Model created in COMSOL Multiphysics 6.4

Validation of Dynamic Coefficients in Hydrodynamic Bearings

Introduction

In this verification example, we will validate the built-in functionality for calculating dynamic coefficients, specifically, the stiffness and damping coefficients of hydrodynamic bearings. The verification is carried out on a generic cylindrical hydrodynamic journal bearing with a diameter-to-width ratio of 1.

The dynamic coefficients are first determined using the *infinitesimal perturbation* method available within the Hydrodynamic Bearing interface. Then, the dynamic coefficients are computed using the *finite perturbation* approach. Finally, the results from both methods are compared to evaluate consistency and accuracy.

Model Definition

The hydrodynamic bearing is modeled by a single cylindrical surface. No fluid domains are required because the thickness of the fluid film is modeled implicitly.

The journal has a width and diameter of 20 cm, with a constant bearing clearance of 100 μm. The runner spins at a constant speed of 1000 rad/s, and is subjected to an external load ranging from 100 N to 1 MN. The lubricant viscosity is assumed to follow the isothermal pressure–viscosity relation proposed by Barus (Ref. 1):

$$\mu = \mu_0 e^{\xi p}$$

where μ_0 is the viscosity at zero pressure, ξ is a lubricant-dependent pressure–viscosity coefficient, and p is the fluid pressure. Additionally, it is assumed that the density can be described by the isothermal pressure–density relation suggested by Dowson and Higgison (Ref. 1):

$$\rho = \rho_0\left(1 + \frac{0.6p}{1.7p + 10^9\,\mathrm{Pa}}\right)$$

where ρ_0 is the density at zero pressure. The parameters used in the model are summarized in Table 1.

TABLE 1: FLUID PARAMETERS.

Parameter	**Value**	**Unit**
μ_0	75	mPa·s
ξ	25	GPa^{-1}
ρ_0	860	kg/m^3

Linearized dynamic coefficients of a fluid-film bearing can be determined using different approaches. This includes *infinitesimal perturbation* and *finite perturbation*, where the latter is sometime also referred to as *numerical perturbation*. In the Hydrodynamic Bearing interface, the computed dynamic coefficients are based on the *infinitesimal perturbation* approach.

The linearized stiffness and damping coefficients are defined as

$$\mathbf{k} = -\frac{\partial \mathbf{F}_\mathrm{j}}{\partial \mathbf{u}_\mathrm{j}} \qquad \mathbf{c} = -\frac{\partial \mathbf{F}_\mathrm{j}}{\partial \mathbf{v}_\mathrm{j}}$$

where $\mathbf{k}$ and $\mathbf{c}$ are the stiffness and damping matrix, $\mathbf{u}_\mathrm{j}$ and $\mathbf{v}_\mathrm{j}$ are the displacement and velocity of the journal, and $\mathbf{F}_\mathrm{j}$ is the total fluid force on the journal.

If the infinitesimal variations are replaced by finite differences, the stiffness and damping coefficients can be approximated by

$$\mathbf{k} \approx -\frac{\Delta \mathbf{F}_\mathrm{j}}{\Delta \mathbf{u}_\mathrm{j}} = -\frac{\mathbf{F}_{\delta u} - \mathbf{F}_{\mathrm{j}0}}{\delta u} \qquad \mathbf{c} \approx -\frac{\Delta \mathbf{F}_\mathrm{j}}{\Delta \mathbf{v}_\mathrm{j}} = -\frac{\mathbf{F}_{\delta v} - \mathbf{F}_{\mathrm{j}0}}{\delta v}$$

where $\mathbf{F}_{\mathrm{j}0}$ is the total fluid force without the perturbation, δu and δv are the finite perturbations of the displacement and velocity, while $\mathbf{F}_{\delta \mathrm{u}}$ and $\mathbf{F}_{\delta \mathrm{v}}$ are the total fluid force including the perturbations.

This is the approach used for the finite perturbation.

Results and Discussion

Figure 1 compares the stiffness coefficients obtained using the built-in functionality with those calculated through *finite perturbation*. The results show excellent agreement, with the two methods matching to a very high degree of precision.

Similarly, Figure 2 shows the corresponding comparison of the associated damping coefficients. Once again, the results demonstrate a very close match between the two approaches.

Figure 3 illustrates a comparison between the pressure perturbation determined with the built-in functionality and the finite perturbation approach. The pressure perturbations obtained with the two distinct approaches are visually indistinguishable.

The small deviations observed can be attributed to the perturbation sizes selected for the finite perturbation method. In general, it is recommended that a convergence study is performed to determine proper perturbation sizes.

