Created in COMSOL Multiphysics 6.2

Nonlinear Magnetostrictive Transducer

Introduction

Magnetostriction describes the change in dimensions of a material due to a change in its magnetization. This phenomenon is a manifestation of magnetoelastic coupling, which is exhibited by all magnetic materials to some extent. The effects related to magnetoelastic coupling are described by various names. The *Joule effect* describes the change in length due to a change in the magnetization state of the material. This *magnetostrictive effect* is used in transducers for applications in sonars, acoustic devices, active vibration control, position control, and fuel injection systems.

Magnetostriction has a quantum-mechanical origin. The magneto-mechanical coupling takes place at the atomic level due to spin-orbit coupling. From a system level, the material can be assumed to consist of a number of tiny ellipsoidal magnets which rotate due to the torque produced by the externally applied magnetic field. The rotation of these elemental magnets produces a dimensional change leading to free strain in the material. The strain (or magnetostriction) has a nonlinear dependence on the magnetic field and the mechanical stress in the material.

This tutorial demonstrates how to model the nonlinear response of a magnetostrictive material.

Model Definition

A typical magnetostrictive transducer shown in Figure 1 has a steel housing enclosing a drive coil. A magnetostrictive material is placed in the core that works as an actuator when a magnetic field is applied by passing a current through the drive coil.

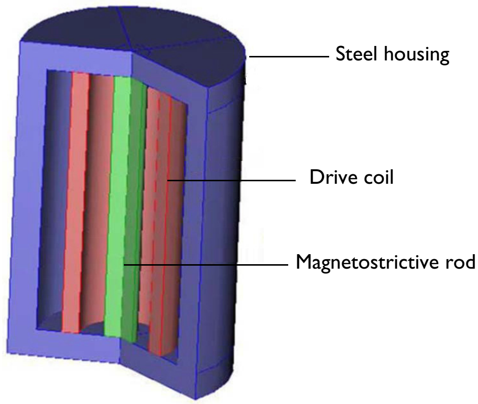


Figure 1: Sectional view of a cylindrical transducer.

Due to the rotational symmetry of the geometry, the problem is solved as a 2D axisymmetric model, which leads to reduced computation time. The corresponding 2D axisymmetric geometry is shown in Figure 2.

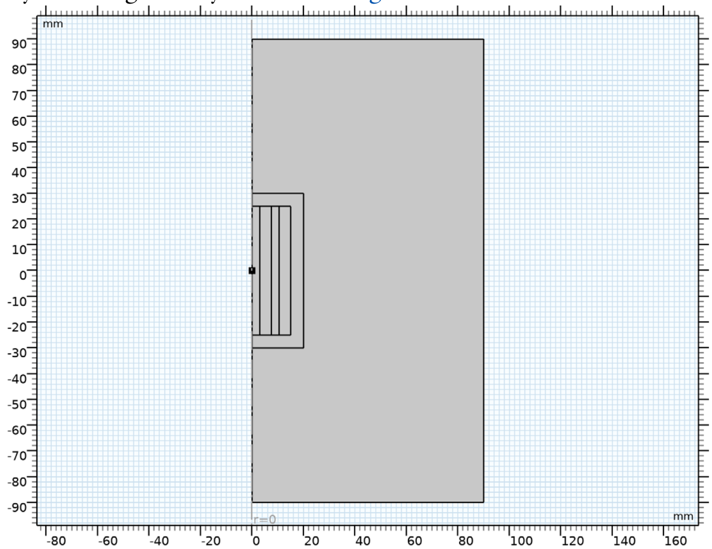


Figure 2: 2D axisymmetric view of a magnetostrictive transducer surrounded by an air domain. The geometric dimensions are in millimeters.

It is assumed that the current in the coil is DC, and hence it can be solved as a stationary problem. The first study performed considers a constant current density of 10^6 A/m^2 in the coil. A second study is set up where the current density in the drive coil is varied from 0 to 10^7 A/m^2 using the parametric sweep feature in COMSOL. The solution from this parametric sweep is then used to generate the characteristic nonlinear magnetostriction (λ) vs. magnetic field ($\boldsymbol{H}$) curve. The ramping of current density using the parametric sweep option is performed under the assumption that the current in the coil changes quasi-statically without producing any inductive effect.

NOTES ON THE MAGNETIC AND MAGNETOELASTIC PROBLEMS

An air domain is created around the transducer to realistically model the magnetic flux path. The boundaries of this air domain are magnetically insulated which ensures that flux does not diverge out of the modeling domain. An alternative technique of implementing this air domain in COMSOL Multiphysics involves the use of infinite elements. For more information on infinite elements, please refer to the *AC/DC Module User's Guide*.

