Model created in COMSOL Multiphysics 6.4

# Electromagnetic Forces on Parallel Current-Carrying Wires

## *Introduction*

One ampere is defined as the constant current in two straight parallel conductors of infinite length and negligible circular cross section, placed one meter apart in vacuum, that produces a force of $2 \times 10^{-7}$ newton per meter of length (N/m). This example shows a setup of two parallel wires in the spirit of this definition, but with the difference that the wires have finite cross sections.

For wires with circular cross section carrying a uniform current density as in this example, the mutual magnetic force is the same as for line currents. This can be understood by the following arguments: Start from a situation where both wires are line currents ($\mathbf{I}$). Each line current is subject to a Lorentz force per meter ($\mathbf{I} \times \mathbf{B}$), where the magnetic flux density ($\mathbf{B}$) is the one produced by the other wire. Now, give one wire a finite radius. It follows directly from circular symmetry and Maxwell–Ampère's law that, outside this wire, the produced flux density is exactly the same as before so the force on the remaining line current is unaltered. Further, the net force on the wire with the distributed current density must be of exactly the same magnitude (but with opposite direction) as the force on the line current so that force did not change either. If the two wires exchange places, the forces must still be the same, and it follows from symmetry that the force is independent of wire radius as long as the wire cross sections do not intersect. The wires can even be cylindrical shells or any other shape with circular symmetry. For an experimental setup, negligible cross section is required as resistive voltage drop along the wires and Hall effect may cause electrostatic forces that increase with wire radius but such effects are not included in this example.

The force between the wires is computed using two different methods: first automatically by integrating the stress tensor on the boundaries, then by integrating the volume (Lorentz) force density over the wire cross section. The results converge to $2 \times 10^{-7}$ N/m for the 1 ampere definition, as expected.

## *Model Definition*

The application is built using the 2D Magnetic Fields interface. The modeling plane is a cross section of the two wires and the surrounding air.

### DOMAIN EQUATIONS

The equation formulation assumes that the only nonzero component of the magnetic vector potential is $A_z$. This corresponds to all currents being perpendicular to the modeling plane. The following equation is solved:

$$\nabla \times (\mu^{-1} \nabla \times A_z \mathbf{e}_z) = J_z^e \mathbf{e}_z$$

where μ is the permeability of the medium and $J_z^e$ is the externally applied current density. $J_z^e$ is set so that the applied current in the wires equals 1 A, but with different signs.

Surrounding the air is an infinite element domain. For details, see the *AC/DC Module User's Guide*.

## *Results and Discussion*

The magnetic flux density of the two current carrying coils is shown in Figure 1 where the direction and magnitude of the magnetic flux density are illustrated with streamlines and color scale, respectively.

