Created in COMSOL Multiphysics 6.2

Force Calculation 3 — Magnetic Torque BEM FEM

Introduction

A common way to determine electromagnetic forces on a (rigid) body, is to integrate the stresses originating from jumps in the electromagnetic field on its exterior surface. These stresses can be expressed by means of the Maxwell surface stress tensor[1]. In case of magnetostatics, the surface stress tensor is based directly on the magnetic flux density $\mathbf{B}$ and the magnetic field $\mathbf{H}$ on the boundary. Therefore, an accurate force computation requires accurate knowledge of the boundary fluxes.

Within the context of the magnetic scalar potential formulation[2], it makes sense to do a comparison between the *boundary element method* (BEM) and the *finite element method* (FEM). As opposed to the finite element method, for the boundary element method the flux normal to the boundary enters the equations directly as a degree of freedom. This allows for accurate flux computations without the need for reaction force integrals or weak constraints[3]. It is a potential advantage when doing force calculations.

In order to investigate the accuracy and general behavior of both methods, conditions are chosen for which the analytical solution is known. Results are analyzed. Mesh convergence is investigated by checking the behavior for different mesh sizes.

Model Definition

This model is a continuation of the *Magnetic Force BEM FEM* verification model. A single magnetized rod of one meter length, is placed in a perpendicular external field $\mathbf{B}_\mathrm{e}$ (see Figure 1). The relative permeability is assumed to be one everywhere. The strength of the external field is chosen such that the analytical model predicts a torque on the rod, of one newton-meter exactly.

1. For more information about the Maxwell surface stress tensor, see the chapter on *Electromagnetic Forces* in the *AC/DC Module User's Guide*.

2. This is the formulation used by the *Magnetic Fields, No Currents* and the *Magnetic Fields, No Currents, Boundary Elements* interfaces. For more information on this formulation, see the *AC/DC Module User's Guide*.

3. For more information on flux computation methods, see the chapter *Computing Accurate Fluxes* in the *COMSOL Multiphysics Reference Manual*.

THE ANALYTICAL MODEL

In the *Magnetic Force BEM FEM* verification model (the previous tutorial in this series), the analogy between an electrostatics problem and the magnetic scalar potential formulation is used to establish an analytical model that describes the force between two magnetized rods. *If you are new to this concept, consider starting with that tutorial first.* In this tutorial, for the torque, a similar approach is chosen.

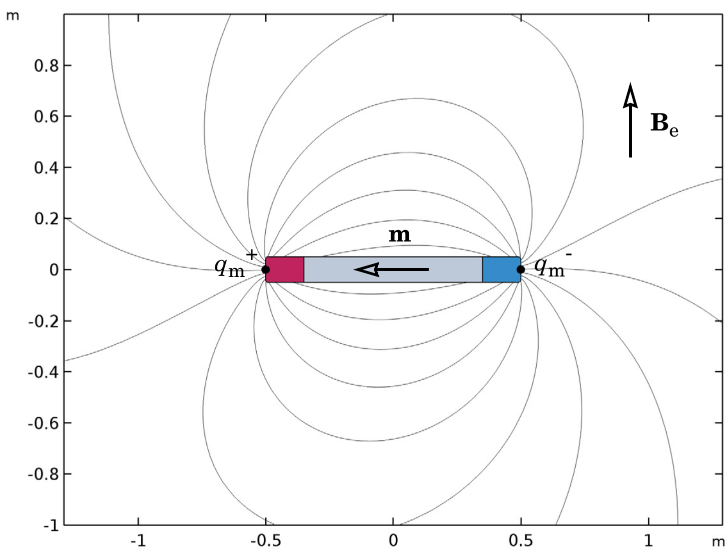


Figure 1: A single magnetized rod placed in a perpendicular external field. The situation is described analytically by means of two magnetic point charges forming a magnetic dipole.

A magnetized rod of one meter length, is described by means of two magnetic point charges q_{m1} and q_{m2} (in webers), at a distance of one meter apart (see Figure 1). The charge accumulated at the poles is the same as the one used for the two parallel rods discussed in the *Magnetic Force BEM FEM* model:

$$q_{m1} = q_{m2} = q_m = \sqrt{\frac{4\pi\mu_0}{2 - 1/\sqrt{2}}}\,. \tag{1}$$

Again, in order to make the point charge approximation accurate the rod is given a thickness that is small compared to the other distances involved, making this rod identical to either one of the two rods discussed previously.

The two point charges form a magnetic dipole, having a magnetic dipole moment **m** (in ampere-meter^2) equal to

$$\mathbf{m} = p_{\mathrm{m}}\mathbf{l} = \frac{q_{\mathrm{m}}}{\mu_0}\mathbf{l}\,. \tag{2}$$

Here, **l** is a vector pointing from q_{m}^- to q_{m}^+, with a magnitude equal to the distance between the two charges (one meter in this case). The division by μ_0 is essentially a unit conversion[4] from the magnetic charge q_{m} in webers, to the magnetic pole strength p_{m} in ampere-meters. Now, for a given external field $\mathbf{B}_{\mathrm{e}}$, the torque acting on this magnetic dipole is given by

$$\tau = \mathbf{m} \times \mathbf{B}_{\mathrm{e}}\,. \tag{3}$$

In a similar fashion, assuming the external field is oriented perpendicular with respect to the rod (that is, the angle θ is equal to 90°), we have:

$$|\tau| = |\mathbf{m}||\mathbf{B}_{\mathrm{e}}|\sin\theta = \frac{q_{\mathrm{m}}}{\mu_0}|\mathbf{B}_{\mathrm{e}}|\,. \tag{4}$$

Finally, given that **m** points in the negative x direction, the external field needed in order to achieve one newton-meter of torque pointing in the positive y direction, is given by

$$\mathbf{B}_{\mathrm{e}} = \begin{bmatrix} 0 \\ 0 \\ \mu_0/q_{\mathrm{m}} \end{bmatrix}. \tag{5}$$

This is all assuming the relative permeability equals one everywhere.

4. Notice that in SI units, there are two conflicting units in use for magnetic charge: webers and ampere-meters.

MODELING APPROACH

After setting up the general problem (geometry, physics, mesh, and such), two studies are performed. The first one focuses on the performance of the boundary element method alone. The second one repeats a similar investigation for a combined FEM-BEM model, where the finite element method is used to model the rod itself and its direct vicinity (marked as *solid* domains[5], see Figure 2).

Both studies use the same geometry. The geometry contains a single rod enclosed by a *force probe surface* (for accurate torque calculation, see On Auxiliary Surfaces). The rod and the probe are contained in a solid sphere and a cubic surface. The cube is used for applying the external field. It is not depicted here, as it is about five times the size of the sphere.

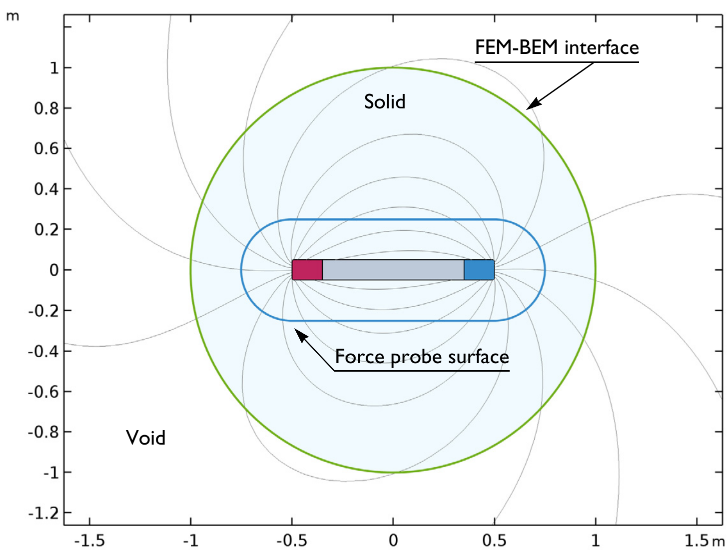


Figure 2: A 2D representation of the used geometry. The solid parts are solved for using both FEM and BEM. The void is solved using the BEM only.