The drive coil is modeled as a homogenized current-carrying domain. Individual wires and their electrical conductivity are not resolved. It is assumed that the externally applied current density in the coil is known *a priori*. In a 2D axisymmetric model, the external current density is the total current through the coil divided by the longitudinal cross-section area (coil length times coil thickness). The coil can also be modeled alternately using the Multiturn Coil Domain feature available in the AC/DC Module. Please refer to the *AC/DC Module User's Guide* for more details on using this alternative technique.

Traditionally, the magnetic flux density (also called the **B**-field) is obtained as a function of the applied magnetic field (the **H**-field). Such relationship is usually called a B-H curve. The steel housing used in this example is designed to create a closed magnetic flux path, thereby minimizing flux leakage. The nonlinear magnetic behavior of the steel housing is modeled by using a B-H curve to specify the magnetic constitutive relation in the material. The nonlinear B-H curve is obtained by choosing the material Soft Iron (Without Losses) from the AC/DC material library. Incorporation of a nonlinear B-H curve helps in modeling magnetic saturation effects at a sufficiently high magnetic field. Furthermore, you can examine the results of the model to find out specific locations in a material where magnetic saturation has taken place whereas other regions of that material have remained unsaturated.

The stress in the magnetostrictive material is modeled as

$$S = \mathrm{C_H}:(\varepsilon - \varepsilon_{\mathrm{me}}(\mathbf{M}))$$

The material is assumed to be isotropic, so that the elasticity tensor $\mathrm{C_H}$ can be represented in terms of two parameters, Young's modulus and Poisson's ratio.

The magnetostrictive strain is modeled as the following quadratic isotropic function of the magnetization field **M**:

$$\varepsilon_{\mathrm{me}} = \frac{3}{2}\frac{\lambda_s}{M_s^2}\mathrm{dev}(\mathbf{M} \otimes \mathbf{M})$$

where λ_s is the saturation magnetostriction, which is the maximum magnetostrictive strain reached at the saturation magnetization M_s. The tensor product of two vectors is defined as

$$(\mathbf{M} \otimes \mathbf{M})_{ij} = M_i M_j$$

Note that the magnetostrictive strain is represented by a deviatoric tensor. This is because the deformation can be related to the magnetic domain rotation associated with the magnetization of the material; such process should not change the material volume.

Nonlinear magnetization in the magnetostrictive material is found from the nonlinear relation

$$\mathbf{M} = M_s L(|\mathbf{H}_{\text{eff}}|)\frac{\mathbf{H}_{\text{eff}}}{|\mathbf{H}_{\text{eff}}|}$$

where L is the Langevin function

$$L = \coth\left(\frac{3\chi_0|\mathbf{H}_{\text{eff}}|}{M_s}\right) - \frac{M_s}{3\chi_0|\mathbf{H}_{\text{eff}}|}$$

with χ_0 being the magnetic susceptibility in the initial linear region, and the effective magnetic field in the material is given by

$$\mathbf{H}_{\text{eff}} = \mathbf{H} + \frac{3\lambda_s}{\mu_0 M_s^2} S_{\text{dev}}\mathbf{M}$$

where $S_{\text{dev}} = \text{dev}(S_{\text{el}})$ is the deviatoric part of the elastic stress tensor. The second term in the above relation represents the mechanical stress contribution to the effective field, and thus to the material magnetization, which is called the *Villari effect*. Note that the term is also proportional to magnetization, which implies that some applied magnetic field is needed for the Villari effect to occur. Thus, pure mechanics load cannot produce any magnetization of the material.

In addition, the magnetization and magnetic field are related to each other and to the **B**-field by

$$\mathbf{B} = \mu_0(\mathbf{H} + \mathbf{M})$$

The effective tangential piezomagnetic coupling coefficients can be computed as

$$\mathrm{d} = \frac{\partial \varepsilon_{\text{me}}}{\partial \mathbf{H}} = 3\frac{\lambda_s}{M_s^2}\left(\mathbf{M} \otimes \chi - \frac{1}{3}((I \otimes I) : (\mathbf{M} \otimes \chi))\right)$$

where

$$\chi = \frac{\partial \mathbf{M}}{\partial \mathbf{H}}$$

is the tangential magnetic susceptibility. An important observation from the above formula is that the piezomagnetic coefficients should reach their maximum (or minimum) at certain strength of the applied bias field. This is because $\mathbf{M}$ is zero at zero applied field, while χ tends to zero at large applied field magnitudes because of saturation.

The piezomagnetic coupling tensor $\mathbf{d}$ is a third order tensor. Due to the symmetry, it can be conventionally represented by a 3-by-6 matrix d_{HT} with only a few nonzero components.

The material properties used to describe the magnetostrictive material are shown in Table 1.

TABLE 1: MATERIAL PROPERTIES OF THE MAGNETOSTRICTIVE MATERIAL.

