.

COMPLEX EIGENVALUE METHOD FINITE ELEMENT MODEL BOUNDARY CONDITIONS

Finite element model definition and boundary conditions fall into two categories when attempting to predict brake noise. The first category contains code independent model assumptions and boundary conditions. These are generic or relative finite element model boundary conditions that are independent of or do not depend on the specific analysis code being used. The second category contains code dependent, code specific, or absolute finite element model assumptions and boundary conditions. The code dependent boundary conditions are also dictated by the pre-stressed normal mode analysis specific to and preceding the complex eigenvalue extraction.

Examples of Code Independent Boundary Conditions

Examples of code independent model assumptions and boundary conditions that can be used with either Abaqus/Standard or ANSYS generically are as follows:

Is the front steering knuckle the terminating component of the front brake assembly and should it be fixed to ground at the front strut clevis bracket attachment, the lower ball joint, and the steering arm? Or, should the steering arm and lower ball joint be pinned to ground?

Should the rotor be bonded to the hub, or bolted, or should a wheel simulator be constructed and correlated to a modal model of a wheel, and then correlated to a wheel simulator model? Or should a single-or double-ended dynamometer boundary condition be modeled?

Should a constrained-layer damping shim be included with the lining shoe - and how to model the adhesive? Should pad chamfers be modeled - or not?

Should the lining retention clips be modeled with all interferences and contact maintained - or can the pinch arms of the clips be truncated and the clips bonded or tied to the bracket abutments?

Since model geometry is normally received at nominal component position, the shoe is usually centered between the bracket abutments and lining retention clips. The centered position of the shoe creates gaps between the leading and trailing shoe tabs to the clips in the tangential direction. When the rotor is virtually rotated one of the gaps closes and the other gap widened as the shoe translates into the lining retention clip surface on the bracket abutment. This contact initiation with a large gap can cause convergence issues. These convergence issues can be minimized by indexing or translating the shoe tab to close the gap such that the shoe is already in contact position with the trailing lining retention clip on the bracket abutment (in the direction of the stop).

Can the slide pins and piston(s) be modeled with no separation contact or radial couples (surface-to-surface contact - as if there is no gap between pins and pin bores and piston and piston bores and only radially coupling) or should pin and piston gaps to pin bores and piston bores be maintained and surface-to-surface augmented Lagrange or penalty contact be established.

All of these generic or code independent boundary condition choices must be correlated to component FRF and system dynamometer hardware testing in order to develop modeling and analysis guidelines. These generic boundary conditions will not be discussed in detail in this paper. This paper is more concerned with the specific or code dependent boundary conditions such as the contact stiffness scale factor and the default specification in the commercial codes Abaqus/Standard and ANSYS Structural.

Examples of Code Dependent Boundary Conditions

Examples of code dependent model assumptions and boundary conditions that are specific to either Abaqus/Standard or ANSYS Structural and depend on which code you are using and are discussed as follows:

The element formulations chosen should contain all quadratic elements unless brake hardware exists and linear element component models have been correlated to corresponding free-free and installed component FRF modal testing. Linear tetrahedron elements should not be used in general. Specifically Abaqus/Standard C3D10 quadratic tetrahedron elements are approximately equivalent to ANSYS quadratic tetrahedron elements. Abaqus/Standard C3D20R quadratic hexahedron elements are approximately equivalent to ANSYS quadratic hexahedron elements. It should be noted that the mid-side nodes of the quadratic elements should lie on the curved surfaces of curved boundary components and not simply halfway between the chordal or straight-sided segmented geometry of the curved surface.

As component models are developed for the brake components of the system, component and subsystem free-free and installed normal modes analyses should be solved with Abaqus/Standard and ANSYS in order to prove that the initial component and subsystem finite element models are equivalent. Figure 1 depicts the subsystem finite element model of a rotor and hub assembly. Figure 2 (Appendix) depicts the Abaqus/Standard and ANSYS Structural modal frequency equivalency indicating the models are equivalent. Figure 3 depicts the subsystem finite element model of an installed carrier bracket together with the brake housing and slide pins connected to the bracket pin bores by no separation contact (radial couples). Figure 4 (Appendix) depicts the Abaqus/Standard and ANSYS Structural modal frequency equivalency indicating the subsystem models are equivalent. Material properties for all components should be consistent and equivalent between the Abaqus/Standard and ANSYS Structural solutions.