The mesh for the two studies is different. The first study uses a boundary mesh only, for the exterior surface of the rod, the force probe, and the cube. For the second study a volumetric mesh is added to the solid domains, as this is required for solving with the finite element method. Both studies use a parametric sweep to refine the mesh near the poles (see On Mesh Convergence Studies).

---

5. Here, "solid" refers to a geometrical entity that can be equipped with a volumetric mesh. Objects of type "surface" on the other hand, can only be equipped by a boundary mesh. Their interior is referred to as a *void*.

The results are investigated on the poles and on the force probe surface. Both for the BEM model as well as the combined FEM-BEM model, the total torque is determined as a function of the mesh refinement parameter.

ON AUXILIARY SURFACES

The total torque on the rod is determined by integrating the Maxwell surface stress tensor over its exterior boundaries, taking the torque axis into account. As the fields near the poles will be highly concentrated, so will the stress tensor. The quantity entering the boundary integral will therefore be concentrated in a few mesh elements only, with a value close to zero everywhere else. These conditions result in poor numerical accuracy.

If there is enough space around the body of interest, one alternative is to introduce an auxiliary surface enclosing it while keeping a certain distance (in this model, referred to as a force probe surface). Since the singular behavior that occurs near the poles fades away at a distance, the stress tensor behaves more nicely here. The resulting integral is much more accurate.

ON MESH CONVERGENCE STUDIES

A mesh convergence study typically refines the mesh in some region that is considered to be important. During postprocessing, the behavior of the result as a function of mesh refinement is analyzed. The general assumption is that a well-behaved model (one that does not contain singularities and such) will approach the "true" solution, as the mesh is further refined. Consequently, for any particular mesh, the mesh convergence study gives an indication as to the margin of error that may be expected.

You could interpret the mesh as the *finite numerical resolution*[6] if you will, not to be confused with approximations done in the model itself. For example, the analytical model approximates the poles using point charges. This implies the rod is considered to be infinitely thin. As the actual rod used in the model has a finite thickness, the results will never perfectly correspond to the analytical model, even when using an infinitely fine mesh.

6. That is, taking the shape functions into account.

MESH CONVERGENCE STUDY BEM

During the first study, an inspection of the Maxwell surface stress tensor on the rods poles shows that the boundary element method is capable of producing smooth fields, even for relatively coarse meshes (see Figure 3).

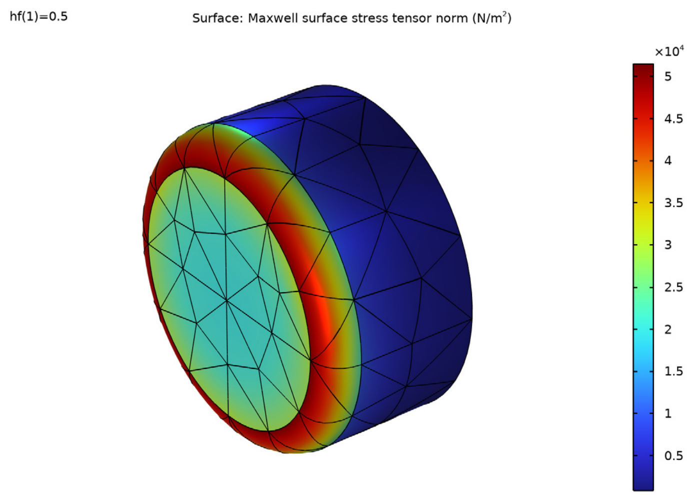


Figure 3: The Maxwell surface stress tensor at the rods pole, when using the boundary element method and a mesh scaling factor of one half.

Furthermore, the stresses are strongly concentrated on the fillets. This effect becomes more pronounced the smaller the radius of the fillet becomes. Without the fillet, the stress tensor would reach a singularity even. This is the reason why torque calculations on bodies with sharp edges are generally inaccurate[7].

7. Notice that this specifically applies to methods based on surface integration; volume integration may be a different matter.

On the force probe surface, the results are more smooth (that is, less erratic and distributed over a larger surface area). The scale reaches a value of about 9.4 N/m^2, as opposed to the value of ~$5 \cdot 10^4$ N/m^2 for the pole surface (see Figure 4).

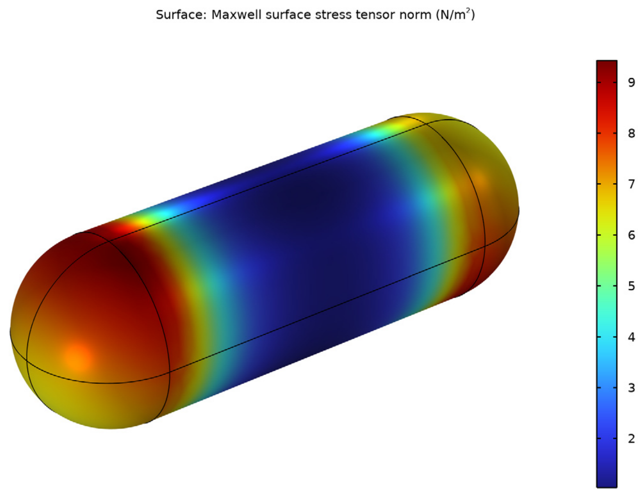


Figure 4: The Maxwell surface stress tensor on the force probe surface.

When using the BEM and integrating over the rods exterior, one finds for the calculated total torque, an error of about ~0.02% to ~1% (depending on the mesh scaling parameter; see Figure 5). The torque calculated using the force probe surface is more constant, and settles around ~0.2%. These are decent figures for a numerical model like this, certainly when considering the analytical model is not a hundred percent accurate either.

MESH CONVERGENCE STUDY FEM

During the second study, an inspection of the stress tensor on the rods poles shows singularities for low values of the mesh scaling factor *hf*. This is partly due to some of the mesh elements becoming linearized[8]. The linearized mesh elements are caused by the tetrahedra enclosing the curved surface (the tetrahedra are needed for solving with the FEM).

When comparing the total torque to the analytical model, an integration of the stress tensor over the rods surface gives an error that converges from somewhere in the interval 100–600% to about 0.3–7.0%, depending on *hf*. The force probe surface gives a more accurate value, at about 0.5–2.0%.

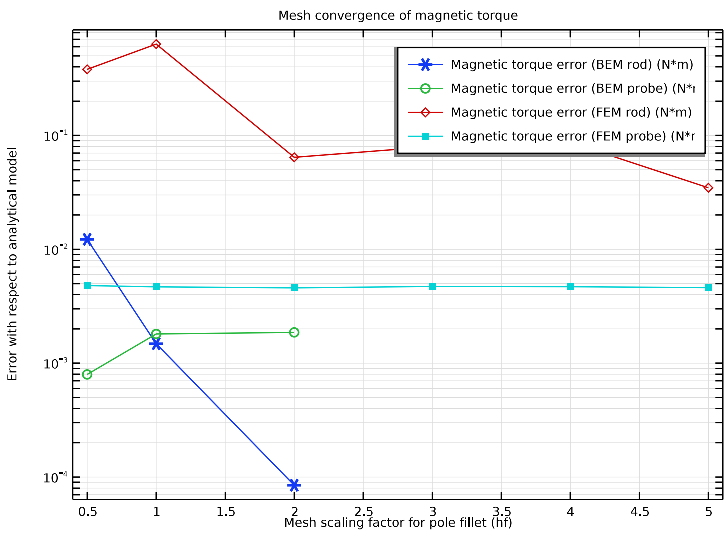


Figure 5: The error with respect to the analytical model, for both the rod surface integral and the probe surface integral, for both the BEM- and the combined FEM-BEM model.