Material property	**Value**	**Description**
E	$60 \cdot 10^9$ Pa	Young's modulus
ν	0.45	Poisson's ratio
ρ	7870 kg/m^3	Density
σ	$5.96 \cdot 10^6$ S/m	Electric conductivity
ε_{r}	1	Relative permittivity
λ_{s}	$2 \cdot 10^{-4}$	Saturation magnetostriction
M_{s}	$1.5 \cdot 10^6$ A/m	Saturation magnetization

The lower end of the magnetostrictive rod is modeled as fixed, while the upper one can be mechanically loaded in order to study the Villari effect.

COUPLING THE MAGNETIC AND STRUCTURAL PROBLEMS

The implementation is straightforward as you make use of a predefined multiphysics coupling interface available in COMSOL called Nonlinear Magnetostriction.

Selecting this interface in the Model Wizard adds **Structural Mechanics** and **Magnetic Fields** interfaces together with the corresponding multiphysics coupling feature, **Nonlinear Magnetostriction**.

Most of the settings you need to configure the coupling are found in the **Settings** window for either the coupling feature or the **Ampère's Law, Nonlinear Magnetostrictive** feature added under the **Magnetic Fields** interface.

Results and Discussion

The results obtained from the first study, where a constant external current density of 10^6 A/m^2 is applied to the coil. Figure 3 shows the von Mises stress in the magnetostrictive material as a surface plot. This plot indicates that the stress due to magnetostriction is uniformly zero everywhere except the region near the bottom surface of the rod due to the fixed constraint boundary condition that was applied to this end of the rod. This is because the free strain due to magnetostriction should not produce any stress unless the material is mechanically constrained. Figure 4 shows that the corresponding strain field caused by the magnetostriction is also fairly uniform in the material except at the fixed end of the rod.

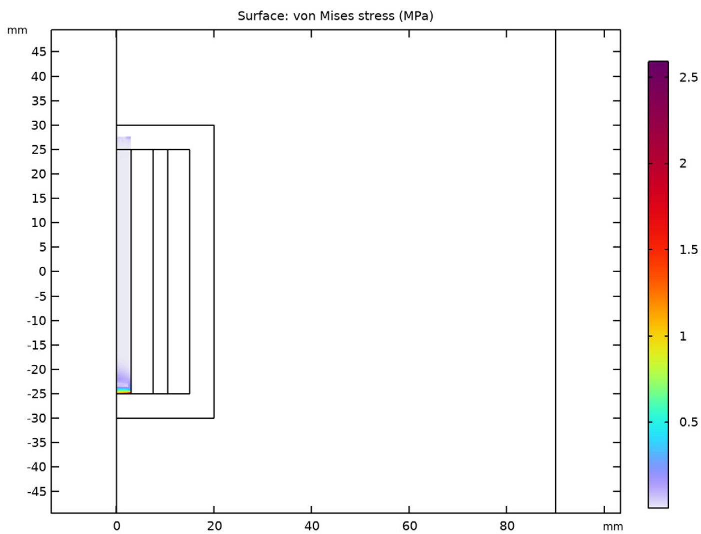


Figure 3: Surface plot of the von Mises stress and a scaled deformation plot of the displacement.

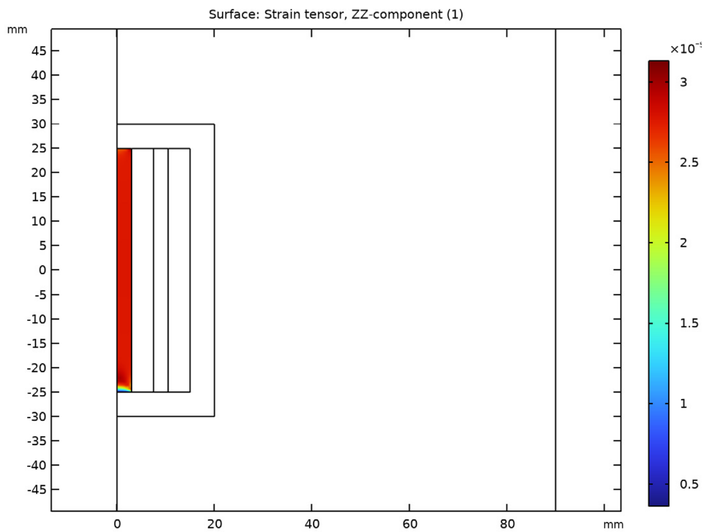


Figure 4: Surface plot of the axial strain component.

Figure 5 shows the magnetic flux concentration in the magnetostrictive core due to the closed magnetic path provided by the steel housing. The magnetic flux density in the rod is mostly uniform. Fringe effects can be seen at both ends of the rod where majority of the magnetic flux is forced to curl into the steel housing.