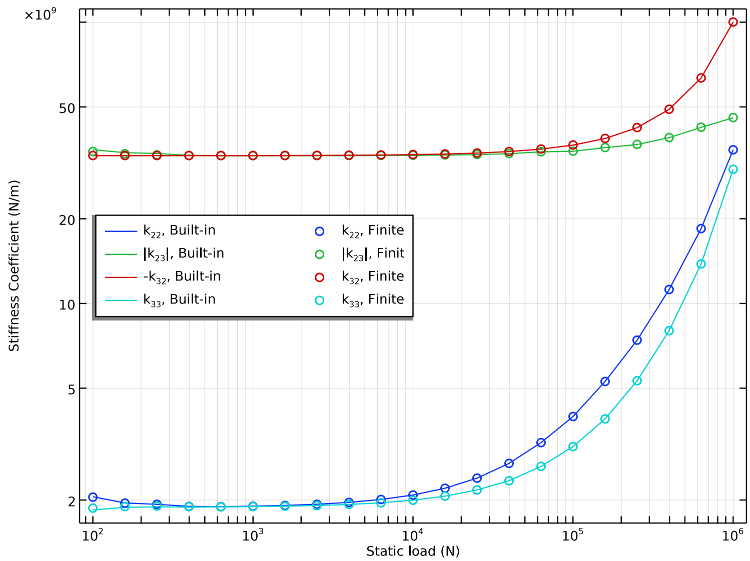


Figure 1: Comparison of stiffness coefficients.

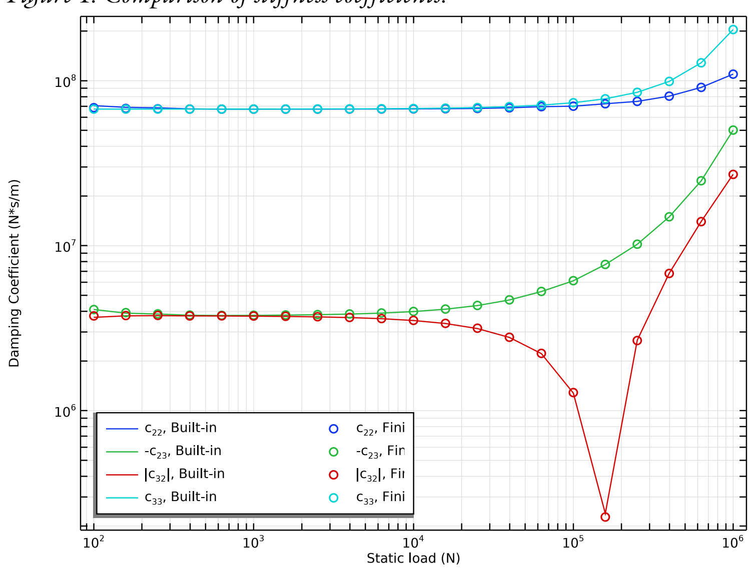


Figure 2: Comparison of damping coefficients.

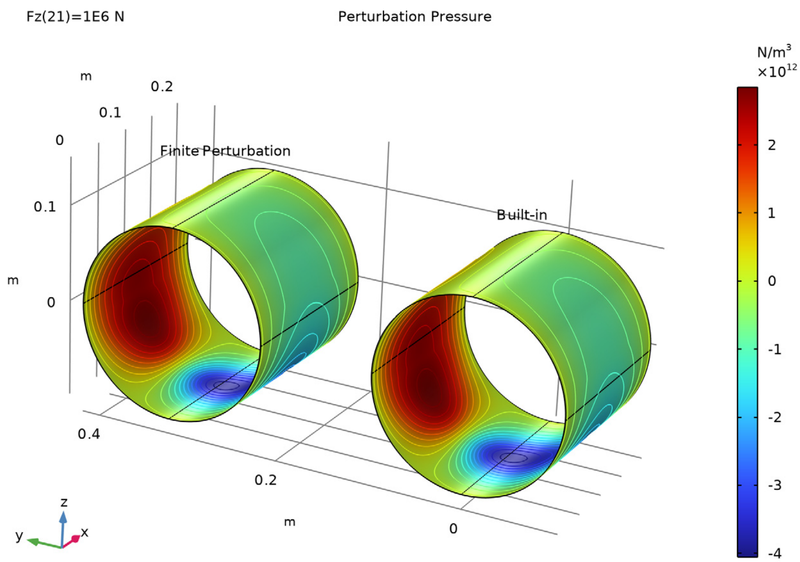


Figure 3: Comparison of perturbation pressure due to displacement perturbation in the z direction.

Reference

1. B.J. Hamrock, S.R. Schmid, and B.O. Jacobson, *Fundamentals of Fluid Film Lubrications*, Marcel Dekker, 2004.

Application Library path: `Rotordynamics_Module/Verification_Examples/dynamic_coefficients_finite_perturbation`

Modeling Instructions

The first step in building the model is to add the required physics interface and study. This model uses a Hydrodynamic Bearing interface and a stationary study.

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics** > **Rotordynamics** > **Hydrodynamic Bearing (hdb)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Next, define the parameters that are needed for setting up the model.