Surface-to-surface contact (augmented Lagrange or penalty) should be chosen such that the contact penetrations and contact forces under prestress or preload are approximately equivalent. This will ensure that the nonlinear contact forces - when linearized and converted to springs - will produce the same subsystem modal results. Both Abaqus/Standard and ANSYS calculate the underlying contact element stiffness for a contact pair, however Abaqus/Standard and ANSYS apply different contact stiffness scale factors in the default set-up of the pre-stress or preload initiation phase of the solution procedure. The default Abaqus/Standard contact procedure multiplies the underlying element contact stiffness (where the contact stiffness scale factor is defined as $K = 1.0$) by ten, therefore beginning the solution with a contact stiffness scale factor of 10.0 and never varying the value during the solution. The default ANSYS contact procedure or programmed determined contact stiffness determines the underlying element contact stiffness ($K = 1.0$) and essentially divides it by ten, therefore beginning the programmed determined contact solution with a contact stiffness scale factor of 0.1. If convergence problems develop, ANSYS, by default, updates (reduces) the contact stiffness scale factor at each subsequent iteration to facilitate convergence.

For Abaqus/Standard and ANSYS complex eigenvalue solutions to correlate and be realistic, the contact stiffness scale factors for each contact pair must be in the range of one to ten at final solution ensuring the contact forces, corresponding penetrations and linearized springs are equivalent. Contact stiffness scale factors of one or ten for prestressed analyses - modal or otherwise - are generally accepted values for ANSYS and Abaqus/Standard. More about this will be discussed later.

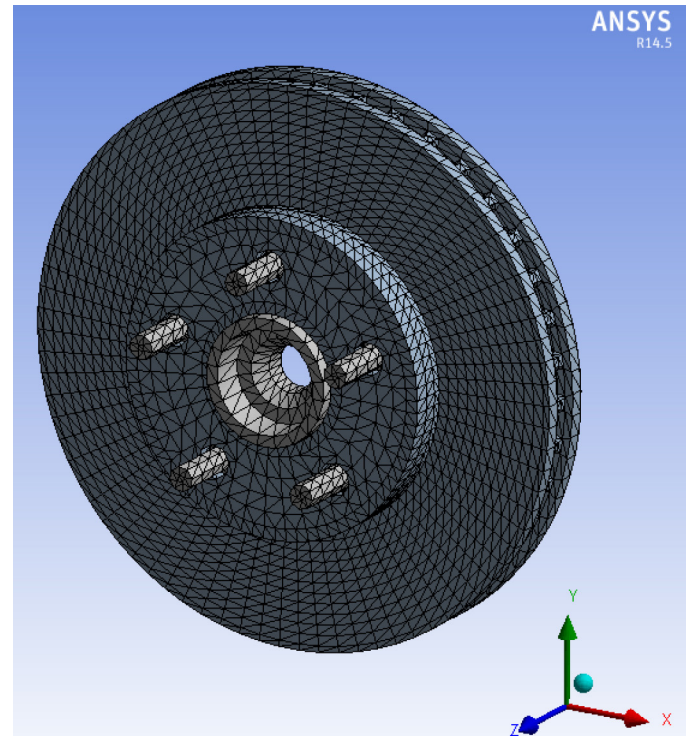


Figure 1. Subsystem model of rotor and hub for modal correlation in Abaqus/Standard and ANSYS Structural.