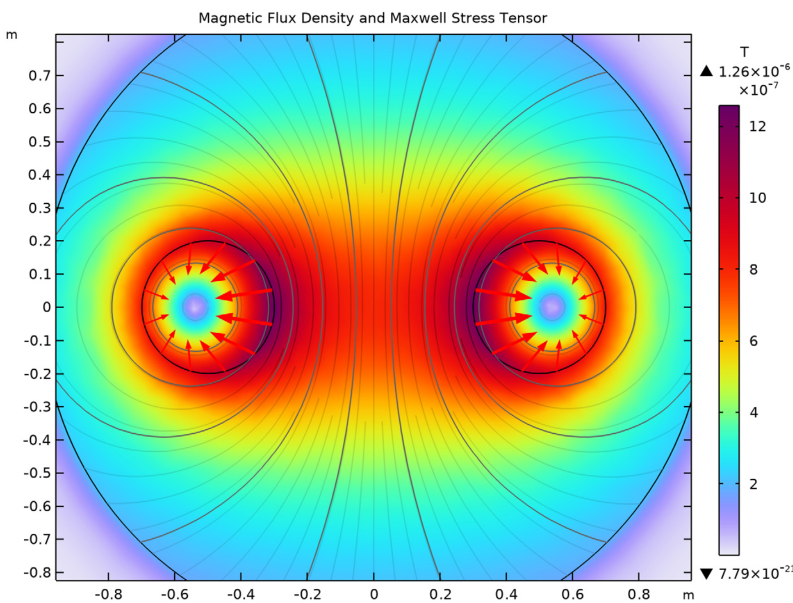



*Figure 1: The magnetic flux density of the two current carrying coils.*

As shown in Figure 1, the Maxwell surface stress tensor on the boundaries of conductors is represented by arrow plots. The expression for the surface stress reads

$$\mathbf{n}_1 \cdot T_2 = -\frac{1}{2}(\mathbf{H} \cdot \mathbf{B})\mathbf{n}_1 + (\mathbf{n}_1 \cdot \mathbf{H})\mathbf{B}^T$$

where $\mathbf{n}_1$ is the boundary normal pointing out from the conductor wire and $T_2$ the stress tensor of air. The closed line integral of this expression around the circumference of either wire evaluates to $-1.99 \times 10^{-7}$ N/m. The minus sign indicates that the force between the wires is repulsive. The software automatically provides the coordinate components of the force on each wire.

The volume force density is given by

$$\mathbf{F} = \mathbf{J} \times \mathbf{B} = \left[-J_z^{\mathrm{e}} \cdot B_y, J_z^{\mathrm{e}} \cdot B_x, 0\right]$$

The surface integral of the $x$ component of the volume force on the cross section of a wire gives the result $-2.00 \times 10^{-7}$ N/m.

By refining the mesh and re-solving the problem, you can verify that the solution with both methods converges to $-2 \times 10^{-7}$ (N/m), as discussed in detail next. The volume force density integral is typically the most accurate one for reasons explained in the *COMSOL Multiphysics Reference Manual*.

### MESH CONVERGENCE

In order to investigate the accuracy of the model, it is recommended to perform a systematic mesh convergence analysis of the desired entity, here the force on the wire. In Figure 2 and Figure 3, the mesh convergence is shown for the absolute errors in the Maxwell surface stress method and the volumetric Lorentz force method, respectively. The

Lorentz force is 2–3 orders of magnitude more accurate than the Maxwell stress tensor force for a given mesh density.

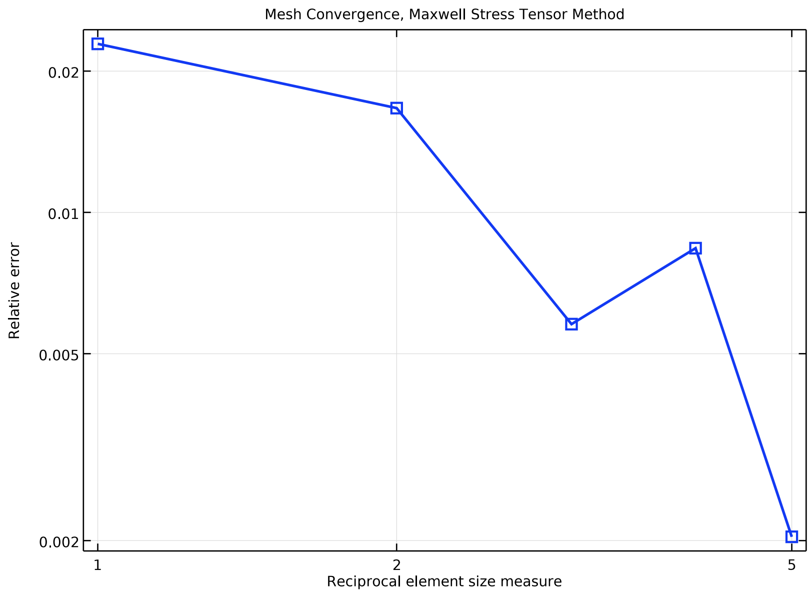



*Figure 2: Mesh convergence is shown for the force computation using the Maxwell surface stress method.*