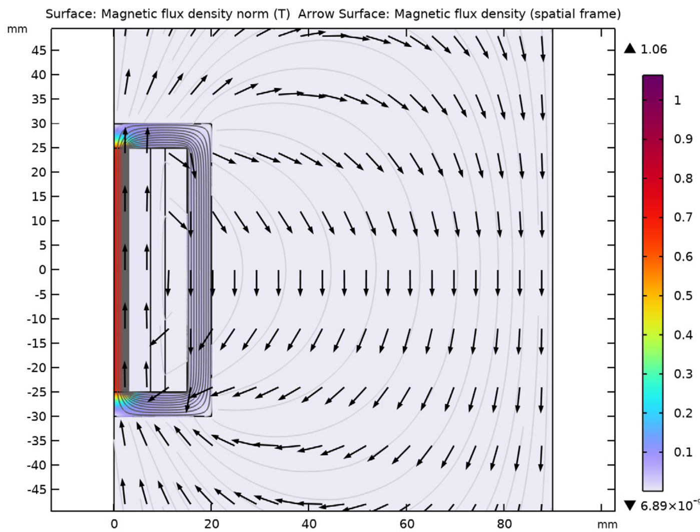


Figure 5: Surface plot of the norm of the magnetic flux density and a normalized arrow plot of its r and z-components showing the closed flux path in the model.

Figure 6 shows an interesting postprocessing feature in COMSOL Multiphysics. The solution obtained from the 2D axisymmetric model has been revolved by 225 degrees for 3D visualization of the solution. On solving a 2D axisymmetry model, COMSOL

Multiphysics automatically creates a 3D solution dataset by revolving the solution, which is then plotted as a 3D plot.

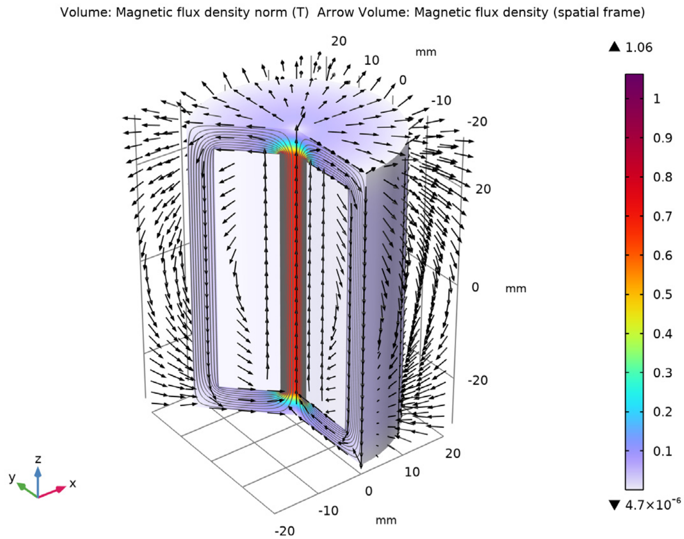


Figure 6: A 225 degree sectional view in 3D of the norm of the magnetic flux density in the magnetostrictive rod, steel housing and in the region within the housing. The solution in the outer air domain has been suppressed to get a better view. The normalized arrow plot shows the direction of the magnetic flux density.

Figure 7 shows the magnetostriction curve of the material obtained from the parametric study that simulated a quasi-static ramping up of the current density in the coil for three different values of the mechanical load. The corresponding B-H curve is shown in Figure 8. Because the magnetic field is oriented mostly along the axial direction, only the

Z-components of the corresponding vectors are plotted. Note the significantly nonlinear behavior in the region where the magnetic field H_z varies between 5 to 20 kA/m.

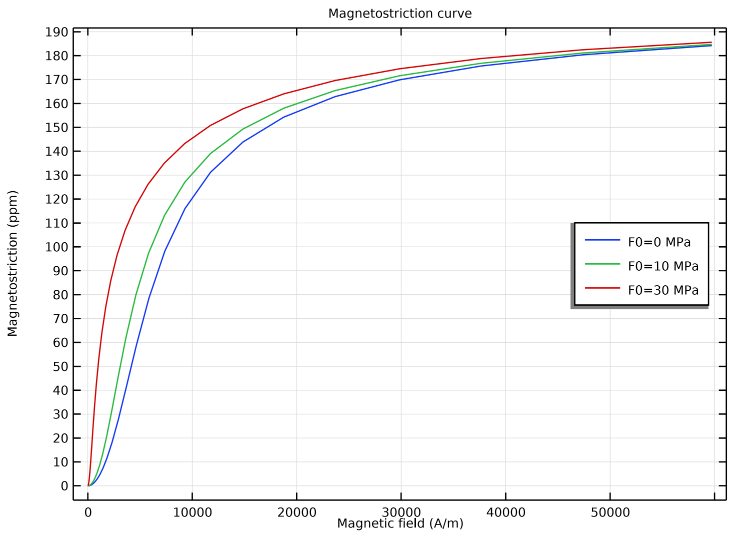


Figure 7: Magnetostriction versus magnetic field (at a point on the magnetostrictive material.