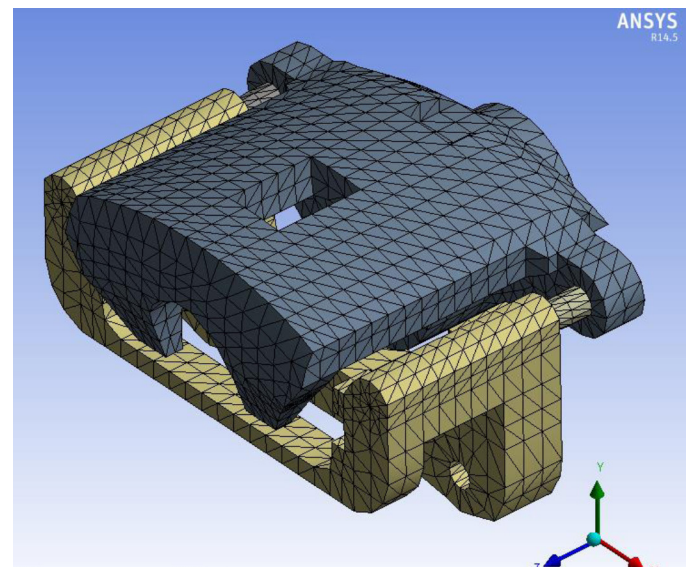


Figure 3. Subsystem model of installed bracket, housing, and slide pins having no separation contact (radial couples) to bracket pin bores for modal correlation in Abaqus/Standard and ANSYS Structural.

COMPLEX EIGENVALUE AS PREDICTIVE OR CORRELATION TOOL WHEN BRAKE HARDWARE EXISTS

When brake hardware does exist and there are brake components available for FRF testing and correlation as well as dynamometer test data or vehicle data to identify brake squeal frequencies, the use of the complex eigenvalue method to predict unstable brake systems

frequencies is possible. Two examples are presented here. Since hardware and test data exists, only one commercial code - Abaqus/Standard was used for the analyses.

System 1 Dynamometer Test/Analysis Correlation-Rear Disk Brake Hardware Exists

Figure 5 depicts the finite element model of a rear disk brake caliper supported using a double-ended dynamometer boundary condition. Component modal FRFs were acquired and correlated to component modal analyses for model validation as well as the installed rotor FRFs to the double-ended dynamometer boundary condition.

Figure 6, Figure 7, and Figure 8 (Appendix) depict three dynamometer test results plots of sound pressure level vs. frequency. Figure 9 (Appendix) depicts the Abaqus/Standard complex mode solution of System 1 negative damping values vs. frequency.

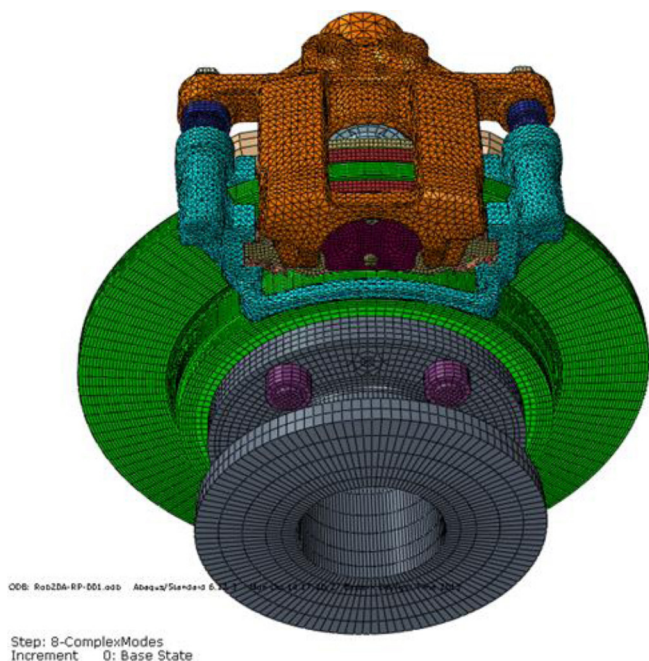


Figure 5. Rear brake System 1 hardware simulation mounted on double-ended brake dynamometer. Brake hardware and dynamometer test data exists.

System 2 Dynamometer Test/Analysis Correlation-Front Disk Brake Hardware Exists

Figure 10 and Figure 11 depict the finite element model (outboard view and inboard view) of a front disk brake caliper supported using a double-ended dynamometer boundary condition. Component modal FRFs were acquired and correlated to component modal analyses for model validation as well as the installed rotor FRFs to the double-ended dynamometer boundary condition.

Figure 12 (Appendix) depicts the dynamometer test results plot of sound pressure level vs. frequency. Figure 13 (Appendix) depicts the Abaqus/Standard complex mode solution of System 2 negative damping values vs. frequency.