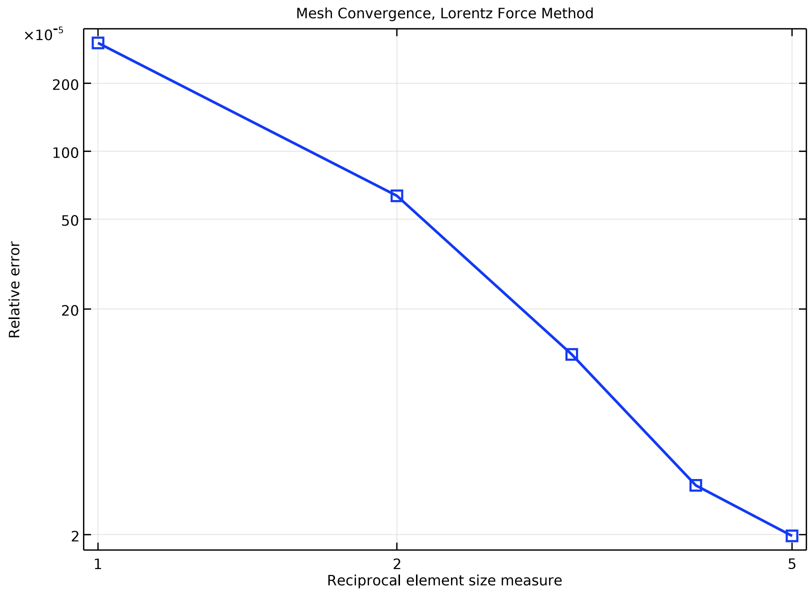



*Figure 3: Mesh convergence is shown for the force computation using the volumetric Lorentz force method.*

---

**Application Library path:** `ACDC_Module/Introductory_Electromagnetic_Forces/parallel_wires`

---

## *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**.

2 In the **Select Physics** tree, select **AC/DC** > **Electromagnetic Fields** > **Magnetic Fields (mf)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

## 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 |
|---|---|---|---|
| r_wire | 0.2[m] | 0.2 m | Wire radius |
| I0 | 1[A] | 1 A | Total current |
| J0 | I0/(pi*r_wire^2) | 7.9577 A/m² | Current density |
| N | 1 | 1 | Mesh multiplier |

## GEOMETRY 1

Add a circle for the main air domain. The outer layer will constitute an infinite element domain to approximate a region extending to infinity.

### *Circle 1 (c1)*

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

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

3 In the **Radius** text field, type 1.5.

4 Click to expand the **Layers** section. In the table, enter the following settings:

| Layer name | Thickness (m) |
|---|---|
| Layer 1 | 0.5 |

### *Circle 2 (c2)*

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

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

3 In the **Radius** text field, type r_wire.

4 Locate the **Position** section. In the **x** text field, type 0.5.

### *Circle 3 (c3)*

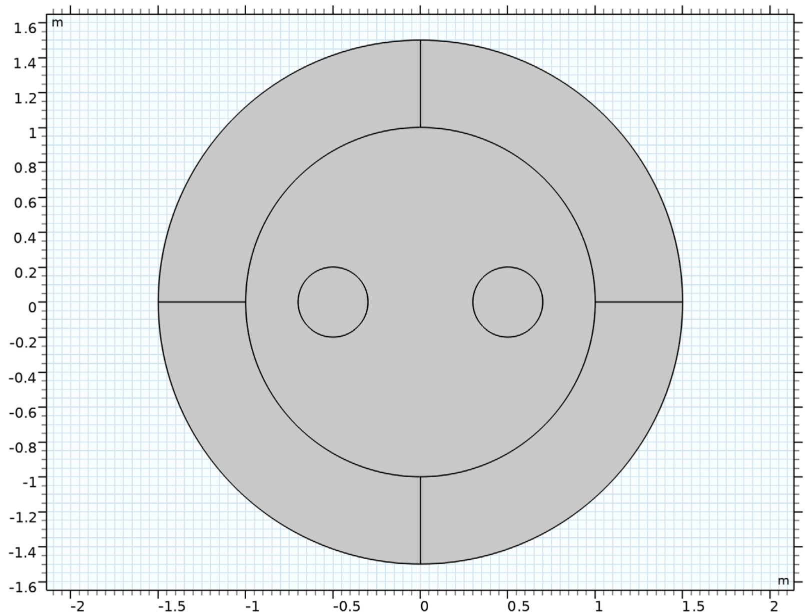
1 In the **Geometry** toolbar, click **Circle**.