Application Library path: `ACDC_Module/Introductory_Electromagnetic_Forces/force_calculation_03_magnetic_torque_bem`

---

8. If a shape shows strong curvature with respect to the mesh element size, a second-order mesh element may become inverted. That is, its boundaries may become self-overlapping. Usually, as a fallback, linear mesh elements are chosen. For more information on inverted mesh elements, see the *COMSOL Multiphysics Reference Manual.*

Modeling Instructions

This first section discusses how to set up the geometry, the selections, the physics, and the mesh. The actual studies are performed in sections *Modeling Instructions — Mesh Convergence Study BEM*, and *Modeling Instructions — Mesh Convergence Study FEM*.

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 **AC/DC>Magnetic Fields, No Currents>Magnetic Fields, No Currents, Boundary Elements (mfncbe)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **AC/DC>Magnetic Fields, No Currents>Magnetic Fields, No Currents (mfnc)**.
5 Click **Add**.
6 In the **Select Physics** tree, select **AC/DC>Magnetic Fields, No Currents>Magnetic Fields, No Currents, Boundary Elements (mfncbe)**.
7 Click **Add**.
8 Click **Study**.
9 In the **Select Study** tree, select **General Studies>Stationary**.
10 Click **Done**.

GEOMETRY 1

As the main focus will be on the physics and numerics, a detailed treatment of the geometry building procedure lies outside the scope of this tutorial. Instead, the geometry has been prepared in the file `force_calculation_01_introduction.mph`. You can start by inserting the geometry sequence from that file.

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.
2 Browse to the model's Application Libraries folder and double-click the file `force_calculation_01_introduction.mph`.
3 In the **Insert Sequence** dialog box, select **Geometry 2 (Magnetic Torque Verification)** in the **Select geometry sequence to insert** list.

4 Click **OK**.

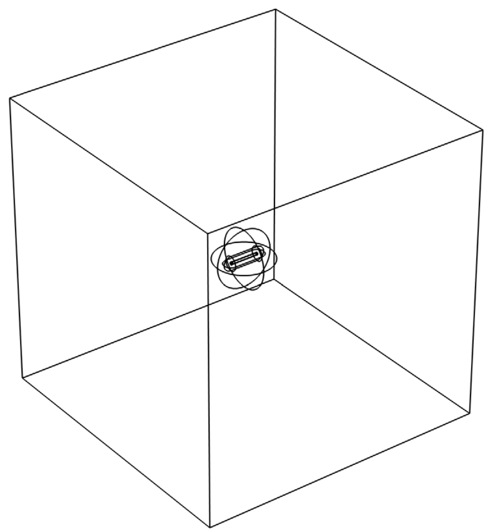
Form Union (fin)

1 In the **Geometry** toolbar, click **Build All**.

2 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

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

4 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.



You now have built the geometry. It contains a single rod in a spherical domain, encapsulated by a cubic surface. The sphere represents the boundary between the **Magnetic Fields, No Currents** interface, and the **Magnetic Fields, No Currents, Boundary Elements 2** interface (see Figure 2). The cube will be used to apply the external field.

The modeling instructions for this geometry can be found in the *Introduction* tutorial (that is, the first model in this force verification series), section *Modeling Instructions — Magnetic Torque Verification Geometry*.

If you have limited interest in manually building geometries in COMSOL Multiphysics (because you intend to use CAD software for example), feel free to proceed with setting the parameters and selections. If you are new to COMSOL however, it may be worthwhile to take some time and have a look at the instructions in the *Introduction* tutorial, as it will help you get familiar with the basics.

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.

   Notice that the parameter list has been populated already. These five parameters (`Rl`, `Ra`, `Rr`, `Rrf`, and `Cs`) have been imported automatically, together with the geometry sequence that is based on them.
3. Click **Load from File**.
4. Browse to the model's Application Libraries folder and double-click the file `force_calculation_c_mtorque_parameters.txt`.

   Nine more items have been added to the list. The first two of these are related to the mesh and the geometry. The last seven represent the analytical model used to set the correct remanent flux density and external field strength. The parameters `mu` and `Bez` follow from Equation 2 and Equation 5 respectively.

In the following part, some selections are made. These selections will be used later on, when assigning domain features or building the mesh for instance.

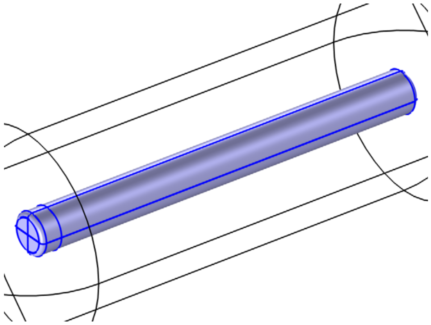
In order to make life easier, you can define the selections by typing or pasting their list of entity numbers (as seen in the instructions for **Explicit 3**). If you want to practice using the selection tools in the **Graphics** window, you should try to reproduce the images and check whether the list of selected entities turns up the same (*for details on selecting, panning and zooming, please consult the reference manual*).

DEFINITIONS

Magnetized Rod Domain

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Magnetized Rod Domain` in the **Label** text field.
3. Select Domains 3 and 4 only.

4 Click the **Zoom to Selection** button in the **Graphics** toolbar.



Magnetized Rod Surface

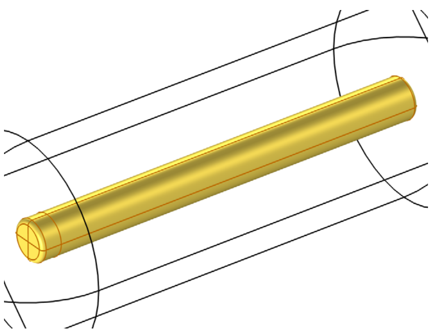
1 In the **Definitions** toolbar, click **Adjacent**.

2 In the **Settings** window for **Adjacent**, type `Magnetized Rod Surface` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog box, select **Magnetized Rod Domain** in the **Input selections** list.

5 Click **OK**.



Magnetized Rod Pole Surface

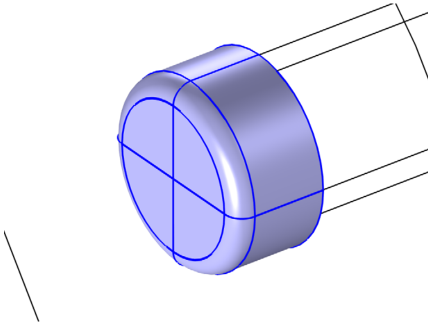
1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Magnetized Rod Pole Surface` in the **Label** text field.

3 Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.

4 Select Domain 3 only.

5 Click the **Zoom to Selection** button in the **Graphics** toolbar.



Magnetized Rod Pole Fillet

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Magnetized Rod Pole Fillet` in the **Label** text field.

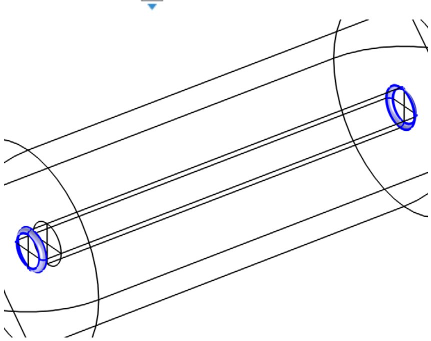
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `17, 19, 22, 24, 42-45` in the **Selection** text field (*that is, the boundaries comprising the pole fillets*).