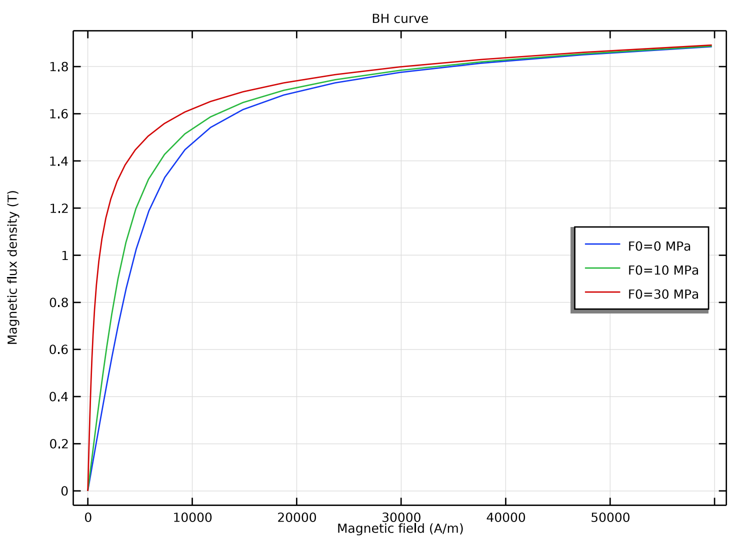


Figure 8: Magnetic flux density versus magnetic field at a point on the magnetostrictive material.

Finally, Figure 9 shows the components of the tangential piezomagnetic coupling matrix in case of no mechanical loading.

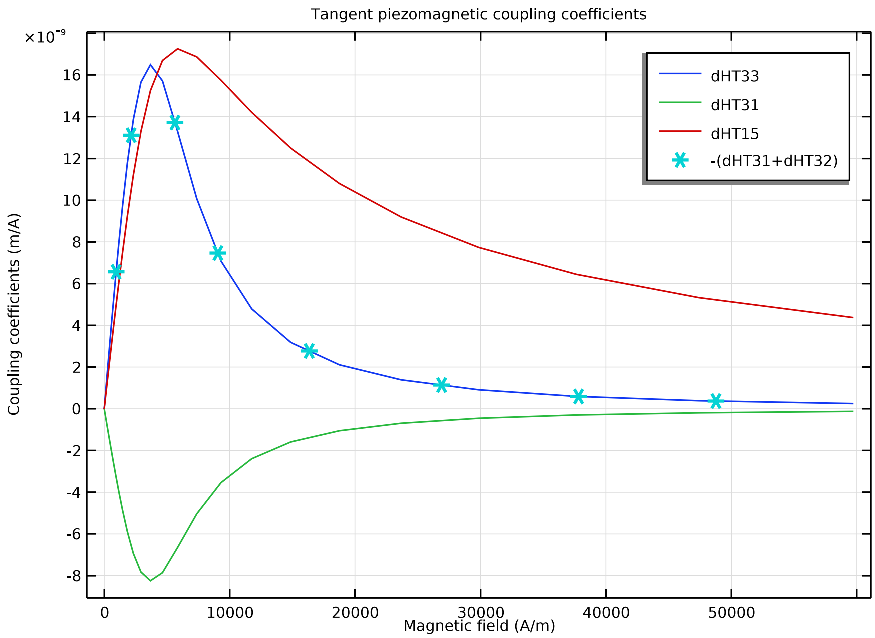


Figure 9: Tangential piezomagnetic coupling coefficients at a point on the magnetostrictive material.

Reference

1. S. Chikazumi, *Physics of Ferromagnetism*, Oxford University Press, New York, 1997.

Application Library path: `Structural_Mechanics_Module/Magnetomechanics/nonlinear_magnetostriction`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **2D Axisymmetric**.
2 In the **Select Physics** tree, select **Structural Mechanics>Electromagnetics–Structure Interaction>Magnetostriction>Nonlinear Magnetostriction**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Stationary**.
6 Click **Done**.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2 In the **Settings** window for **Geometry**, locate the **Units** section.
3 From the **Length unit** list, choose **mm**.

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `3`.
4 In the **Height** text field, type `50`.
5 Locate the **Position** section. In the **z** text field, type `-25`.
6 Click **Build Selected**.

Copy 1 (copy1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Copy**.
2 Select the object **r1** only.
3 In the **Settings** window for **Copy**, locate the **Displacement** section.
4 In the **r** text field, type `7.5`.
5 Click **Build Selected**.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `20`.
4 In the **Height** text field, type `5`.
5 Locate the **Position** section. In the **z** text field, type `-30`.

