



104 INDUSTRIAL DRIVE
FRANKFORT, NY. 13340
TEL: 315-895-7454
FAX: 315-895-7268
WWW.C-FLEX.COM

C-FLEX BEARING CO. , INC.

PIVOT BEARING FEA SIMULATION PROCEDURE

Written by: Kevin Lake

Date: 12/6/2013

Revised: 2/4/2014

COPYRIGHT/PROPRIETARY INFORMATION NOTIFICATION

The information contained in this document is the sole property of the C-Flex Bearing Company inc. Any unauthorized reproduction or theft of the information contained within this document is expressly prohibited and may be punishable under local, state, and/or federal law.

Table of Contents

Section	Page
I. Introduction	3
II. Solid Model Creation	4
A. General	
III. Simulation Settings	5
A. Creating a New Study	
B. Duplicating a Study	
C. Simulation Settings	
IV. Material Properties	6
A. General	
V. Meshing	7
A. Creating Mesh	
VI. Running Simulation	8
A. Simulating	
VII. Displaying Results	10
A. Displays	
B. Animation	
C. Iso Clipping	
D. Section Clipping	

I. Introduction

The purpose of this document is to provide a uniform method for modeling and simulating pivot bearings produced by the C-Flex Bearing Company. The procedure outlined in this manual should be utilized by all persons attempting to perform a finite element analysis (FEA) of the pivot bearings in order to provide consistent analysis results and prevent errors that may give inaccurate results. The CAD software that this manual uses is the current company package of SOLIDWORKS® 2013 x64 edition with the SOLIDWORKS simulation professional package. Users who do not have these software packages should attempt to match as close as possible the methods and settings outlined herein using the commands and options available in their software packages.

II. Solid Model Creation

A. General

Finite element analysis utilizes a mathematical approximation which is applied to individual elements of an object to perform calculations. If a model is improperly constructed then even a properly set up simulation of it will produce errors. The most common error is to have sharp inside corners. When the software tries to model these sharp corners it will adapt and create smaller and smaller elements at the corner in an attempt to increase accuracy. An inside corner with a radius of zero will create a mathematical singularity which the computer will interpret as infinite stress and therefore give ever increasing peak stress values until the simulator eventually fails at a random point and outputs the last known values for stress. One point where this can occur is the connection between the sleeve and the core. Because of this, the solid model of the pivot bearing should be created as a single part whenever possible. Attempting to create the model as an assembly of sleeves and core will result in an improper fillet at the core/sleeve connection and produce inaccurate results. Similar caution should also be taken when the bearing is integrated into a larger assembly.