6 Click **OK**.

7 Click the **Zoom to Selection** button in the **Graphics** toolbar.



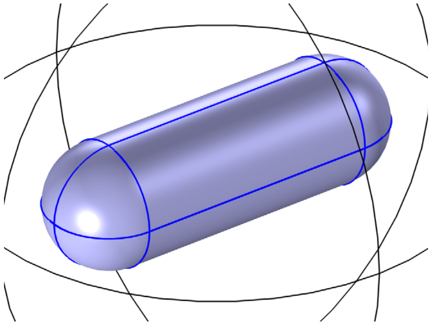
Force Probe Domain

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Force Probe Domain` in the **Label** text field.

3 Select Domains 2–4 only (*that is, both the magnetized rod, as well as the domain enclosing it*).

4 Click the **Zoom to Selection** button in the **Graphics** toolbar.



Force Probe Surface

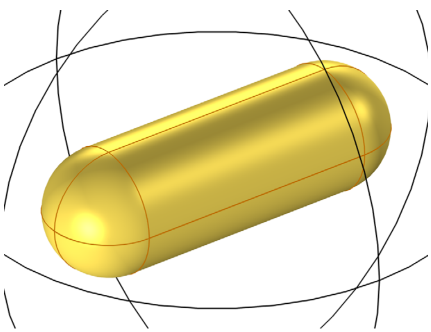
1 In the **Definitions** toolbar, click **Adjacent**.

2 In the **Settings** window for **Adjacent**, type `Force Probe Surface` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog box, select **Force Probe Domain** in the **Input selections** list.

5 Click **OK**.



External Field In

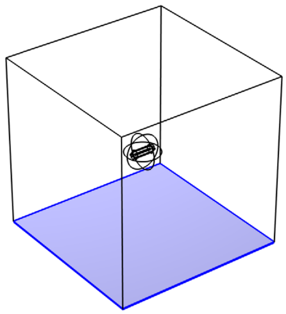
1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `External Field In` in the **Label** text field.

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

4 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundary 3 only.



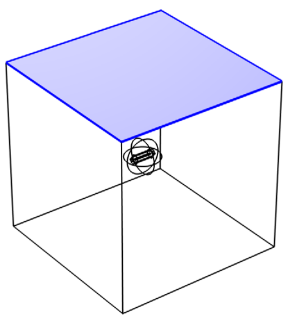
External Field Out

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `External Field Out` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 4 only.



Exterior Cube Surface

1 In the **Definitions** toolbar, click **Explicit**.

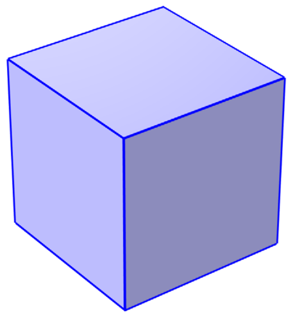
2 In the **Settings** window for **Explicit**, type `Exterior Cube Surface` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `1-5, 54` in the **Selection** text field (*that is, the boundaries comprising the cube*).

6 Click **OK**.



MATERIALS

Next, will be the materials. For this model we only have one material, with a relative permeability of one everywhere. Please make sure the material selection is set to **All domains and voids**. Here, a "void" represents a domain that will not be meshed. Still, the boundary element method is able to solve for it. This is one of the main advantages of the BEM, when compared to the finite element method (*for more information on this, please consult the reference manual*).

Material 1 (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **All domains and voids**.
4. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1

MAGNETIC FIELDS, NO CURRENTS, BOUNDARY ELEMENTS (MFNCBE)

Now that the materials have been set and double-checked, let us have a look at the physics. The first physics interface, **Magnetic Fields, No Currents, Boundary Elements**, is used for solving the entire model using the boundary element method only. The second and third interface will be coupled. **Magnetic Fields, No Currents** will solve a finite element problem in the rod itself and its direct vicinity (see Figure 2). **Magnetic Fields, No Currents, Boundary Elements 2** will solve a boundary element problem for the cubic void only.

Start with the first physics interface, by setting the correct selection and adjusting the far-field approximation tolerance.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic Fields, No Currents, Boundary Elements (mfncbe)**.
2 In the **Settings** window for **Magnetic Fields, No Currents, Boundary Elements**, locate the **Domain Selection** section.
3 Click **Clear Selection**.
4 Click **Paste Selection**.
5 In the **Paste Selection** dialog box, type `-1, 1-4` in the **Selection** text field (that is, everything except the infinite void). Notice that *finite voids* are indicated with a negative domain number. The *infinite void* is numbered `0`.
6 Click **OK**.
7 Click the **Show More Options** button in the **Model Builder** toolbar.
8 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.
9 Click **OK**.
10 In the **Settings** window for **Magnetic Fields, No Currents, Boundary Elements**, click to expand the **Far-Field Approximation** section.
11 In the **Relative tolerance** text field, type `1e-6`.

   Reducing the relative tolerance for the far-field approximation improves the accuracy of the BEM solution, without hindering performance too much (*for more information on the far-field approximation settings, please consult the reference manual or use context help*).

Next, add separate flux conservation features for the force probe domain and the magnetized rod. This will allow for determining the Maxwell surface stress tensor on their exterior boundaries.

Magnetic Flux Conservation 2

1 In the **Physics** toolbar, click **Domains** and choose **Magnetic Flux Conservation**.
2 In the **Settings** window for **Magnetic Flux Conservation**, locate the **Domain Selection** section.
3 From the **Selection** list, choose **Force Probe Domain**.
4 Click the **Zoom to Selection** button in the **Graphics** toolbar.

Magnetic Flux Conservation 3

1 In the **Physics** toolbar, click **Domains** and choose **Magnetic Flux Conservation**.
2 In the **Settings** window for **Magnetic Flux Conservation**, locate the **Domain Selection** section.
3 From the **Selection** list, choose **Magnetized Rod Domain**.
4 Locate the **Constitutive Relation B-H** section. From the **Magnetization model** list, choose **Remanent flux density**.
5 From the $\|\mathbf{B}_r\|$ list, choose **User defined**. In the associated text field, type `sqrt(Brx^2+Brz^2)`.
6 Specify the **e** vector as

`Brx/sqrt(Brx^2+Brz^2)`	x
`0`	y
`Brz/sqrt(Brx^2+Brz^2)`	z

Observe that the second magnetic flux conservation feature partly overrides the first one. Therefore, it is important to have them in this order: the probe domain first, the rod domain second. For the force calculation features the order is less important.

Force Calculation 1

1 In the **Physics** toolbar, click **Domains** and choose **Force Calculation**.
2 In the **Settings** window for **Force Calculation**, locate the **Domain Selection** section.
3 From the **Selection** list, choose **Magnetized Rod Domain**.
4 Locate the **Force Calculation** section. In the **Force name** text field, type `BEM_rod`.
5 Specify the $\mathbf{r}_{ax}$ vector as

0	x
1	y
0	z

Force Calculation 2

1 In the **Physics** toolbar, click **Domains** and choose **Force Calculation**.
2 In the **Settings** window for **Force Calculation**, locate the **Domain Selection** section.
3 From the **Selection** list, choose **Force Probe Domain**.
4 Locate the **Force Calculation** section. In the **Force name** text field, type `BEM_probe`.

5 Specify the $\mathbf{r}_{\mathrm{ax}}$ vector as

0	x
1	y
0	z

These force calculation features provide the machinery for automated torque calculation. The total torque on the chosen domain selection will be determined by integrating the stress tensor over its exterior boundaries, taking the torque axis into account.

Finally, apply the external field. In order to see the selections for the exterior boundary conditions, zoom out a bit.

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

Magnetic Flux Density 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Magnetic Flux Density**.

2 In the **Settings** window for **Magnetic Flux Density**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **External Field In**.