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

**3** In the **Radius** text field, type `r_wire`.

**4** Locate the **Position** section. In the **x** text field, type `-0.5`.

**5** Click **Build All Objects**.




## DEFINITIONS

Define an infinite element region in the outer domains.

### *Infinite Element Domain 1 (ie1)*

**1** In the **Definitions** toolbar, click **Infinite Element Domain**.

**2** Select Domains 1–4 only.

**3** In the **Settings** window for **Infinite Element Domain**, locate the **Geometry** section.

**4** From the **Type** list, choose **Cylindrical**.

## ADD MATERIAL

**1** In the **Materials** 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** Click the **Add to Component** button in the window toolbar.

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

### MAGNETIC FIELDS (MF)

By default, the first material you select will apply to your entire geometry. Air is defined with a zero conductivity, and relative permittivity and permeability both equal to 1. These properties are the same as those of vacuum, which is the assumed material in the definition of the ampere. Since the model assumes a given static and uniform current distribution, the electric conductivity of the wires does not appear in the equations, so it is safe to use the same properties in the wires too.

*External Current Density 1*

1. In the **Physics** toolbar, click **Domains** and choose **External Current Density**.
2. Select Domain 6 only(the wire on the left).
3. In the **Settings** window for **External Current Density**, locate the **External Current Density** section.
4. Specify the $\mathbf{J}_e$ vector as

| | |
|---|---|
| 0 | x |
| 0 | y |
| J0 | z |

*External Current Density 2*

1. In the **Physics** toolbar, click **Domains** and choose **External Current Density**.
2. Select Domain 7 only(the wire on the right).
3. In the **Settings** window for **External Current Density**, locate the **External Current Density** section.
4. Specify the $\mathbf{J}_e$ vector as

| | |
|---|---|
| 0 | x |
| 0 | y |
| -J0 | z |

The definition of the physics of the system is now complete. Add a Force Calculation feature to make Maxwell's stress tensor available as a variable.

*Force Calculation 1*

1. In the **Physics** toolbar, click **Domains** and choose **Force Calculation**.
2. Select Domain 6 only.

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

4 In the **Force name** text field, type `wire1`.

*Force Calculation 2*

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

2 Select Domain 7 only.

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

4 In the **Force name** text field, type `wire2`.

The infinite element domain requires some attention when meshing. As it is steeply scaled in the radial direction to model a very large geometry (approximating a geometry extending to infinity), the mesh will effectively be stretched in that direction. A structured mesh is indicated in this case to prevent poor element quality. The Magnetic Fields interface can automatically create an appropriate mesh for this application.

### MESH 1

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

   The automatically created mesh applies a Mapped operation on the finite element domain. Modify it according to the following instructions.