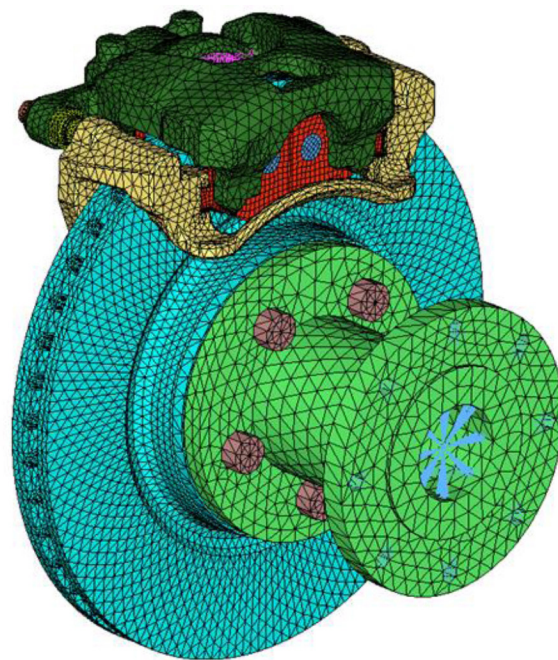


Figure 10. Front brake System 2 hardware simulation mounted on double-ended brake dynamometer - outboard view. Brake hardware and dynamometer test data exists.

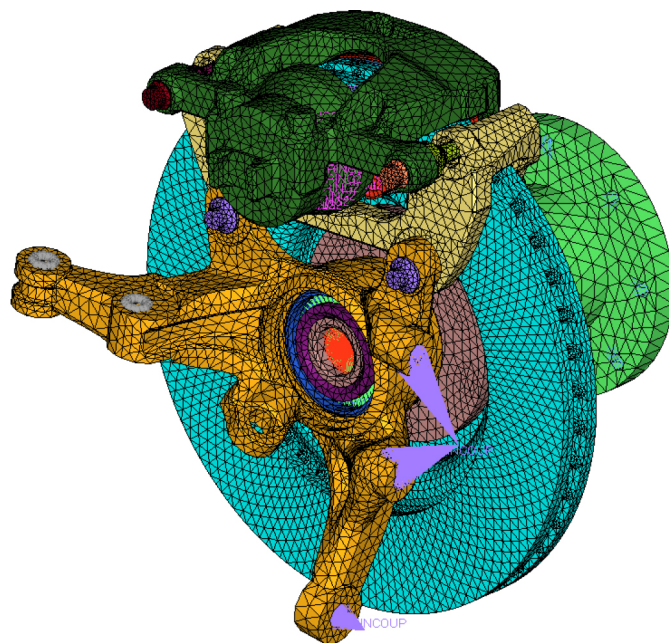


Figure 11. Front brake System 2 hardware simulation mounted on double-ended brake dynamometer - inboard view. Brake hardware and dynamometer test data exists.

COMPLEX EIGENVALUE AS PREDICTIVE TOOL WHEN HARDWARE DOES NOT EXIST

When brake hardware does not exist or there are no components to correlate FRFs nor any dynamometer data or vehicle data to correlate analysis results, the use of the complex eigenvalue method to predict brake noise becomes much more difficult, if not impossible.

For brake noise prediction without hardware to be accurate, it must be certain that finite element models and boundary and contact specifications are created and solved using rules and assumptions generated from historical data and captured knowledge from previous test correlation to existing hardware. If two different commercial codes are to be used (or if two analysts and one commercial code are used) to predict brake noise, then the two input models, assumptions, and boundary conditions must be exactly equivalent to produce identical complex eigenvalue results.

PRE-STRESSED ANALYSIS - CONTACT STIFFNESS SCALE FACTOR

The brake pressure and corresponding brake reaction torque in conjunction with specified interface coefficients of friction result in normal and tangential nonlinear forces that must be linearized in order to execute the pre-stressed normal modes analysis. Contact stiffness of all contact interfaces must be accurately specified to for solution accuracy. There are basically two choices for surface-to-surface contact specification for the pre-stressed modal analysis method - penalty contact method and augmented Lagrange method. Pure Lagrange method is not an option because of the convergence difficulty with large scale pre-stressed normal modes analysis solutions with numerous contact surfaces. Therefore, for penalty contact or augmented Lagrange contact, the values of the contact stiffness are extremely important. Too high a value of contact stiffness scale factor may lead to slow convergence and too low a value of contact stiffness scale factor, while facilitating convergence, reduces accuracy.