4 Locate the **Magnetic Flux Density** section. In the B_{n} text field, type `Bez`.

Zero Magnetic Scalar Potential 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Zero Magnetic Scalar Potential**.

2 In the **Settings** window for **Zero Magnetic Scalar Potential**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **External Field Out**.

Notice that the zero scalar potential boundary condition actually serves a double purpose. First of all, it is an *equipotential surface* forcing the field in a vertical direction near the boundary. Secondly, it provides a zero potential reference, making the scalar potential field unique. Without it, only its spatial derivative would be unique and solving with a direct solver would not work (*for details on under constrained problems, please consult the reference manual*).

The next two interfaces will essentially perform the same simulation as the first one. Observe that the **Magnetic Fields, No Currents** interface does not need a zero potential constraint (since it will be coupled).

What remains are the settings for the remanent flux density and the force calculation features. Proceed by adding them.

MAGNETIC FIELDS, NO CURRENTS (MFNC)

1 Click the **Zoom to Selection** button in the **Graphics** toolbar.

2 In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic Fields, No Currents (mfnc)**.

Magnetic Flux Conservation 2

1 In the **Physics** toolbar, click **Domains** and choose **Magnetic Flux Conservation**.

2 In the **Settings** window for **Magnetic Flux Conservation**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Magnetized Rod Domain**.

4 Locate the **Constitutive Relation B-H** section. From the **Magnetization model** list, choose **Remanent flux density**.

5 From the $\|\mathbf{B}_\mathrm{r}\|$ list, choose **User defined**. In the associated text field, type `sqrt(Brx^2+Brz^2)`.

6 Specify the **e** vector as

`Brx/sqrt(Brx^2+Brz^2)`	x
`0`	y
`Brz/sqrt(Brx^2+Brz^2)`	z

Notice that the finite element method does not need a separate flux conservation feature for the probe domain. This is because for the FEM, all boundaries within the physics domain selection will be equipped with the appropriate boundary variable definitions by default (the surface stress tensor in this case). The BEM on the other hand, only defines these variables for the boundaries that it solves for.

Force Calculation 1

1 In the **Physics** toolbar, click **Domains** and choose **Force Calculation**.

2 In the **Settings** window for **Force Calculation**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Magnetized Rod Domain**.

4 Locate the **Force Calculation** section. In the **Force name** text field, type `FEM_rod`.

5 Specify the $\mathbf{r}_\mathrm{ax}$ vector as

`0`	x
`1`	y
`0`	z

Force Calculation 2

1 In the **Physics** toolbar, click **Domains** and choose **Force Calculation**.

2 In the **Settings** window for **Force Calculation**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Force Probe Domain**.

4 Locate the **Force Calculation** section. In the **Force name** text field, type `FEM_probe`.

5 Specify the $\mathbf{r}_{\mathrm{ax}}$ vector as

0	x
1	y
0	z

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

MAGNETIC FIELDS, NO CURRENTS, BOUNDARY ELEMENTS 2 (MFNCBE2)

The last interface solves for the void between the sphere and the cube. It applies the same exterior boundary conditions as the **Magnetic Fields, No Currents, Boundary Elements** interface.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic Fields, No Currents, Boundary Elements 2 (mfncbe2)**.

2 In the **Settings** window for **Magnetic Fields, No Currents, Boundary Elements**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `-1` in the **Selection** text field (*that is, just the finite void*).

6 Click **OK**.

Magnetic Flux Density 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Magnetic Flux Density**.

2 In the **Settings** window for **Magnetic Flux Density**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **External Field In**.

4 Locate the **Magnetic Flux Density** section. In the B_{n} text field, type `Bez`.

Zero Magnetic Scalar Potential 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Zero Magnetic Scalar Potential**.

2 In the **Settings** window for **Zero Magnetic Scalar Potential**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **External Field Out**.

If this selection shows up as "not applicable", it probably means for the physics interface you have selected one void too much (it should only be "Finite void 1").

MULTIPHYSICS

Magnetic Scalar-Scalar Potential Coupling 1 (msspc1)

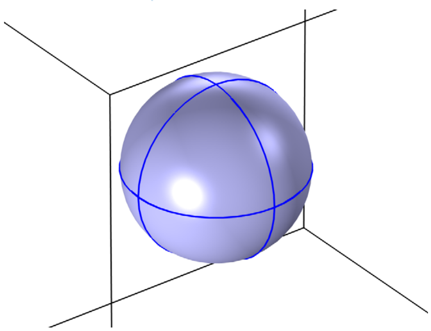
1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary> Magnetic Scalar-Scalar Potential Coupling**.

2 In the **Settings** window for **Magnetic Scalar-Scalar Potential Coupling**, locate the **Coupled Interfaces** section.

3 From the **Secondary interface (magnetic scalar potential)** list, choose **Magnetic Fields, No Currents, Boundary Elements 2 (mfncbe2)**.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **All boundaries**.

5 Click the **Zoom to Selection** button in the **Graphics** toolbar.



This coupling constrains the difference between the FEM scalar potential and the BEM scalar potential to zero on the boundary (*that is, it makes them equal*).

MESH 1

Next will be a manual mesh. Special care is taken to create a fine mesh for the fillets on the magnetized rod.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

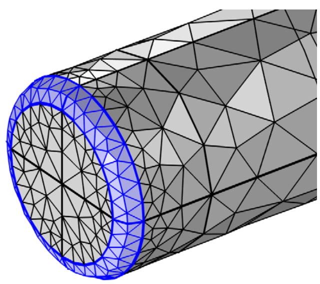
3 From the **Element size** list, choose **Finer**.

Free Triangular 1

1. In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.
2. In the **Settings** window for **Free Triangular**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Magnetized Rod Surface**.

Size 1

1. Right-click **Free Triangular 1** and choose **Size**.
2. In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Magnetized Rod Pole Fillet**.
4. Locate the **Element Size** section. Click the **Custom** button.
5. Locate the **Element Size Parameters** section.
6. Select the **Maximum element size** check box. In the associated text field, type `Rrf/hf`.
7. Click **Build All**.



Notice that the mesh uses the parameter `hf`. Setting this parameter from `1` to `2` and so on, refines the mesh on the fillet. This will be of particular interest during the *mesh convergence studies*.

Free Triangular 2

1. In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.
2. In the **Settings** window for **Free Triangular**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Force Probe Surface**.

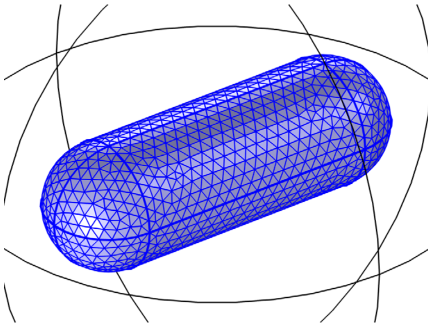
Size 1

1. Right-click **Free Triangular 2** 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 `Rr`.

6 Click **Build All**.

7 Click the **Zoom to Selection** button in the **Graphics** toolbar.



As the force probe surface mesh does not depend on the parameter `hf`, it will not vary during the mesh convergence study.

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 **Selection** list, choose **Exterior Cube Surface**.

Distribution 1

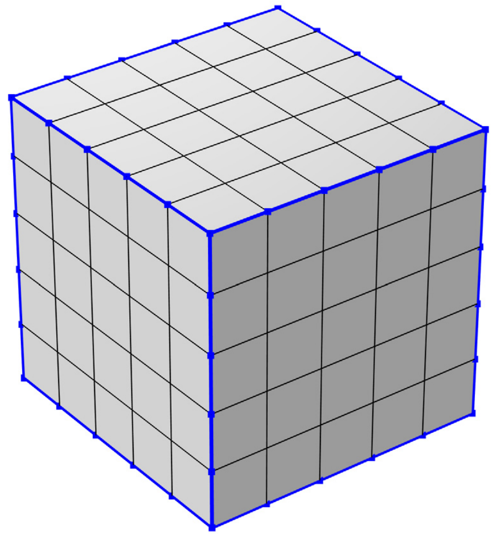
1 Right-click **Mapped 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Edge Selection** section.