Bearing Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, type `Bearing Parameters` in the **Label** text field.

3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
C	100[um]	1E-4 m	Clearance
mu0	75[mPa*s]	0.075 Pa·s	Viscosity at zero pressure
xi	25e-9[m^2/N]	2.5E-8 1/Pa	Viscosity-pressure coefficient
rho0	860[kg/m^3]	860 kg/m³	Density at zero pressure
omega	1000[rad/s]	1000 rad/s	Rotational velocity
R	10[cm]	0.1 m	Radius of the journal
W	2*R	0.2 m	Width of the journal
Fz	1000[N]	1000 N	Static load

Perturbation Parameters

Add an additional parameter node for the parameters required to perform the finite perturbation.

1 In the **Home** toolbar, click Pi **Parameters** and choose **Add** > **Parameters**.

2 In the **Settings** window for **Parameters**, type `Perturbation Parameters` in the **Label** text field.

3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
u_pert	0.1[nm]	1E-10 m	Displacement perturbation
v_pert	u_pert*omega	1E-7 m/s	Velocity perturbation
acuy	0	0	Activation of displacement perturbation in y direction
acuz	0	0	Activation of displacement perturbation in z direction
acvy	0	0	Activation of velocity perturbation in y direction
acvz	0	0	Activation of velocity perturbation in z direction

4 In the **Home** toolbar, click Pi **Parameter Case**.

5 In the **Settings** window for **Case**, type `Displacement Perturbation (Y)` in the **Label** text field.

6 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Description
acuy	1	Activation of displacement perturbation in y direction

7 In the **Home** toolbar, click Pi **Parameter Case**.

8 In the **Settings** window for **Case**, type `Displacement Perturbation (Z)` in the **Label** text field.

9 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Description
acuz	1	Activation of displacement perturbation in z direction

10 In the **Home** toolbar, click Pi **Parameter Case**.

11 In the **Settings** window for **Case**, type `Velocity Perturbation (Y)` in the **Label** text field.

12 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Description
acvy	1	Activation of velocity perturbation in y direction

13 In the **Home** toolbar, click **Parameter Case**.

14 In the **Settings** window for **Case**, type `Velocity Perturbation (Z)` in the **Label** text field.

15 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Description
acvz	1	Activation of velocity perturbation in z direction

16 In the **Home** toolbar, click **Parameter Case**.

17 In the **Settings** window for **Case**, type `Verification` in the **Label** text field.

GEOMETRY 1

Create a geometry for the bearing.

Cylinder 1 (cyl1)

1 In the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Object Type** section.

3 From the **Type** list, choose **Surface**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `R`.

5 In the **Height** text field, type `W`.

6 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

MATERIALS

Next, add a material node to specify the density and viscosity of the fluid.

Oil

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Oil` in the **Label** text field.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Dynamic viscosity	mu	`mu0*exp(xi*hdb.p)`	Pa·s	Basic
Density	rho	`rho0*(1+0.6*hdb.p/(1.7*hdb.p+1e9))`	kg/m³	Basic

HYDRODYNAMIC BEARING (HDB)

Set up the Hydrodynamic Bearing node, starting by activating the determination of dynamic coefficients.

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Hydrodynamic Bearing (hdb)**.
2. In the **Settings** window for **Hydrodynamic Bearing**, locate the **Dynamic Coefficients** section.
3. Select the **Calculate dynamic coefficients** checkbox.

Hydrodynamic Journal Bearing 1

Set up the Hydrodynamic Bearing node to perform a static analysis of the hydrodynamic bearing based on an external load.

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Hydrodynamic Bearing (hdb)** click **Hydrodynamic Journal Bearing 1**.
2. In the **Settings** window for **Hydrodynamic Journal Bearing**, locate the **Bearing Properties** section.
3. In the C text field, type `C`.
4. From the $\mathbf{X}_c$ list, choose **From geometry**.
5. Locate the **Journal Properties** section. From the **Specify** list, choose **Load**.
6. Specify the $\mathbf{W}_j$ vector as

`-Fz`	z

7. In the Ω text field, type `omega`.

MESH 1

Next, create a structured mesh for the bearing.

Mapped 1

1. In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.
2. In the **Settings** window for **Mapped**, locate the **Boundary Selection** section.
3. From the **Geometric entity level** list, choose **Remaining**.