III. Simulation Settings

A. Creating a new study

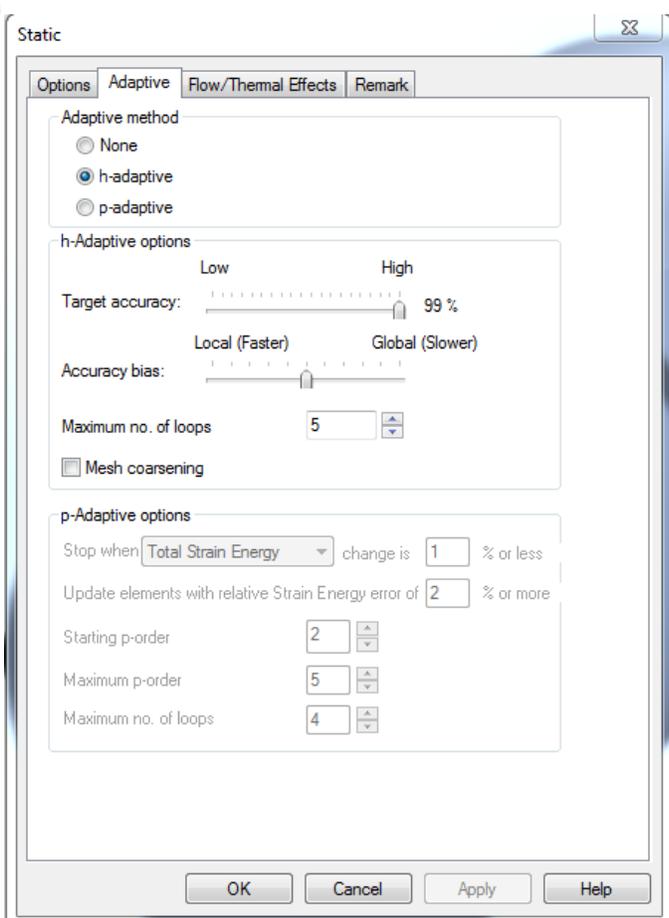
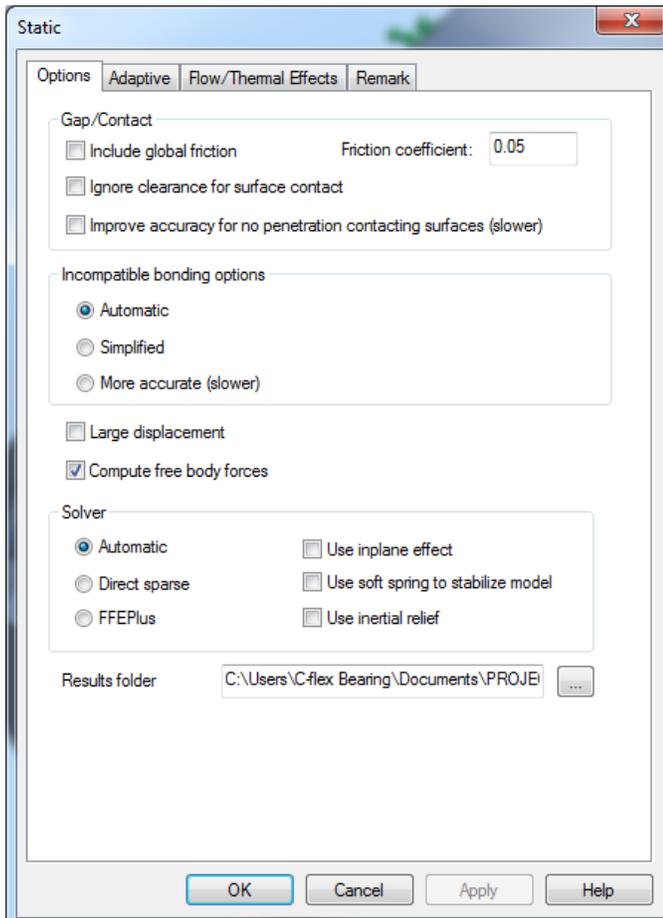
1. On the Simulation ribbon select the arrow under Study Advisor and select New Study.
2. Select the Static option and select the green check mark

B. Duplicating a study

1. Right click on an existing completed study at the bottom of the screen and select duplicate study.

C. Simulation settings

1. Select the study to be conducted at the bottom of the screen
2. Right click on the  icon in the menu to the left and select properties. Change the settings in the Options and Adaptive tabs to match as shown below. Use the h-adaptive method and make sure target accuracy is set to 99% and mesh coarsening is NOT checked. Select OK.



IV. Material Properties

A. General

1. To create a new material right click on the  icon in the left menu and select apply/edit material.
2. Right click on the custom materials folder and select New Category. Give the category a name.
3. Right click on that category and select new material. Give the material a proper name.

Since the springs are the most critical part of the entire bearing, the material they are made of (420 stainless steel) will be used for the entire model. Use the material properties as shown below:

420 Stainless Steel Simulation Material Properties	
Model type	Linear Elastic Isotropic
Units	English (IPS)
Default failure criterion	Max Von Mises Stress
Elastic modulus	29007546 psi
Poisson's ratio	0.28
Shear modulus	11167905 psi
Mass density	0.281793 lb/in ³
Tensile strength	294000 psi
Compressive strength	230000 psi
Yield strength	197000 psi
Thermal expansion coefficient	6.111111159e-006 /°F
Thermal conductivity	0.000240746
Specific heat	70.2207

Apply the material, save if necessary, and select close.