3 From the **Selection** list, choose **All edges**.

4 Click **Build All**.

5 Click the **Zoom to Selection** button in the **Graphics** toolbar.



Modeling Instructions — Mesh Convergence Study BEM

STUDY 1

A mesh convergence study typically investigates the behavior of the result as a function of mesh refinement (see section On Mesh Convergence Studies). For any particular mesh, the mesh convergence study gives an indication as to the margin of error that may be expected.

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

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

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

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 (*make sure the parameter unit is cleared*):

Parameter name	Parameter value list	Parameter unit
hf (Mesh scaling factor for pole fillet)	0.5 1 2	

Step 1: Stationary

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

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

3 In the tables, enter the following settings:

Physics interface, Multiphysics couplings	Solve for
Magnetic Fields, No Currents, Boundary Elements (mfncbe)	Yes
Magnetic Fields, No Currents (mfnc)	No
Magnetic Fields, No Currents, Boundary Elements 2 (mfncbe2)	No
Magnetic Scalar-Scalar Potential Coupling 1 (msspc1)	No

Before hitting compute, *it is recommended to save your model.*

Solving should take about 5 minutes on an average desktop machine. The required RAM is somewhere around 6 GB. If the model is too demanding for your current setup, consider changing the parameter setting `hf` from `0.5 1 2`, to `0.5 1` (that is, solving for the coarser meshes only).

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

RESULTS

Since the main region of interest lies inside the sphere, it is convenient to add a selection to the solution (**All domains** in this case). This will force the plots to exclude the cube (which is a void).

In the **Model Builder** window, expand the **Results** node.

Selection

1 In the **Model Builder** window, expand the **Results>Datasets** node.

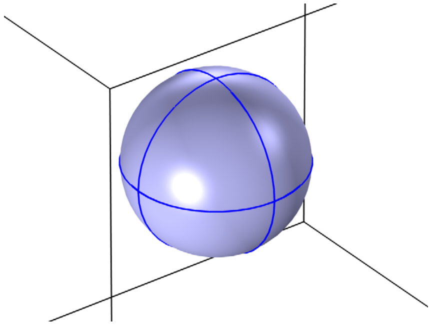
2 Right-click **Study 1/Solution 1 (sol1)** and choose **Selection**.

3 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

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

5 From the **Selection** list, choose **All domains**.

6 Click the **Zoom to Selection** button in the **Graphics** toolbar.



The first plot group considers the norm of the surface stress tensor on the poles. The stress tensor plot will be combined with a mesh plot, to show the relation between the mesh and the resulting fields. Start by adding a new plot group.

Maxwell Surface Stress Tensor (BEM Rod)

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Maxwell Surface Stress Tensor (BEM Rod)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

4 From the **Parameter value (hf)** list, choose **0.5**.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

6 Click to expand the **Selection** section. From the **Geometric entity level** list, choose **Boundary**.

7 From the **Selection** list, choose **Magnetized Rod Pole Surface**.

Surface 1

1 Right-click **Maxwell Surface Stress Tensor (BEM Rod)** and choose **Surface**.

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

3 In the **Expression** text field, type `sqrt(mfncbe.nToutx_BEM_rod^2+ mfncbe.nTouty_BEM_rod^2+mfncbe.nToutz_BEM_rod^2)`.

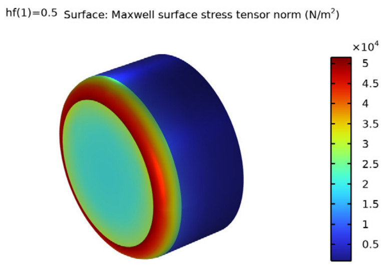
Notice that for longer expressions like this one, *the easiest way to go, is to copy-paste them directly from this *.pdf file to COMSOL.*

Variables like `mfncbe.nToutx_BEM_rod` have been generated by the force calculation features. You can find them using the buttons in the top-right corner of the **Expression** section, just above the text input field for the expression. There is some autocomplete functionality too (try pressing Ctrl+Space, when having focus in the text input field).

4 Select the **Description** check box. In the associated text field, type `Maxwell surface stress tensor norm`.

5 In the **Maxwell Surface Stress Tensor (BEM Rod)** toolbar, click **Plot**.

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



Surface 2

1 In the **Model Builder** window, right-click **Maxwell Surface Stress Tensor (BEM Rod)** and choose **Surface**.

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

3 In the **Expression** text field, type `0`.

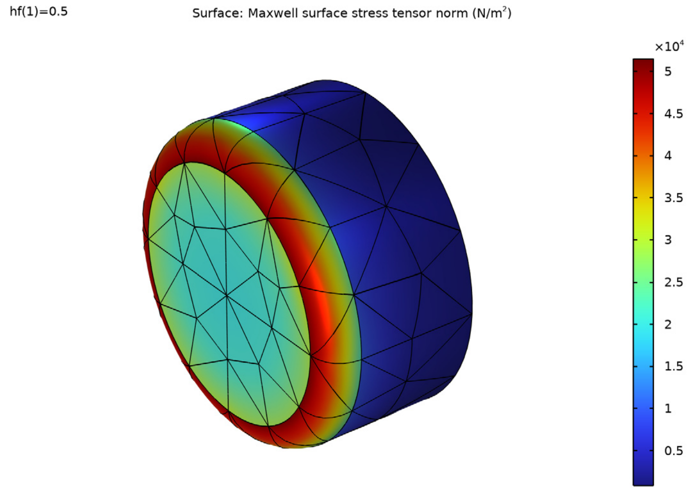
4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

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

7 Select the **Wireframe** check box.

8 In the **Maxwell Surface Stress Tensor (BEM Rod)** toolbar, click **Plot**.



One of the things to notice, is that the solution depends on the chosen mesh. Feel free to investigate the plot for different values of the mesh scaling factor `hf`.

Next, create a plot for the surface stress tensor on the force probe surface.

Maxwell Surface Stress Tensor (BEM Probe)

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Maxwell Surface Stress Tensor (BEM Probe)` in the **Label** text field.

Since the probe surface is much larger than the rod pole, it is convenient to have two separate views. This way, the first view can be kept zoomed in on the pole, while the second one focuses on the probe. Using a single view for both, would result in having to adjust the camera settings, every time when switching between the first and second plot group.

3 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

Surface 1

1 Right-click **Maxwell Surface Stress Tensor (BEM Probe)** and choose **Surface**.

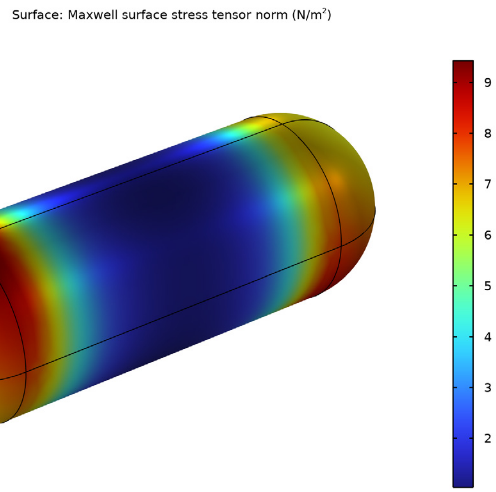
2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `sqrt(mfncbe.nToutx_BEM_probe^2+mfncbe.nTouty_BEM_probe^2+mfncbe.nToutz_BEM_probe^2)`.