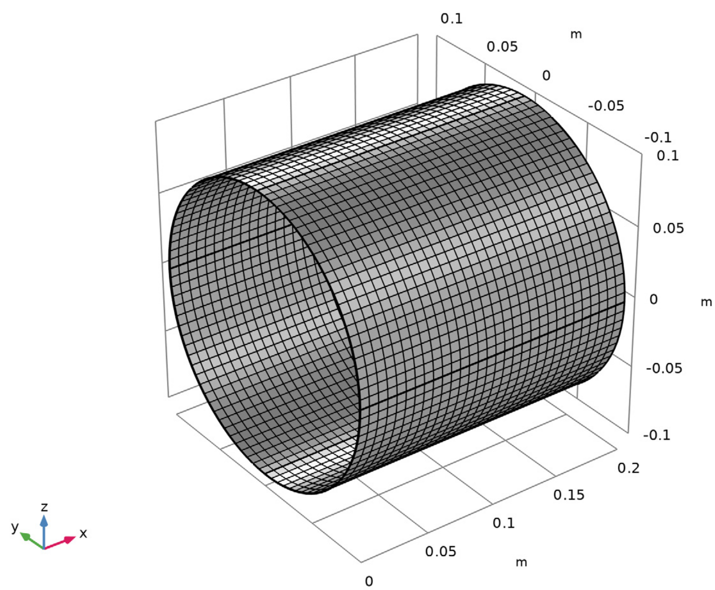
Distribution 1

1. Right-click **Mapped 1** and choose **Distribution**.
2. Select Edges 1, 2, 4, and 6 only.
3. In the **Settings** window for **Distribution**, locate the **Distribution** section.
4. In the **Number of elements** text field, type 24.

Distribution 2

1. In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
2. Select Edge 3 only.
3. In the **Settings** window for **Distribution**, locate the **Distribution** section.
4. In the **Number of elements** text field, type 30.
5. Click **Build All**.

   The mesh should look similar to the mesh shown below.



STUDY: INFINITESIMAL PERTURBATION

Now, set up the stationary study. Use an auxiliary sweep to analyze the dynamic coefficients over a range of external loads.

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for **Study**, type `Study: Infinitesimal Perturbation` in the **Label** text field.

Step 1: Stationary

1 In the **Model Builder** window, under **Study: Infinitesimal Perturbation** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Fz (Static load)	`10^range(2,0.2,6)`	N

6 In the **Study** toolbar, click **Compute**.

RESULTS

Follow the instruction below to create a plot showing how the stiffness coefficients evolve as function of the external load.

Stiffness Coefficients

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Stiffness Coefficients` in the **Label** text field.

3 Locate the **Plot Settings** section.

4 Select the **x-axis label** checkbox. In the associated text field, type `Static load (N)`.

5 Select the **y-axis label** checkbox. In the associated text field, type `Stiffness Coefficient (N/m)`.

6 Locate the **Legend** section. From the **Position** list, choose **Middle left**.

Built-in

1 Right-click **Stiffness Coefficients** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	**Unit**	**Description**
`hdb.hjb1.k22`	`N/m`	`k<sub>22</sub>, Built-in`
`abs(hdb.hjb1.k23)`	`N/m`	`|k<sub>23</sub>|, Built-in`
`-hdb.hjb1.k32`	`N/m`	`-k<sub>32</sub>, Built-in`
`hdb.hjb1.k33`	`N/m`	`k<sub>33</sub>, Built-in`

4 In the **Label** text field, type `Built-in`.

5 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

6 Click the **y-Axis Log Scale** button in the **Graphics** toolbar.

7 In the **Stiffness Coefficients** toolbar, click **Plot**.

Damping Coefficients

Now, create a similar plot for the damping coefficients.

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **x-axis label** checkbox. In the associated text field, type `Static load (N)`.

4 Select the **y-axis label** checkbox. In the associated text field, type `Damping Coefficient (N*s/m)`.

5 In the **Label** text field, type `Damping Coefficients`.

6 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

Built-in

1 Right-click **Damping Coefficients** and choose **Global**.

2 In the **Settings** window for **Global**, type `Built-in` in the **Label** text field.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
`hdb.hjb1.c22`	`N*s/m`	`c<sub>22</sub>, Built-in`
`-hdb.hjb1.c23`	`N*s/m`	`-c<sub>23</sub>, Built-in`
`abs(hdb.hjb1.c32)`	`N*s/m`	`|c<sub>32</sub>|, Built-in`
`hdb.hjb1.c33`	`N*s/m`	`c<sub>33</sub>, Built-in`

4 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

5 Click the **y-Axis Log Scale** button in the **Graphics** toolbar.

6 In the **Damping Coefficients** toolbar, click **Plot**.

HYDRODYNAMIC BEARING (HDB)

To validate the dynamic coefficients determined by the built-in functionality, perform a finite perturbation of the fluid pressure. Start by deactivating the automatic determination of dynamic coefficients.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Hydrodynamic Bearing (hdb)**.

2 In the **Settings** window for **Hydrodynamic Bearing**, locate the **Dynamic Coefficients** section.

3 Clear the **Calculate dynamic coefficients** checkbox.

Hydrodynamic Journal Bearing 1

Add a copy of the existing Hydrodynamic Journal Bearing node.

Hydrodynamic Journal Bearing 2

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Hydrodynamic Bearing (hdb)** right-click **Hydrodynamic Journal Bearing 1** and choose **Duplicate**.