6 Click **Build Selected**.

Copy 2 (copy2)

1 In the **Geometry** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **r2** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **z** text field, type 55.

5 Click **Build Selected**.

Rectangle 3 (r3)

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type 5.

4 In the **Height** text field, type 50.

5 Locate the **Position** section. In the **r** text field, type 15.

6 In the **z** text field, type -25.

7 Click **Build Selected**.

Rectangle 4 (r4)

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type 90.

4 In the **Height** text field, type 180.

5 Locate the **Position** section. In the **z** text field, type -90.

6 Click **Build Selected**.

7 Click the **Zoom Extents** button in the **Graphics** toolbar.

Union 1 (uni1)

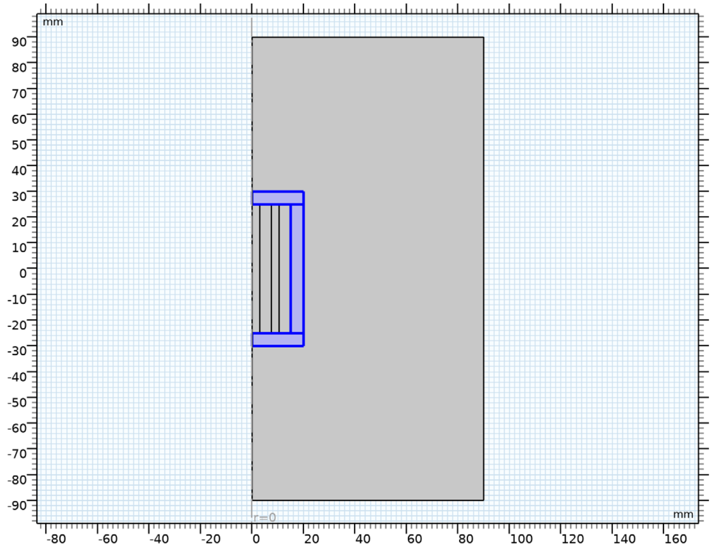
1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

See the figure below for the objects that need to be selected in the next step.

2 Select the objects **copy2**, **r2**, and **r3** only.

3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** check box.



Point 1 (pt1)

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

2 In the **Settings** window for **Point**, click **Build All Objects**.

3 Click the **Zoom Extents** button in the **Graphics** toolbar.

GLOBAL DEFINITIONS

Parameters 1

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

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
J0	1e6[A/m^2]	1E6 A/m²	Current density
F0	0[MPa]	0 Pa	Mechanical load

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 3 only.

The solid mechanics equations will be solved only in the magnetostrictive material.

MAGNETIC FIELDS (MF)

In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic Fields (mf)**.

Ampère's Law in Solids 1

1 In the **Physics** toolbar, click **Domains** and choose **Ampère's Law in Solids**.

2 Select Domain 2 only.

3 In the **Settings** window for **Ampère's Law in Solids**, locate the **Constitutive Relation B-H** section.

4 From the **Magnetization model** list, choose **B-H curve**.

Ampère's Law, Nonlinear Magnetostrictive 1

1 In the **Model Builder** window, click **Ampère's Law, Nonlinear Magnetostrictive 1**.

2 In the **Settings** window for **Ampère's Law, Nonlinear Magnetostrictive**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Manual**.

4 Click **Clear Selection**.

5 Select Domain 3 only.

External Current Density 1

1 In the **Physics** toolbar, click **Domains** and choose **External Current Density**.

2 Select Domain 5 only.

3 In the **Settings** window for **External Current Density**, locate the **External Current Density** section.

4 Specify the $\mathbf{J}_e$ vector as

0	r
J0	phi
0	z

ADD MATERIAL

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

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

3 In the tree, select **Built-in>Air**.

4 Right-click and choose **Add to Component 1 (comp1)**.

5 In the tree, select **AC/DC>Soft Iron (Without Losses)**.

6 Right-click and choose **Add to Component 1 (comp1)**.

7 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Air (mat1)

Select Domains 1 and 4–6 only.

Soft Iron (Without Losses) (mat2)

1 In the **Model Builder** window, click **Soft Iron (Without Losses) (mat2)**.

2 Select Domain 2 only.

Magnetostrictive

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

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

3 Select Domain 3 only.

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

Property	**Variable**	**Value**	**Unit**	**Property group**
Young's modulus	E	60e9	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.45	1	Young's modulus and Poisson's ratio
Density	rho	7870	kg/m³	Basic
Initial magnetic susceptibility	chi0_iso ; chi0ii = chi0_iso, chi0ij = 0	200	1	Magnetostrictive
Interdomain coupling	alphaJA_iso ; alphaJAii = alphaJA_iso, alphaJAij = 0	0	1	Jiles-Atherton model parameters
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	5.96e6	S/m	Basic

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic
Saturation magnetization	Ms	1.5e6	A/m	Magnetostrictive
Saturation magnetostriction	lambdas	200[ppm]	1	Magnetostrictive

SOLID MECHANICS (SOLID)

In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundary 6 only.