4 Select the **Description** check box. In the associated text field, type `Maxwell surface stress tensor norm`.

5 In the **Maxwell Surface Stress Tensor (BEM Probe)** toolbar, click **Plot**.

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



The field is more smooth here. The scale reaches a value of about 9.4 N/m^2, as opposed to the value of ~$5 \cdot 10^4$ N/m^2 for the pole surface.

Finally, evaluate the error for the total torque as a function of `hf`. The error norm $||\delta||$ is expressed in terms of the norm of the difference between the calculated torque vector $\mathbf{T}$ and the analytical result τ, that is: $||\delta|| = ((T_x - \tau_x)^2 + (T_y - \tau_y)^2 + (T_z - \tau_z)^2)^{1/2}$.

In this case, for the analytical result we have $\tau = [0,1,0]^T$. Notice that since the analytical result has a magnitude of one, the error can be interpreted both as a relative (dimensionless) error, or as an absolute error (in Newton-meters).

Global Evaluation 1

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

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

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

Expression	Description
`sqrt(mfncbe.Tx_BEM_rod^2+(mfncbe.Ty_BEM_rod-1[N*m])^2+mfncbe.Tz_BEM_rod^2)`	`Magnetic torque error (BEM rod)`
`sqrt(mfncbe.Tx_BEM_probe^2+(mfncbe.Ty_BEM_probe-1[N*m])^2+mfncbe.Tz_BEM_probe^2)`	`Magnetic torque error (BEM probe)`

5 Click **Evaluate**.

TABLE 1

1 Go to the **Table 1** window.

For the BEM rod, the error should be about 0.01-0.2% to ~1%, depending on `hf`. The BEM probe result is more constant, and settles around ~0.2%.

Modeling Instructions — Mesh Convergence Study FEM

MESH 1

For this part, the finite element interface **Magnetic Fields, No Currents** is used. Therefore, some domains will have to be meshed using a tetrahedral mesh (indicated with "solid" in Figure 2). Start by creating a duplicate of the BEM mesh.

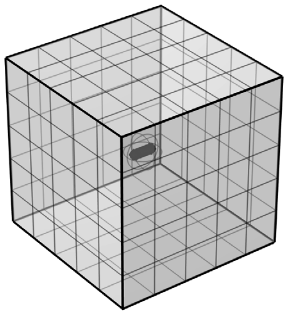
In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Duplicate**.

MESH 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Meshes** right-click **Mesh 2** and choose **Build All**.

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

3 Click the **Transparency** button in the **Graphics** toolbar.

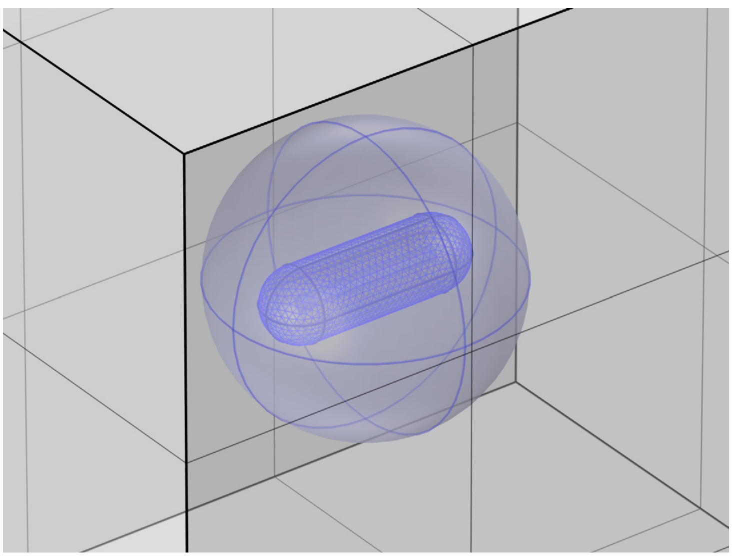


Free Tetrahedral 1

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

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

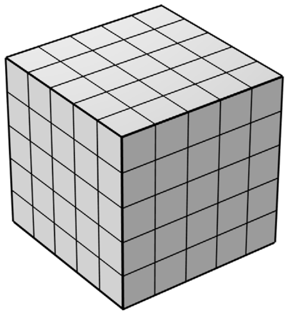
3 Click the **Zoom to Selection** button in the **Graphics** toolbar.



Notice that only the sphere has been filled with tetrahedra, not the cube. This is because the sphere is a solid domain, whereas the cube is a surface (a void). The setting responsible for this is the cube's **Object Type**.

4 Click the **Transparency** button in the **Graphics** toolbar.

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



Next, add a new study, specifically for the two coupled physics interfaces.

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 **Physics interfaces in study** and **Multiphysics couplings in study** subsections. In the tables, enter the following settings:

Physics interface, Multiphysics couplings	Solve for
Magnetic Fields, No Currents, Boundary Elements (mfncbe)	No
Magnetic Fields, No Currents (mfnc)	Yes
Magnetic Fields, No Currents, Boundary Elements 2 (mfncbe2)	Yes
Magnetic Scalar-Scalar Potential Coupling 1 (msspc1)	Yes

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

5 Right-click and choose **Add Study**.

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

STUDY 2

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

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

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 (*make sure the parameter unit is cleared*):

Parameter name	Parameter value list	Parameter unit
hf (Mesh scaling factor for pole fillet)	`0.5 1 2 3 4 5`	

Step 1: Stationary

Before hitting compute, *it is recommended to save your model.*

Solving should take about 4 minutes on an average desktop machine. The required RAM is somewhere around 6 GB. If the model is too demanding for your current setup, consider changing the parameter setting `hf` from `0.5 1 2 3 4 5`, to `0.5 1 2 3` (that is, solving for the coarser meshes only).

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

RESULTS

First, add a selection to the solution (**All domains**, in the same fashion as done for the BEM study).

Selection

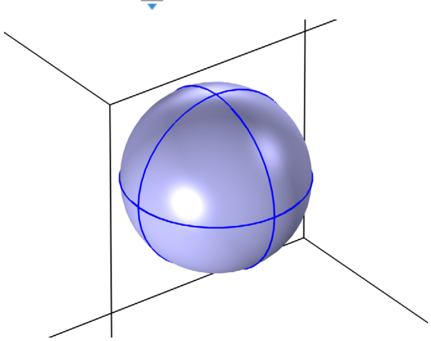
1 In the **Model Builder** window, right-click **Study 2/Solution 6 (sol6)** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

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

4 From the **Selection** list, choose **All domains**.

5 Click the **Zoom to Selection** button in the **Graphics** toolbar.



The next plot group will be about the norm of the surface stress tensor on the poles (as for the BEM study).

Maxwell Surface Stress Tensor (FEM Rod)

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Maxwell Surface Stress Tensor (FEM Rod)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/ Parametric Solutions 2 (sol7)**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

5 Locate the **Selection** section. From the **Geometric entity level** list, choose **Boundary**.

6 From the **Selection** list, choose **Magnetized Rod Pole Surface**.

Surface 1

1 Right-click **Maxwell Surface Stress Tensor (FEM Rod)** and choose **Surface**.

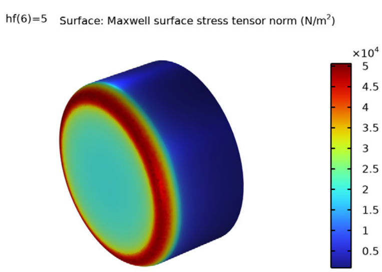
2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `sqrt(mfnc.nToutx_FEM_rod^2+ mfnc.nTouty_FEM_rod^2+mfnc.nToutz_FEM_rod^2)`.