Make use of the `withsol()` operator to pick up the displacement and velocities from the first study. In addition, add a small perturbation multiplied by the activation condition.

2 In the **Settings** window for **Hydrodynamic Journal Bearing**, locate the **Journal Properties** section.

3 From the **Specify** list, choose **Displacement**.

4 Specify the $\mathbf{u}_j$ vector as

`withsol('sol1',hdb.uJx,setval(Fz,Fz))`	x
`withsol('sol1',hdb.uJy,setval(Fz,Fz)) + acuy*u_pert`	y
`withsol('sol1',hdb.uJz,setval(Fz,Fz)) + acuz*u_pert`	z

5 From the **Velocity of the journal** list, choose **Velocity field**.

6 Specify the $\mathbf{v}_j$ vector as

`withsol('sol1',hdb.vJx,setval(Fz,Fz))`	x
`withsol('sol1',hdb.vJy,setval(Fz,Fz)) + acvy*v_pert`	y
`withsol('sol1',hdb.vJz,setval(Fz,Fz)) + acvz*v_pert`	z

ADD STUDY

Add another stationary study to use for performing the finite perturbation.

1 In the **Study** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Stationary**.

4 Click the **Add Study** button in the window toolbar.

5 In the **Study** toolbar, click **Add Study** to close the **Add Study** window.

STUDY: INFINITESIMAL PERTURBATION

Now, disable the second Hydrodynamic Journal Bearing node in the first study. This allows you to recompute the study correctly.

Step 1: Stationary

1 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

2 Select the **Modify model configuration for study step** checkbox.

3 In the tree, select **Component 1 (comp1)** > **Hydrodynamic Bearing (hdb)** > **Hydrodynamic Journal Bearing 2**.

4 Click **Disable**.

STUDY: FINITE PERTURBATION

Set up the second stationary study similarly to the first.

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study: Finite Perturbation` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

1 In the **Model Builder** window, under **Study: Finite Perturbation** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Study Extensions** section.

3 Select the **Auxiliary sweep** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Fz (Static load)	`10^range(2,0.2,6)`	N

Parametric Sweep

Now, add a parametric sweep to use for looping over the parameter cases.

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 From the **Sweep type** list, choose **Parameter switch**.

4 Click **Add**.

5 In the table, enter the following settings:

Switch	Cases	Case numbers
Perturbation Parameters	All	range(1,1,5)

6 In the **Study** toolbar, click **Compute**.

RESULTS

Next, follow the instructions below to determine the stiffness coefficients based on finite perturbation.

Stiffness Coefficient, k22 (Finite Perturbation)

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `Stiffness Coefficient, k22 (Finite Perturbation)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Finite Perturbation/ Solution 2 (sol2)**.

Global Evaluation 1

1 Right-click **Stiffness Coefficient, k22 (Finite Perturbation)** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`hdb.hjb2.Fjy`	`N`	`Fluid load on journal, y-component`

Global Evaluation 2

1 Right-click **Global Evaluation 1** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 From the **Dataset** list, choose **Study: Finite Perturbation/Parametric Solutions 1 (sol3)**.

4 From the **Perturbation Parameters** list, choose **From list**.

5 In the **Perturbation Parameters** list box, select **Displacement Perturbation (Y)**.

Stiffness Coefficient, k22 (Finite Perturbation)

1 In the **Model Builder** window, click **Stiffness Coefficient, k22 (Finite Perturbation)**.

2 In the **Settings** window for **Evaluation Group**, locate the **Transformation** section.

3 From the **Transformation type** list, choose **General**.

4 In the **Expression** text field, type -(gev2-gev1)/u_pert.

5 In the **Column header** text field, type k₂₂, Finite Pert..

6 In the **Stiffness Coefficient, k22 (Finite Perturbation)** toolbar, click **Evaluate**.

Stiffness Coefficient, k23 (Finite Perturbation)

1 Right-click **Stiffness Coefficient, k22 (Finite Perturbation)** and choose **Duplicate**.

2 In the **Settings** window for **Evaluation Group**, type Stiffness Coefficient, k23 (Finite Perturbation) in the **Label** text field.

3 Locate the **Transformation** section. In the **Expression** text field, type abs(-(gev2-gev1)/u_pert).

4 In the **Column header** text field, type |k₂₃|, Finite Pert..

Global Evaluation 2

1 In the **Model Builder** window, expand the **Stiffness Coefficient, k23 (Finite Perturbation)** node, then click **Global Evaluation 2**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 In the **Perturbation Parameters** list box, select **Displacement Perturbation (Z)**.

4 In the **Stiffness Coefficient, k23 (Finite Perturbation)** toolbar, click **Evaluate**.

Stiffness Coefficient, k32 (Finite Perturbation)

1 In the **Model Builder** window, right-click **Stiffness Coefficient, k22 (Finite Perturbation)** and choose **Duplicate**.