2 Right-click **Component 1 (comp1)** > **Mesh 1** and choose **Edit Physics-Induced Sequence**.

*Size*

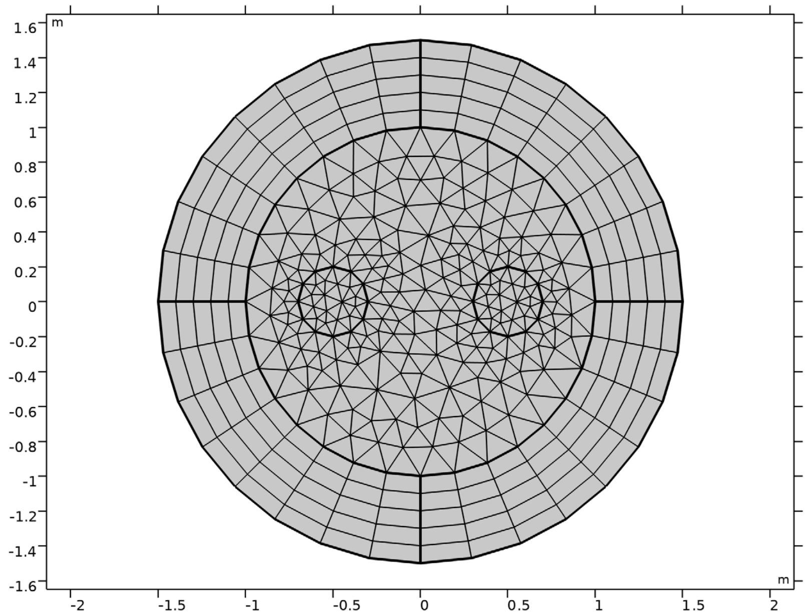
1 In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh 1** click **Size**.

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

3 From the **Predefined** list, choose **Coarser**.

4 Click to expand the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.2`.

**5** Click **Build All**.




The mesh should look like in the figure.

## STUDY 1

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

## RESULTS

*Magnetic Flux Density and Maxwell Stress Tensor*

**1** In the **Settings** window for **2D Plot Group**, click to expand the **Title** section.

**2** From the **Title type** list, choose **Label**.

**3** In the **Label** text field, type `Magnetic Flux Density and Maxwell Stress Tensor`.

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

The default plot shows the norm of the magnetic flux density. Note that the value inside the infinite element domain has no physical relevance.

*Arrow Line 1*

**1** Right-click **Magnetic Flux Density and Maxwell Stress Tensor** and choose **Arrow Line**.

**2** In the **Settings** window for **Arrow Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** >

**Magnetic Fields** > **Equivalent face load** > **mf.nToutx_wire1,mf.nTouty_wire1 - Maxwell surface stress tensor**.

3 Locate the **Coloring and Style** section.

4 Select the **Scale factor** checkbox. In the associated text field, type 300000.

5 In the **Magnetic Flux Density and Maxwell Stress Tensor** toolbar, click **Plot**.

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

### *Arrow Line 2*

1 Right-click **Magnetic Flux Density and Maxwell Stress Tensor** and choose **Arrow Line**.

2 In the **Settings** window for **Arrow Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Magnetic Fields** > **Equivalent face load** > **mf.nToutx_wire2,mf.nTouty_wire2 - Maxwell surface stress tensor**.

3 Locate the **Coloring and Style** section.

4 Select the **Scale factor** checkbox. In the associated text field, type 300000.

5 In the **Magnetic Flux Density and Maxwell Stress Tensor** toolbar, click **Plot**.

The plot shows the Maxwell stress tensor distribution on the surface of the wires. The total force on each wire is evaluated as the surface integral of the stress tensor and is available as a postprocessing variable.

### *Global Evaluation 1*

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

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Magnetic Fields** > **Mechanical** > **Electromagnetic force - N** > **mf.Forcex_wire1 - Electromagnetic force, x-component**.

3 Click **Evaluate**.

## TABLE 1

1 Go to the **Table 1** window. The force in the $x$ direction on the first wire evaluates to a value between $-2.0\times10^{-7}$ N/m and $-1.9\times10^{-7}$ N/m. Note that the force is given per meter, since the 2D model has an out-of-plane thickness of 1 m.

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Magnetic Fields** > **Mechanical** > **Electromagnetic force - N** > **mf.Forcex_wire2 - Electromagnetic force, x-component**.