This boundary condition simulates that the lower surface of the magnetostrictive rod is fixed to the base of the transducer housing.

MESH 1

Free Quad 1

1 In the **Mesh** toolbar, click **Free Quad**.

2 In the **Settings** window for **Free Quad**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2 and 3 only.

Size 1

1 Right-click **Free Quad 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section.

5 Select the **Maximum element size** check box. In the associated text field, type `0.75`.

Free Triangular 1

1 In the **Mesh** toolbar, click **Free Triangular**.

2 In the **Settings** window for **Free Triangular**, click **Build All**.

STUDY 1

In the **Home** toolbar, click **Compute**.

DEFINITIONS

View 1

In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

Axis

1 In the **Model Builder** window, expand the **View 1** node, then click **Axis**.
2 In the **Settings** window for **Axis**, locate the **Axis** section.
3 In the **r maximum** text field, type 60.
4 In the **r minimum** text field, type -52.
5 In the **z minimum** text field, type -45.
6 In the **z maximum** text field, type 45.
7 Click **Update**.

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Surface 1**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 From the **Unit** list, choose **MPa**.
4 Locate the **Coloring and Style** section. From the **Color table transformation** list, choose **Nonlinear**.
5 Set the **Color calibration parameter** value to **-1**.

   The first default plot shows the von Mises stress in the magnetostrictive core along with a scaled deformation plot, which should be similar to that shown in Figure 3.

Surface 1

1 In the **Model Builder** window, expand the **Stress, 3D (solid)** node, then click **Surface 1**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 From the **Unit** list, choose **MPa**.
4 Locate the **Coloring and Style** section. From the **Color table transformation** list, choose **Nonlinear**.
5 Set the **Color calibration parameter** value to **-1**.

Follow the steps outlined below to create Figure 4.

Strain (solid)

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Strain (solid)` in the **Label** text field.

Surface 1

1 Right-click **Strain (solid)** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics> Strain>Strain tensor (material and geometry frames)>solid.eZZ - Strain tensor, ZZ-component**.

3 In the **Strain (solid)** toolbar, click **Plot**.

Compare the resulting plot with that in Figure 4.

Follow the steps outlined below to create Figure 5.

Arrow Surface 1

1 In the **Model Builder** window, right-click **Magnetic Flux Density Norm (mf)** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, locate the **Expression** section.

3 In the **r-component** text field, type `mf.Br`.

4 In the **z-component** text field, type `mf.Bz`.

5 Locate the **Arrow Positioning** section. Find the **r grid points** subsection. In the **Points** text field, type `20`.

6 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Normalized**.

7 From the **Color** list, choose **Black**.

8 In the **Magnetic Flux Density Norm (mf)** toolbar, click **Plot**.

Compare the resulting plot with that in Figure 5.

Follow the steps outlined below to create Figure 6.

Filter 1

1 In the **Model Builder** window, expand the **Magnetic Flux Density Norm, Revolved Geometry (mf)** node.

2 Right-click **Volume 1** and choose **Filter**.

3 In the **Settings** window for **Filter**, locate the **Element Selection** section.

4 In the **Logical expression for inclusion** text field, type `dom!=1`.

This excludes the outer air domain from the plot.

Magnetic Flux Density Norm, Revolved Geometry (mf)

1 In the **Model Builder** window, under **Results** click **Magnetic Flux Density Norm, Revolved Geometry (mf)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 Clear the **Plot dataset edges** check box.

Arrow Volume 1

1 Right-click **Magnetic Flux Density Norm, Revolved Geometry (mf)** and choose **Arrow Volume**.

2 In the **Settings** window for **Arrow Volume**, locate the **Expression** section.

3 In the **r-component** text field, type `mf.Br`.

4 In the **phi-component** text field, type `mf.Bphi`.

5 In the **z-component** text field, type `mf.Bz`.

6 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. From the **Entry method** list, choose **Coordinates**.

7 In the **Coordinates** text field, type `range(-20,4,20)`.

8 Find the **y grid points** subsection. From the **Entry method** list, choose **Coordinates**.

9 In the **Coordinates** text field, type `range(-20,4,20)`.

10 Find the **z grid points** subsection. From the **Entry method** list, choose **Coordinates**.

11 In the **Coordinates** text field, type `range(-30,2.5,30)`.

12 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Normalized**.