2 In the **Model Builder** window, click **Stiffness Coefficient, k22 (Finite Perturbation) 1**.

3 In the **Settings** window for **Evaluation Group**, type Stiffness Coefficient, k32 (Finite Perturbation) in the **Label** text field.

4 Locate the **Transformation** section. In the **Expression** text field, type (gev2-gev1)/u_pert.

5 In the **Column header** text field, type k₃₂, Finite Pert..

Global Evaluation 1

1 In the **Model Builder** window, click **Global Evaluation 1**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
hdb.hjb2.Fjz	N	Fluid load on journal, z-component

Global Evaluation 2

1 In the **Model Builder** window, click **Global Evaluation 2**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
hdb.hjb2.Fjz	N	Fluid load on journal, z-component

4 In the **Stiffness Coefficient, k32 (Finite Perturbation)** toolbar, click **Evaluate**.

Stiffness Coefficient, k33 (Finite Perturbation)

1 In the **Model Builder** window, right-click **Stiffness Coefficient, k23 (Finite Perturbation)** and choose **Duplicate**.

2 In the **Settings** window for **Evaluation Group**, type Stiffness Coefficient, k33 (Finite Perturbation) in the **Label** text field.

3 Locate the **Transformation** section. In the **Expression** text field, type -(gev2-gev1)/u_pert.

4 In the **Column header** text field, type k₃₃, Finite Pert..

Global Evaluation 1

1 In the **Model Builder** window, expand the **Stiffness Coefficient, k33 (Finite Perturbation)** node, then click **Global Evaluation 1**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
hdb.hjb2.Fjz	N	Fluid load on journal, z-component

Global Evaluation 2

1 In the **Model Builder** window, click **Global Evaluation 2**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`hdb.hjb2.Fjz`	`N`	`Fluid load on journal, z-component`

4 In the **Stiffness Coefficient, k33 (Finite Perturbation)** toolbar, click **Evaluate**.

Stiffness Coefficients

To compare the stiffness coefficients obtained with the built-in functionality and finite perturbation, add the stiffness coefficients to the existing plot.

1 In the **Model Builder** window, under **Results** click **Stiffness Coefficients**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 In the **Number of columns** text field, type `2`.

k22 (Finite Perturbation)

1 Right-click **Stiffness Coefficients** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Source** list, choose **Evaluation group**.

4 In the **Label** text field, type `k22 (Finite Perturbation)`.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

6 From the **Color** list, choose **Cycle (reset)**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

8 Click to expand the **Legends** section. Select the **Show legends** checkbox.

9 In the **Stiffness Coefficients** toolbar, click **Plot**.

k23 (Finite Perturbation)

1 Right-click **k22 (Finite Perturbation)** and choose **Duplicate**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Evaluation group** list, choose **Stiffness Coefficient, k23 (Finite Perturbation)**.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle**.

5 In the **Label** text field, type `k23 (Finite Perturbation)`.

k32 (Finite Perturbation)

1 Right-click **k23 (Finite Perturbation)** and choose **Duplicate**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Evaluation group** list, choose **Stiffness Coefficient, k32 (Finite Perturbation)**.

4 In the **Label** text field, type k32 (Finite Perturbation).

k33 (Finite Perturbation)

1 Right-click **k32 (Finite Perturbation)** and choose **Duplicate**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Evaluation group** list, choose **Stiffness Coefficient, k33 (Finite Perturbation)**.

4 In the **Label** text field, type k33 (Finite Perturbation).

5 In the **Stiffness Coefficients** toolbar, click **Plot**.

Stiffness Coefficient, k22 (Finite Perturbation)

Now, create additional evaluation groups to determine the damping coefficients.

Damping Coefficient, c22 (Finite Perturbation)

1 In the **Model Builder** window, right-click **Stiffness Coefficient, k22 (Finite Perturbation)** and choose **Duplicate**.

2 In the **Settings** window for **Evaluation Group**, type Damping Coefficient, c22 (Finite Perturbation) in the **Label** text field.

3 Locate the **Transformation** section. In the **Expression** text field, type -(gev2-gev1)/v_pert.

4 In the **Column header** text field, type c₂₂, Finite Pert..

Global Evaluation 2

1 In the **Model Builder** window, expand the **Damping Coefficient, c22 (Finite Perturbation)** node, then click **Global Evaluation 2**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 In the **Perturbation Parameters** list box, select **Velocity Perturbation (Y)**.

4 In the **Damping Coefficient, c22 (Finite Perturbation)** toolbar, click **Evaluate**.

Damping Coefficient, c23 (Finite Perturbation)

1 In the **Model Builder** window, right-click **Stiffness Coefficient, k23 (Finite Perturbation)** and choose **Duplicate**.

2 In the **Settings** window for **Evaluation Group**, type Damping Coefficient, c23 (Finite Perturbation) in the **Label** text field.