**3** Click **Evaluate**.

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

As expected, the force on the second wire has a similar value but the opposite sign.

Proceed to compare the value with those from the Lorentz force distribution.

## RESULTS

*Surface Integration 1*

**1** In the **Results** toolbar, click **More Derived Values** and choose **Integration** > **Surface Integration**.

**2** Select Domain 6 only.

**3** In the **Settings** window for **Surface Integration**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Magnetic Fields** > **Mechanical** > **Lorentz force contribution, instantaneous value - N/m³** > **mf.FLtzix - Lorentz force contribution, instantaneous value, x-component**.

**4** Click **Evaluate**.

## TABLE 2

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

This time, the value is expected to be consistently closer to $-2{\times}10^{-7}$ N/m. When applicable, Lorentz force integrals usually give more accurate results than the Maxwell stress tensor.

**2** Select Domain 7 only.

## RESULTS

**1** In the **Model Builder** window, click **Surface Integration 1**.

**2** In the **Settings** window for **Surface Integration**, click **Evaluate**.

Once again, integration over the second wire gives a similar but positive result.

## MESH 1

Proceed with the mesh convergence analysis for the force. Create a parameterized mesh.

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

## MESH 2

*Size*

1 In the **Model Builder** window, expand the **Mesh 2** node, then click **Size**.
2 In the **Settings** window for **Size**, locate the **Element Size Parameters** section.
3 In the **Maximum element size** text field, type `0.2/N`.

*Distribution 1*

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Meshes** > **Mesh 2** > **Mapped 1** node, then click **Component 1 (comp1)** > **Meshes** > **Mesh 2** > **Distribution 1**.
2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
3 In the **Number of elements** text field, type `4*N`.

Perform the mesh convergence analysis in a new study.

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

## STUDY 2

Perform a sweep over the mesh multiplier parameter.

*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 From the list in the **Parameter name** column, choose **N (Mesh multiplier)**.
5 Click **Range**.
6 In the **Range** dialog, type `1` in the **Start** text field.
7 In the **Step** text field, type `1`.
8 In the **Stop** text field, type `5`.
9 Click **Replace**.

Define a nonlocal integration coupling to compute the total force from the Lorentz force contribution.

#### DEFINITIONS

*Integration 1 (intop1)*

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 Select Domain 7 only.

#### STUDY 2

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

#### RESULTS

*Magnetic Flux Density (Mesh Convergence Study)*

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

2 From the **Title type** list, choose **Label**.

3 In the **Label** text field, type `Magnetic Flux Density (Mesh Convergence Study)`.

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

*1D Plot Group 3*

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

Plot the absolute error versus the mesh multiplier parameter for the force computed using Maxwell's stress tensor.

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

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

*Global 1*

1 Right-click **1D Plot Group 3** and choose **Global**.

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

3 In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| `abs(mf.Forcex_wire2-2e-7)/2e-7` | N | |

4 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

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

Switch to logarithmic scale and add suitable plot annotations.

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

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

*Mesh Convergence, Maxwell Stress Tensor Method*

1 In the **Model Builder** window, click **1D Plot Group 3**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Label**.

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** checkbox. In the associated text field, type `Reciprocal element size measure`.