V. Meshing

A. Creating Mesh

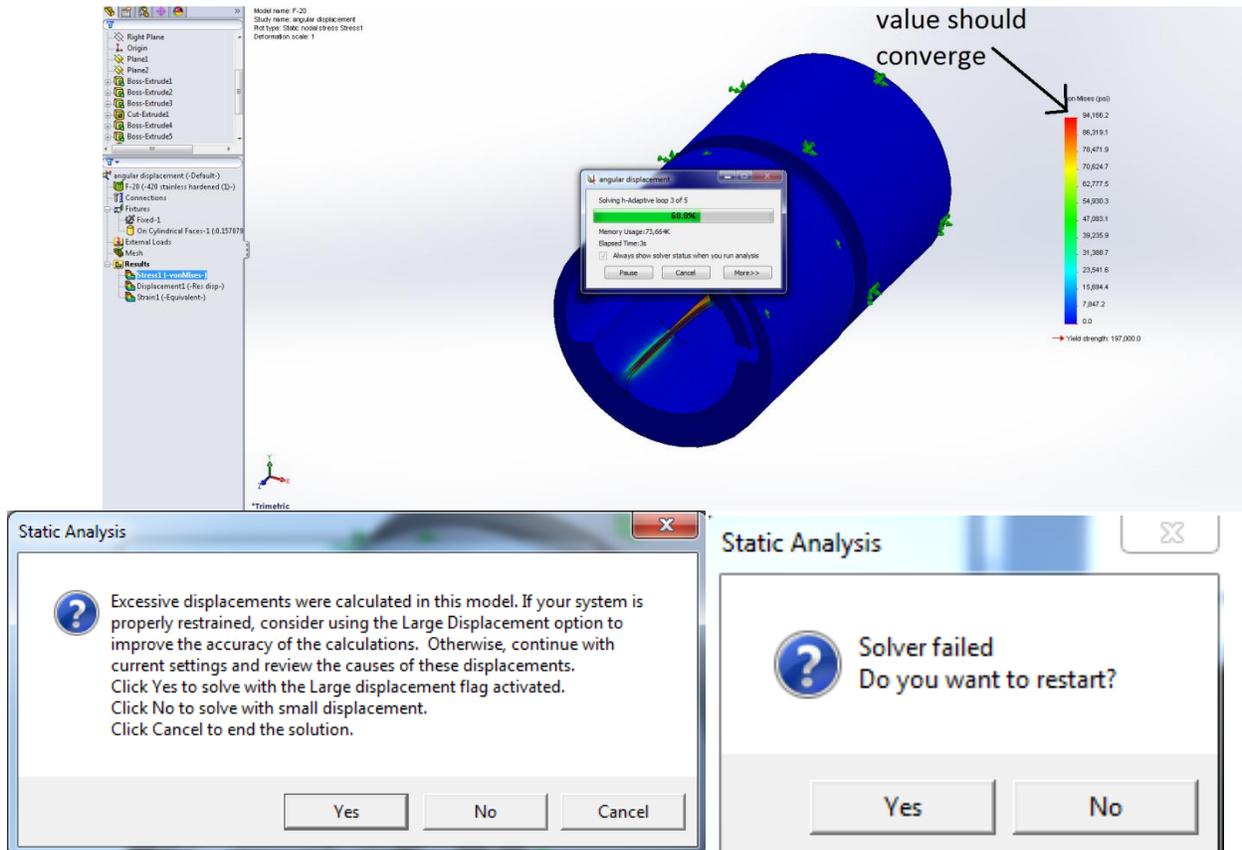
1. Right click on "Mesh" in the left menu and select "create mesh"
2. If a window pops up saying remeshing will delete results select ok
3. Expand all of the sections in the left menu
4. Set the slider under mesh density to the center of the scale
5. Under "Mesh Parameters" select "curvature based mesh", use the default values
6. Under "Advanced" make sure Jacobian points is set to "At Nodes"
7. Under "Options" select "Run (solve) the analysis"
8. Select the green check mark, the simulation will start running.

VI. Running Simulation

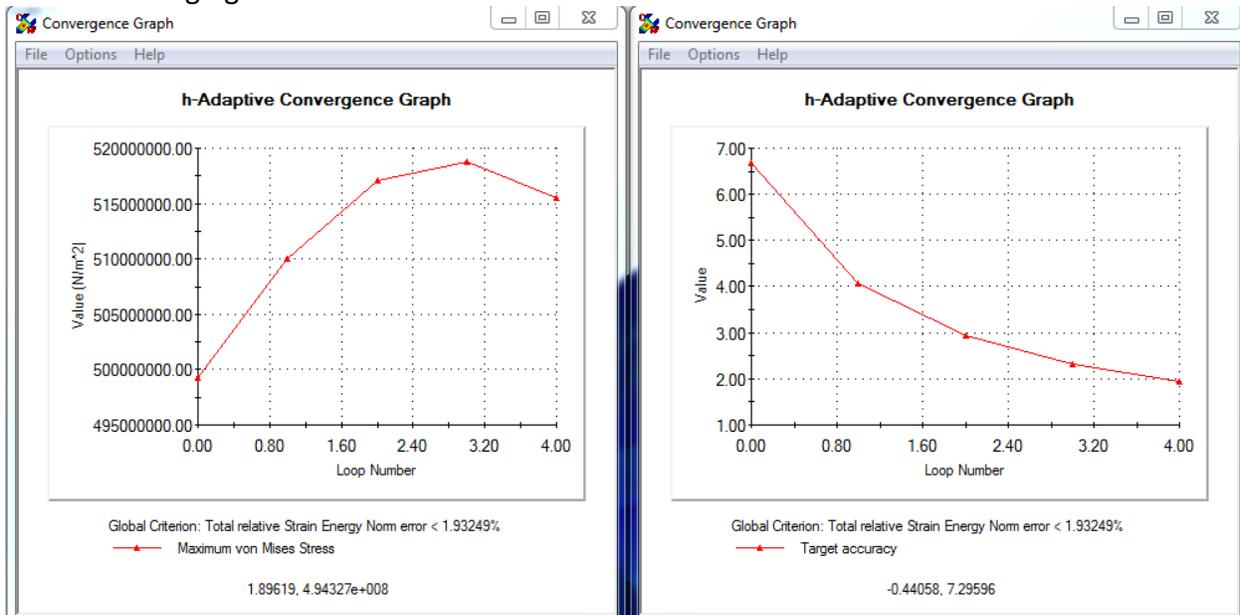
Note: Always rebuild the model ( button), save and re-mesh using the procedure in section VI before solving, even if the model is already meshed from a previous analysis. Otherwise the adaptive meshing process will result in a super fine mesh that will take excessively long to solve.