13 Select the **Scale factor** check box. In the associated text field, type `5`.

14 From the **Color** list, choose **Black**.

15 In the **Magnetic Flux Density Norm, Revolved Geometry (mf)** toolbar, click **Plot**.

SOLID MECHANICS (SOLID)

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundary 9 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

0	r
F0	z

Next, perform an auxiliary continuation sweep on the external current density for three different values of the mechanical load and plot the solution to view the saturation effect in the magnetostrictive core.

ADD STUDY

1 In the **Home** 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 **Add Study** in the window toolbar.

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

STUDY 2

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

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

3 Clear the **Generate default plots** check box.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

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

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 From the list in the **Parameter name** column, choose **J0 (Current density)**.

6 Click **Range**.

7 In the **Range** dialog box, type `0` in the **Start** text field.

8 In the **Step** text field, type `0.1`.

9 In the **Stop** text field, type `7.3`.

10 From the **Function to apply to all values** list, choose **exp10(x)** – **Exponential function (base 10)**.

11 Click **Add**.

Parametric Sweep

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

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
F0 (Mechanical load)	`0 10 30`	`MPa`

Running the entire parametric study based on the settings above will take a few minutes. The exact solution time will vary depending on the specification of the computer being used.

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

RESULTS

Follow the instructions below to create Figure 7 and Figure 8.

Magnetostriction

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/ Parametric Solutions 1 (sol3)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Magnetostriction curve`.

6 Locate the **Plot Settings** section.

7 Select the **x-axis label** check box. In the associated text field, type `Magnetic field (A/m)`.

8 Select the **y-axis label** check box. In the associated text field, type `Magnetostriction (ppm)`.

9 Locate the **Legend** section. From the **Position** list, choose **Middle right**.

Point Graph 1

1 Right-click **Magnetostriction** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type 4 in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multiphysics>Strain>Magnetostrictive strain tensor>npzm1.emeZZ - Magnetostrictive strain tensor, ZZ-component**.

7 Locate the **y-Axis Data** section. From the **Unit** list, choose **ppm**.

8 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

9 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Magnetic Fields>Magnetic>Magnetic field (material and geometry frames) - A/m>mf.HZ - Magnetic field, Z-component**.

10 Click to expand the **Legends** section. Select the **Show legends** check box.

11 Find the **Include** subsection. Clear the **Point** check box.

12 In the **Magnetostriction** toolbar, click **Plot**.

Magnetostriction

In the **Model Builder** window, right-click **Magnetostriction** and choose **Duplicate**.

BH Curve

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

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

3 Locate the **Title** section. In the **Title** text area, type `BH curve`.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Magnetic flux density (T)`.

Point Graph 1

1 In the **Model Builder** window, expand the **BH Curve** node, then click **Point Graph 1**.

2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Magnetic Fields>Magnetic>Magnetic flux density (material and geometry frames) - T>mf.BZ - Magnetic flux density, Z-component**.

3 In the **BH Curve** toolbar, click **Plot**.

Piezomagnetic Coefficients

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/ Parametric Solutions 1 (sol3)**.

4 From the **Parameter selection (F0)** list, choose **First**.

5 Locate the **Title** section. From the **Title type** list, choose **Manual**.

6 In the **Title** text area, type `Tangent piezomagnetic coupling coefficients`.

7 Locate the **Plot Settings** section.

8 Select the **x-axis label** check box. In the associated text field, type `Magnetic field (A/m)`.

9 Select the **y-axis label** check box. In the associated text field, type `Coupling coefficients (m/A)`.

Point Graph 1

1 Right-click **Piezomagnetic Coefficients** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Selection** section.

3 Click  **Paste Selection**.

4 In the **Paste Selection** dialog box, type `4` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multiphysics>Effective properties>Tangent piezomagnetic coupling matrix, Voigt notation - m/A>npzm1.dHT33 - Tangent piezomagnetic coupling matrix, Voigt notation, 33-component**.

7 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

8 In the **Expression** text field, type `mf.HZ`.

9 Click to expand the **Legends** section. Select the **Show legends** check box.

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
`dHT33`

12 Right-click **Point Graph 1** and choose **Duplicate**.

Point Graph 2

1 In the **Model Builder** window, click **Point Graph 2**.

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

3 In the **Expression** text field, type `npzm1.dHT31`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
dHT31

5 Right-click **Point Graph 2** and choose **Duplicate**.

Point Graph 3

1 In the **Model Builder** window, click **Point Graph 3**.

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

3 In the **Expression** text field, type `npzm1.dHT15`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
dHT15

5 Right-click **Point Graph 3** and choose **Duplicate**.

Point Graph 4

1 In the **Model Builder** window, click **Point Graph 4**.

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

3 In the **Expression** text field, type `-(npzm1.dHT31+npzm1.dHT32)`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
-(dHT31+dHT32)

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

6 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

7 From the **Positioning** list, choose **Interpolated**.

8 In the **Piezomagnetic Coefficients** toolbar, click **Plot**.