6 Select the **y-axis label** checkbox. In the associated text field, type `Relative error`.

7 In the **Label** text field, type `Mesh Convergence, Maxwell Stress Tensor Method`.

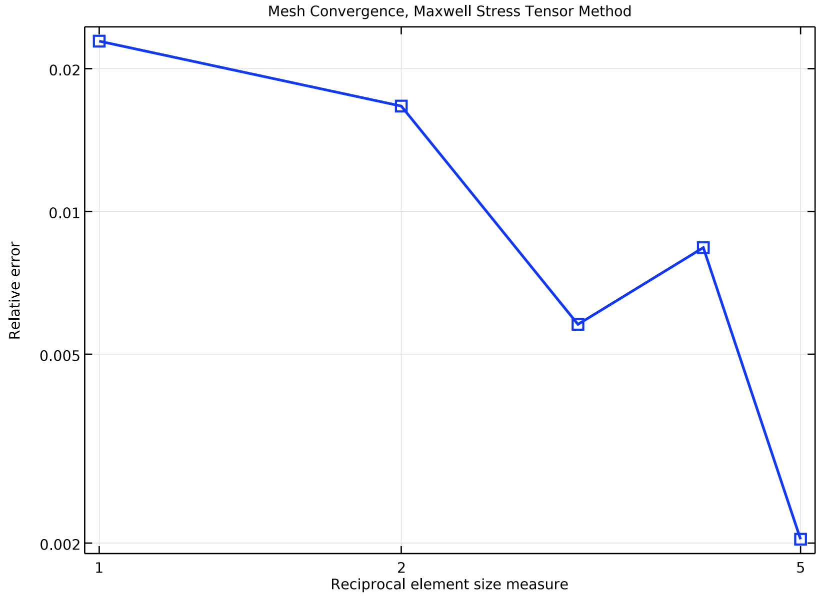
*Global 1*

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

2 In the **Settings** window for **Global**, click to expand the **Legends** section.

3 Clear the **Show legends** checkbox.

**4** In the **Mesh Convergence, Maxwell Stress Tensor Method** toolbar, click **Plot**.




*Mesh Convergence, Maxwell Stress Tensor Method*

Plot the absolute error versus the mesh multiplier parameter for the force computed using the Lorentz force contribution.

*Mesh Convergence, Lorentz Force Method*

**1** In the **Model Builder** window, right-click **Mesh Convergence, Maxwell Stress Tensor Method** and choose **Duplicate**.

**2** In the **Settings** window for **1D Plot Group**, type `Mesh Convergence, Lorentz Force Method` in the **Label** text field.

*Global 1*

**1** In the **Model Builder** window, expand the **Mesh Convergence, Lorentz Force Method** node, then click **Global 1**.

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

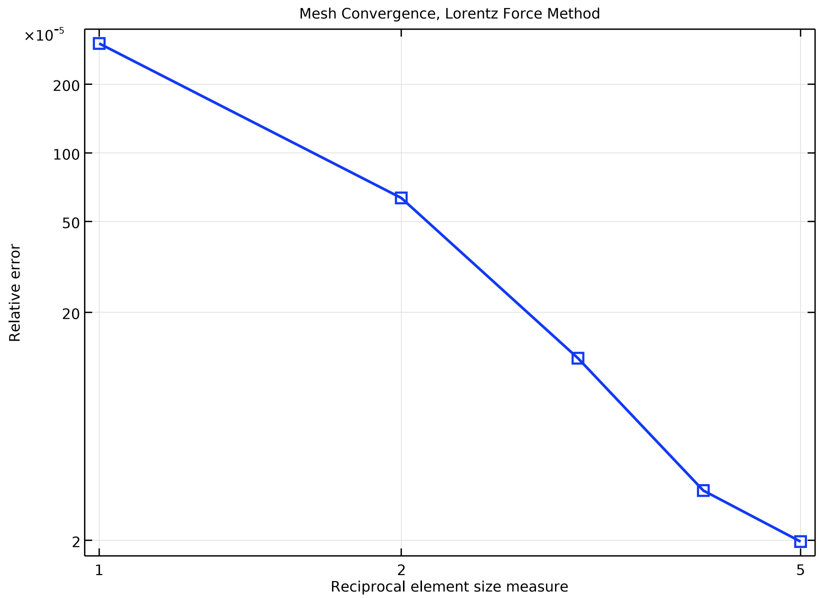
**3** In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| `abs(intop1(mf.FLtzix)-2e-7)/2e-7` | `N/m` | |

*Mesh Convergence, Lorentz Force Method*

**1** In the **Model Builder** window, click **Mesh Convergence, Lorentz Force Method**.

**2** In the **Mesh Convergence, Lorentz Force Method** toolbar, click **Plot**.




Using the Lorentz force method gives results that are 2-3 orders of magnitude more accurate than the ones obtained using Maxwell's stress tensor for a given mesh density.