3 Locate the **Transformation** section. In the **Expression** text field, type (gev2-gev1)/v_pert.

4 In the **Column header** text field, type -c₂₃, Finite Pert..

Global Evaluation 2

1 In the **Model Builder** window, expand the **Damping Coefficient, c23 (Finite Perturbation)** node, then click **Global Evaluation 2**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 In the **Perturbation Parameters** list box, select **Velocity Perturbation (Z)**.

4 In the **Damping Coefficient, c23 (Finite Perturbation)** toolbar, click **Evaluate**.

Damping Coefficient, c32 (Finite Perturbation)

1 In the **Model Builder** window, right-click **Stiffness Coefficient, k32 (Finite Perturbation)** and choose **Duplicate**.

2 In the **Settings** window for **Evaluation Group**, type Damping Coefficient, c32 (Finite Perturbation) in the **Label** text field.

3 Locate the **Transformation** section. In the **Expression** text field, type abs((gev2-gev1)/v_pert).

4 In the **Column header** text field, type |c₃₂|, Finite Pert..

Global Evaluation 2

1 In the **Model Builder** window, expand the **Damping Coefficient, c32 (Finite Perturbation)** node, then click **Global Evaluation 2**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 In the **Perturbation Parameters** list box, select **Velocity Perturbation (Y)**.

4 In the **Damping Coefficient, c32 (Finite Perturbation)** toolbar, click **Evaluate**.

Damping Coefficient, c33 (Finite Perturbation)

1 In the **Model Builder** window, right-click **Stiffness Coefficient, k33 (Finite Perturbation)** and choose **Duplicate**.

2 In the **Settings** window for **Evaluation Group**, type Damping Coefficient, c33 (Finite Perturbation) in the **Label** text field.

3 Locate the **Transformation** section. In the **Expression** text field, type -(gev2-gev1)/v_pert.

4 In the **Column header** text field, type c₃₃, Finite Pert..

Global Evaluation 2

1 In the **Model Builder** window, expand the **Damping Coefficient, c33 (Finite Perturbation)** node, then click **Global Evaluation 2**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 In the **Perturbation Parameters** list box, select **Velocity Perturbation (Z)**.

4 In the **Damping Coefficient, c33 (Finite Perturbation)** toolbar, click **=** **Evaluate**.

Damping Coefficient, c22 (Finite Perturbation), Damping Coefficient, c23 (Finite Perturbation), Damping Coefficient, c32 (Finite Perturbation), Damping Coefficient, c33 (Finite Perturbation), Stiffness Coefficient, k22 (Finite Perturbation), Stiffness Coefficient, k23 (Finite Perturbation), Stiffness Coefficient, k32 (Finite Perturbation), Stiffness Coefficient, k33 (Finite Perturbation)

1. In the **Model Builder** window, under **Results**, Ctrl-click to select **Stiffness Coefficient, k22 (Finite Perturbation)**, **Stiffness Coefficient, k23 (Finite Perturbation)**, **Stiffness Coefficient, k32 (Finite Perturbation)**, **Stiffness Coefficient, k33 (Finite Perturbation)**, **Damping Coefficient, c22 (Finite Perturbation)**, **Damping Coefficient, c23 (Finite Perturbation)**, **Damping Coefficient, c32 (Finite Perturbation)**, and **Damping Coefficient, c33 (Finite Perturbation)**.
2. Right-click and choose **Group**.

Evaluation Groups (Dynamic Coefficients)

1. In the **Settings** window for **Group**, type `Evaluation Groups (Dynamic Coefficients)` in the **Label** text field.
2. In the **Model Builder** window, collapse the **Evaluation Groups (Dynamic Coefficients)** node.

Damping Coefficients

To compare the damping coefficients obtained with the built-in functionality and finite perturbation, add the damping coefficients to the existing plot.

1. In the **Model Builder** window, click **Damping Coefficients**.
2. In the **Settings** window for **1D Plot Group**, locate the **Legend** section.
3. In the **Number of columns** text field, type `2`.

c22 (Finite Perturbation)

1. Right-click **Damping Coefficients** and choose **Table Graph**.
2. In the **Settings** window for **Table Graph**, type `c22 (Finite Perturbation)` in the **Label** text field.
3. Locate the **Data** section. From the **Source** list, choose **Evaluation group**.
4. From the **Evaluation group** list, choose **Damping Coefficient, c22 (Finite Perturbation)**.
5. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.
6. From the **Color** list, choose **Cycle (reset)**.
7. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

8 Locate the **Legends** section. Select the **Show legends** checkbox.

c23 (Finite Perturbation)

1 Right-click **c22 (Finite Perturbation)** and choose **Duplicate**.

2 In the **Settings** window for **Table Graph**, type `c23 (Finite Perturbation)` in the **Label** text field.

3 Locate the **Data** section. From the **Evaluation group** list, choose **Damping Coefficient, c23 (Finite Perturbation)**.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle**.