Abaqus/Standard Contact Stiffness Scale Factor and Defaults

Abaqus/Standard/ has both linear and nonlinear penalty contact stiffness scale factor conventions. In the linear method, as was previously described, Abaqus/Standard determines the contact the underlying interface element stiffness. This value is proprietary and is scaled to a value of one and is defined as the Abaqus/Standard contact stiffness scale factor. Before the solution is initiated, this underlying contact stiffness is multiplied by ten, such that when the Abaqus/Standard solution initiates, the contact stiffness scale factor is ten. Abaqus/Standard defaults dictate that this contact stiffness scale factor value of ten is never updated or reduced throughout the solution unless it is overridden by a manual input value. In the nonlinear method when convergence issues arise, the underlying contact stiffness at the element interface initiates the solution with a contact stiffness scale factor of 0.1 and the solution progresses until the final contact stiffness scale factor at the end of the solution increases to ten [3]. Both the linear and nonlinear contact stiffness scale factor values during solution are depicted in Figure 12.

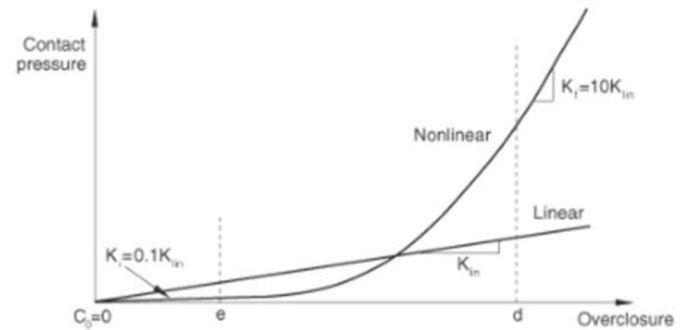


Figure 14. Abaqus/Standard comparison of linear contact stiffness scale factor ($K=1.0$) and nonlinear pressure-overclosure relationships for contact stiffness scale factor ($K=0.1$ initial and $K=10.0$ final)

ANSYS Contact Stiffness Scale Factor and Defaults

As an example, the ANSYS Contact Stiffness Scale Factor (FKN) was varied for the interference analysis of a collar on a shaft (see Figure 15) [4]. Corresponding interface von Mises stress was determined for FKN values of 0.001 to 100.0. It appears that the value of FKN = 1.0 provides sufficient accuracy [Table 1].

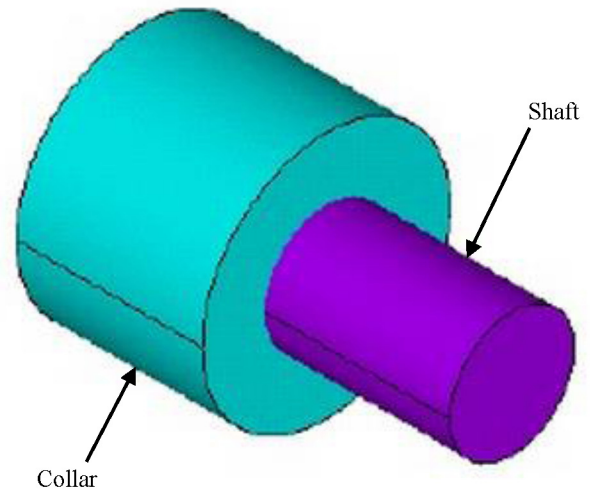


Figure 15. ANSYS example of interference fit between collar and shaft for various contact stiffness scale factor interference stress solutions.

Table 1. Contact interference maximum von Mises stress as a function of ANSYS contact stiffness scale factor for collar-shaft interference example.

ANSYS Contact Stiffness Scale Factor	Contact Interference Maximum von Mises Stress
0.001	4,000
0.01	20,000
0.1	65,000
1.0	91,000
10.0	92,900
100.0	93,000

John Lin, ANSYS, states, “for pretension problems, use a contact stiffness factor greater than one, because penetration can strongly influence the pretension forces [5].” This was demonstrated in the above example.