4 Select the **Description** check box. In the associated text field, type `Maxwell surface stress tensor norm`.

5 In the **Maxwell Surface Stress Tensor (FEM Rod)** toolbar, click **Plot**.

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



Surface 2

1 In the **Model Builder** window, right-click **Maxwell Surface Stress Tensor (FEM Rod)** and choose **Surface**.

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

3 In the **Expression** text field, type `0`.

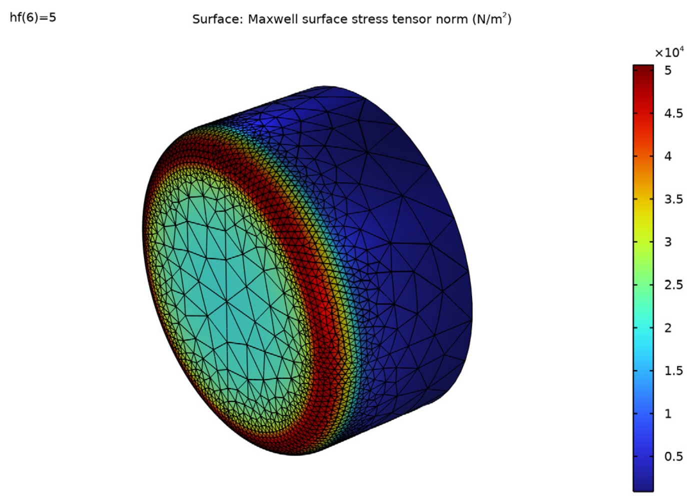
4 Locate the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

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

7 Select the **Wireframe** check box.

8 In the **Maxwell Surface Stress Tensor (FEM Rod)** toolbar, click **Plot**.



Again, feel free to investigate the plot for different values of the mesh scaling factor `hf`. Next, create a plot for the surface stress tensor on the force probe surface.

Maxwell Surface Stress Tensor (FEM Probe)

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Maxwell Surface Stress Tensor (FEM Probe)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 6 (sol6)**.

Select a suitable view (notice that this view has been created when selecting a new view for the second plot group).

4 Locate the **Plot Settings** section. From the **View** list, choose **View 3D 3**.

Surface 1

1 Right-click **Maxwell Surface Stress Tensor (FEM Probe)** and choose **Surface**.

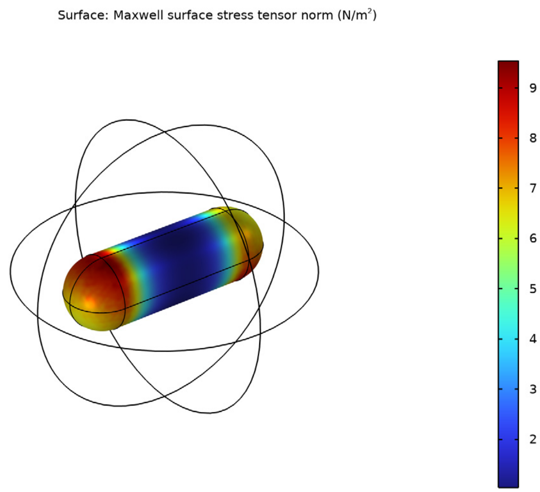
2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `sqrt(mfnc.nToutx_FEM_probe^2+mfnc.nTouty_FEM_probe^2+mfnc.nToutz_FEM_probe^2)`.

4 Select the **Description** check box. In the associated text field, type `Maxwell surface stress tensor norm`.

5 In the **Maxwell Surface Stress Tensor (FEM Probe)** toolbar, click **Plot**.

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



Notice that just as for the BEM study, the field is more smooth here.

Evaluate the error for the total torque as a function of `hf`. Since the analytical result has a magnitude of one, the error can be interpreted both as a relative (dimensionless) error, or as an absolute error (in Newton-meters).

Global Evaluation 2

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

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

3 From the **Dataset** list, choose **Study 2/Parametric Solutions 2 (sol7)**.

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

Expression	Description
`sqrt(mfnc.Tx_FEM_rod^2+(mfnc.Ty_FEM_rod-1[N*m])^2+mfnc.Tz_FEM_rod^2)`	`Magnetic torque error (FEM rod)`
`sqrt(mfnc.Tx_FEM_probe^2+(mfnc.Ty_FEM_probe-1[N*m])^2+mfnc.Tz_FEM_probe^2)`	`Magnetic torque error (FEM probe)`

5 Click **Evaluate**.

TABLE 2

1 Go to the **Table 2** window.

For the FEM rod, the error should converge from somewhere in the interval 100–600% to about 0.3–7.0%, depending on `hf`. The FEM probe result is generally more accurate, at about 0.5–2.0%.

RESULTS

Finally, create a table graph in order to show these results in a more comprehensible form.

Magnetic Torque Error

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

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

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

4 In the **Title** text area, type `Mesh convergence of magnetic torque`.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** check box. In the associated text field, type `Mesh scaling factor for pole fillet (hf)`.

7 Select the **y-axis label** check box. In the associated text field, type `Error with respect to analytical model`.

Table Graph 1

1 Right-click **Magnetic Torque Error** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, locate the **Coloring and Style** section.

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

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

5 In the **Magnetic Torque Error** toolbar, click **Plot**.

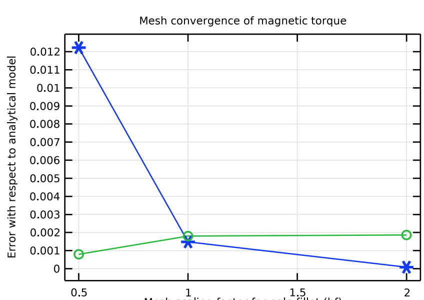
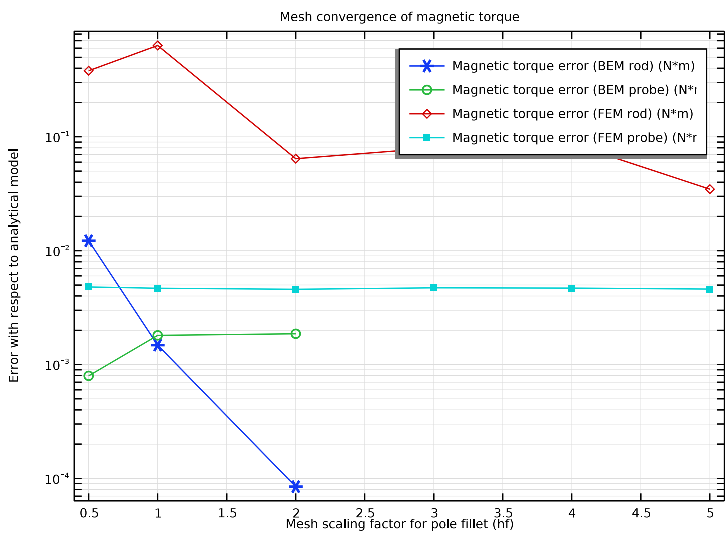


Table Graph 2

1 In the **Model Builder** window, right-click **Magnetic Torque Error** and choose **Table Graph**.
2 In the **Settings** window for **Table Graph**, locate the **Data** section.
3 From the **Table** list, choose **Table 2**.
4 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
5 Locate the **Legends** section. Select the **Show legends** check box.
6 In the **Magnetic Torque Error** toolbar, click **Plot**.
7 Click the **y-Axis Log Scale** button in the **Graphics** toolbar.



Observe that the log scale is a very effective tool for displaying convergence behavior. Without it, only the “FEM rod” curve would be clearly distinguishable from the others.

In conclusion, these results show great potential for the boundary element method, even though its implementation in COMSOL Multiphysics is fairly new.

You have now completed this tutorial series, the other tutorials in this series will refer to the resulting file as `force_calculation_03_magnetic_torque_bem.mph`.