A. Simulation

1. After completing the meshing procedure the simulation will run automatically. The simulator will first mesh the model, then it will iterate solving the simulation 5 times. Do not click on the Solidworks screen, try to use Solidworks or in any other way disturb Solidworks or the program may freeze. A window will pop up for each iteration and disappear when that iteration finishes, then the next iteration will start after a potentially long pause. Make sure that all 5 iterations have finished before attempting to use Solidworks. Watch the results as they change between iterations, the values should converge towards a specific value. If the values do not converge but just keep growing rapidly then the solution will not be correct, likely because a fillet was left out and the program is trying to solve an infinite solution. If a window pops up asking to use the large displacement option select yes. If the simulation fails and it asks to start over select YES.



- Once the simulation has finished right click on Results in the left menu and select "Define Adaptive Convergence Graph". Make "Maximum Von Mises Stress" the only checked box and select the green check. A graph of the maximum stress value from each iteration will be made. The values will likely jump a lot the first couple iterations but should begin to stabilize after that. If the values do not stabilize then go back and check the model for missing fillets or other irregularities.
- Repeat step #3 but only check "Target Accuracy" this time. The values should be converging toward the bottom.



- 5 iterations will usually give reasonably good results. If more accuracy is required then without remeshing or changing anything press the "Run" button in the Simulation ribbon. This will run the simulation for 5 more iterations or until the target accuracy of 1% is reached, whichever occurs first. This second round of simulations will take longer than the first round to complete due to the increasingly refined mesh which consumes more computational resources. A third round is not recommended as there are diminishing returns with continued iterations and solution time will be drastically increased.
- When viewing Von Mises stress, red and orange sections should be clearly visible (assuming that the high stress areas are in a visible location and not hidden behind something else). If the highest value color visible over the entire model is only green or blue then there is likely an infinite stress somewhere small which is skewing the color plot. The color of values are proportional to the lowest value (blue) and the highest value (red) and are linear in between. Colors cannot be compared between different results as red may equal 50kpsi on one plot and 150kpsi on another. If most of a model is 20kpsi and one microscopic point is 1000 kpsi then the entire model will appear blue and therefore not provide much information. If this occurs check the model for missing fillets etc. and simulate again.

VII. Displaying Results

A. Displays

1. To display different results like stress or displacement double click on the type under results in the left menu.
2. To alter the display properties such as color or deformation right click on the type and select "edit definition" or "chart options".

B. Animation

1. Double click on the type of display desired (stress, displacement, etc)
2. Right click on the type and select animate.
3. The program will generate a fast animation automatically. When it starts playing press the stop button.
4. Alter the # of frames and framerate as desired and select a repeat option (loop, etc)
5. To save the animation select "Save as AVI file"
6. Alter the AVI options and save location as desired.
7. Select the green arrow to create/save the animation

C. Iso Clipping- Only displays parts of the models whose values are within a range

1. In the simulation ribbon select "Plot Tools" and select "Iso Clipping"
2. Enter a value in the box, only the sections of the model with a stress/disp/etc value larger than that will be displayed.
3. Select the opposing arrows button to reverse the direction and only display sections with values below that value.
4. Select the "clipping on/off" button at the bottom of the menu to turn Iso clipping off.

D. Section Clipping

1. In the simulation ribbon select "Plot Tools" and select "Section Clipping"
2. Select the button for the type of section (plane, cylinder or sphere)
3. Select the reference entity box then expand the drop down menu on the main view screen and select the front, top or right plane.
4. Adjust the section values as desired
5. Select the "clipping on/off" button at the bottom of the menu to turn Section clipping off.