In the following, it is presented another way to visualize Maxwell Stress tensor together with the Lorentz Force. The plot highlights clearly that the Maxwell Stress tensor is a boundary vector (whose main property is that its surface integral is the total force on the body) and that the Lorentz Force is a volumetric force. Differently from Maxwell Stress tensor, Lorentz Force is an actual volumetric force, and, for the present case where the objects are nonmagnetic conductor, Lorentz Force is the only contribution to the total force.

*Lorentz Force and Maxwell Stress Tensor*

**1** In the **Results** toolbar, click **2D Plot Group**.

**2** In the **Settings** window for **2D Plot Group**, type `Lorentz Force and Maxwell Stress Tensor` in the **Label** text field.

First, remove representation of all the edges, and reproduce only the coil edges twice, one above the other.

**3** Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

**4** Click to expand the **Selection** section. From the **Geometric entity level** list, choose **Domain**.

**5** Select Domains 6 and 7 only.

*Line 1*

**1** Right-click **Lorentz Force and Maxwell Stress Tensor** and choose **Line**.

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

**3** In the **Expression** text field, type `1`.

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

**5** From the **Coloring** list, choose **Uniform**.

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

*Line 2*

Right-click **Line 1** and choose **Duplicate**.

*Transformation 1*

**1** In the **Model Builder** window, right-click **Line 2** and choose **Transformation**.

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

**3** In the **y** text field, type `0.5`.

Where the **Translation** feature is used to displace the two upper circles.

Add a title well representing what is going to be shown.

*Lorentz Force and Maxwell Stress Tensor*

**1** In the **Model Builder** window, under **Results** click **Lorentz Force and Maxwell Stress Tensor**.

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

**3** From the **Title type** list, choose **Manual**.

**4** In the **Title** text area, type `Maxwell Stress Tensor (Red) and Lorentz Force (Blue)`.

Now add Maxwell Stress Tensor representation on both conductors.

*Arrow Line 1*

**1** Right-click **Lorentz Force and Maxwell Stress Tensor** and choose **Arrow Line**.

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

3 In the **x-component** text field, type `try_catch(mf.nToutx_wire1, mf.nToutx_wire2)`.

4 In the **y-component** text field, type `try_catch(mf.nTouty_wire1, mf.nTouty_wire2)`.

5 Locate the **Coloring and Style** section.

6 Select the **Scale factor** checkbox. In the associated text field, type `400000`.

7 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `20`.

Finally add Lorentz Force representation, shifted up.

*Arrow Surface 1*

1 Right-click **Lorentz Force and Maxwell Stress Tensor** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, locate the **Coloring and Style** section.

3 From the **Color** list, choose **Blue**.

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Magnetic Fields** > **Mechanical** > **mf.FLtzix, mf.FLtziy - Lorentz force contribution, instantaneous value**.

5 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type `21`.

6 Find the **y grid points** subsection. In the **Points** text field, type `11`.

7 Locate the **Coloring and Style** section.

8 Select the **Scale factor** checkbox. In the associated text field, type `10000`.

*Transformation 1*

1 Right-click **Arrow Surface 1** and choose **Transformation**.

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

3 In the **y** text field, type `0.5`.

Execute the plot and rescale the view to get a representation of Maxwell Stress Tensor and Lorentz Force. The result will look like in the following figure.

*Lorentz Force and Maxwell Stress Tensor*

1 In the **Model Builder** window, under **Results** click **Lorentz Force and Maxwell Stress Tensor**.

2 In the **Lorentz Force and Maxwell Stress Tensor** toolbar, click **Plot**.

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