c32 (Finite Perturbation)

1 Right-click **c23 (Finite Perturbation)** and choose **Duplicate**.

2 In the **Settings** window for **Table Graph**, type `c32 (Finite Perturbation)` in the **Label** text field.

3 Locate the **Data** section. From the **Evaluation group** list, choose **Damping Coefficient, c32 (Finite Perturbation)**.

c33 (Finite Perturbation)

1 Right-click **c32 (Finite Perturbation)** and choose **Duplicate**.

2 In the **Settings** window for **Table Graph**, type `c33 (Finite Perturbation)` in the **Label** text field.

3 Locate the **Data** section. From the **Evaluation group** list, choose **Damping Coefficient, c33 (Finite Perturbation)**.

4 In the **Damping Coefficients** toolbar, click **Plot**.

Join: Perturbation Pressure (Displacement Z)

The built-in functionality for determining dynamic coefficients automatically determines the perturbations to the fluid pressure due to infinitesimal displacement and velocity perturbations. Create a plot that compares the perturbation pressure due to a displacement perturbation in the z direction.

1 In the **Results** toolbar, click **More Datasets** and choose **Join**.

2 In the **Settings** window for **Join**, type `Join: Perturbation Pressure (Displacement Z)` in the **Label** text field.

3 Locate the **Data 1** section. From the **Data** list, choose **Study: Finite Perturbation/ Parametric Solutions 1 (sol3)**.

4 From the **Solutions** list, choose **One**.

5 From the **Parameter value** list, choose **Displacement Perturbation (Z)**.

6 Locate the **Data 2** section. From the **Data** list, choose **Study: Finite Perturbation/ Parametric Solutions 1 (sol3)**.

7 From the **Solutions** list, choose **One**.

8 Locate the **Combination** section. From the **Method** list, choose **General**.

9 In the **Expression** text field, type `(data1-data2)/u_pert`.

Perturbation Pressure

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Perturbation Pressure` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

4 Click to expand the **Plot Array** section. From the **Array type** list, choose **Linear**.

5 From the **Array axis** list, choose **y**.

6 In the **Relative padding** text field, type `0.6`.

7 Locate the **Color Legend** section. Select the **Show units** checkbox.

Surface 1

Right-click **Perturbation Pressure** and choose **Surface**.

Perturbation Pressure

In the **Model Builder** window, right-click **Surface 1** and choose **Contour**.

Surface: Built-in

1 In the **Settings** window for **Surface**, type `Surface: Built-in` in the **Label** text field.

2 Locate the **Expression** section. In the **Expression** text field, type `hdb.puz`.

Contour: Built-in

1 In the **Model Builder** window, under **Results** > **Perturbation Pressure** click **Contour 1**.

2 In the **Settings** window for **Contour**, type `Contour: Built-in` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `hdb.puz`.

4 Locate the **Coloring and Style** section. Clear the **Color legend** checkbox.

5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface: Built-in**.

6 Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

Surface 2

In the **Model Builder** window, right-click **Perturbation Pressure** and choose **Surface**.

Perturbation Pressure

In the **Model Builder** window, right-click **Surface 2** and choose **Contour**.

Surface: Finite Perturbation

1. In the **Settings** window for **Surface**, type `Surface: Finite Perturbation` in the **Label** text field.
2. Locate the **Data** section. From the **Dataset** list, choose **Join: Perturbation Pressure (Displacement Z)**.
3. Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface: Built-in**.

Contour: Finite Perturbation

1. In the **Model Builder** window, under **Results** > **Perturbation Pressure** click **Contour 2**.
2. In the **Settings** window for **Contour**, type `Contour: Finite Perturbation` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Join: Perturbation Pressure (Displacement Z)**.
4. Locate the **Coloring and Style** section. Clear the **Color legend** checkbox.
5. Locate the **Inherit Style** section. From the **Plot** list, choose **Surface: Built-in**.
6. Locate the **Plot Array** section. Select the **Manual indexing** checkbox.
7. In the **Index** text field, type `1`.

Annotation 1

1. In the **Model Builder** window, right-click **Perturbation Pressure** and choose **Annotation**.
2. In the **Settings** window for **Annotation**, locate the **Annotation** section.
3. In the **Text** text field, type `Built-in`.
4. Locate the **Position** section. In the **X** text field, type `0.5*W`.
5. In the **Z** text field, type `1.5*R`.
6. Locate the **Coloring and Style** section. Clear the **Show point** checkbox.
7. From the **Anchor point** list, choose **Center**.
8. Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

Annotation 2

1. Right-click **Annotation 1** and choose **Duplicate**.
2. In the **Settings** window for **Annotation**, locate the **Annotation** section.
3. In the **Text** text field, type `Finite Perturbation`.
4. Locate the **Plot Array** section. In the **Index** text field, type `1`.

5 In the **Perturbation Pressure** toolbar, click **Plot**.