Simcenter 3D Low Frequency EM
Tutorial #3

Linear Actuator

Version: Simcenter 3D 2019.2
# Table of Contents

1. Introduction .......................................................................................................................... 1

2. CAD Model Pre-processing ..................................................................................................... 2
   2.1. Creating Virtual air region ................................................................................................. 2
   2.2. Creating Remesh region ..................................................................................................... 6
   2.3. Creating Air enclosure region ............................................................................................ 7

3. Finite Element model ............................................................................................................... 8
   3.1. Prepare Materials ............................................................................................................... 9
   3.2. Mesh Mating ...................................................................................................................... 10
   3.3. Generate Mesh .................................................................................................................. 11
   3.4. Define Magnetization direction ......................................................................................... 18

4. Simulation Set-Up ................................................................................................................ 20
   4.1. Create Coils ....................................................................................................................... 20
   4.2. Create excitation ............................................................................................................... 23
   4.3. Setup Motion component ................................................................................................. 24
   4.4. Setup Geometric Parameterization ................................................................................... 26
   4.5. Solve ................................................................................................................................. 30

5. Results .................................................................................................................................. 31
   5.1. B field Result .................................................................................................................... 31
   5.2. Thrust force Results ......................................................................................................... 33
Table of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Linear Actuator tutorial model in Simcenter 3D 2019.2</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>Actuator 3D assembly</td>
<td>2</td>
</tr>
<tr>
<td>3</td>
<td>Create new FEM and Simulation</td>
<td>2</td>
</tr>
<tr>
<td>4</td>
<td>Setup for new FEM and Simulation</td>
<td>3</td>
</tr>
<tr>
<td>5</td>
<td>Wave link all the bodies</td>
<td>3</td>
</tr>
<tr>
<td>6</td>
<td>Part Navigator</td>
<td>4</td>
</tr>
<tr>
<td>7</td>
<td>Modeling mode</td>
<td>4</td>
</tr>
<tr>
<td>8</td>
<td>Sketch Command</td>
<td>4</td>
</tr>
<tr>
<td>9</td>
<td>Extrude for virtual air box creation</td>
<td>4</td>
</tr>
<tr>
<td>10</td>
<td>Edit object display</td>
<td>5</td>
</tr>
<tr>
<td>11</td>
<td>Boolean Subtract</td>
<td>5</td>
</tr>
<tr>
<td>12</td>
<td>Settings for Subtract Boolean operation</td>
<td>5</td>
</tr>
<tr>
<td>13</td>
<td>Extrude remesh region</td>
<td>6</td>
</tr>
<tr>
<td>14</td>
<td>Edit object display for remesh region</td>
<td>6</td>
</tr>
<tr>
<td>15</td>
<td>Boolean Subtraction for remesh region</td>
<td>6</td>
</tr>
<tr>
<td>16</td>
<td>Extrude air region</td>
<td>7</td>
</tr>
<tr>
<td>17</td>
<td>Edit object display for air region</td>
<td>7</td>
</tr>
<tr>
<td>18</td>
<td>Boolean Subtract for air region</td>
<td>7</td>
</tr>
<tr>
<td>19</td>
<td>Pre/Post command</td>
<td>8</td>
</tr>
<tr>
<td>20</td>
<td>Make fem the work part</td>
<td>8</td>
</tr>
<tr>
<td>21</td>
<td>Edit FEM settings</td>
<td>8</td>
</tr>
<tr>
<td>22</td>
<td>Physical Properties</td>
<td>9</td>
</tr>
<tr>
<td>23</td>
<td>Create physical property table</td>
<td>9</td>
</tr>
<tr>
<td>24</td>
<td>Physical Property table</td>
<td>9</td>
</tr>
<tr>
<td>25</td>
<td>Mesh Mating command</td>
<td>10</td>
</tr>
<tr>
<td>26</td>
<td>Settings for Mesh Mating Condition</td>
<td>10</td>
</tr>
<tr>
<td>27</td>
<td>Mesh Mating Condition information</td>
<td>10</td>
</tr>
<tr>
<td>28</td>
<td>Find Exterior Faces command</td>
<td>10</td>
</tr>
<tr>
<td>29</td>
<td>3D Tetrahedral command</td>
<td>11</td>
</tr>
<tr>
<td>30</td>
<td>3D Tetrahedral Mesh Settings</td>
<td>11</td>
</tr>
<tr>
<td>31</td>
<td>Group selection</td>
<td>12</td>
</tr>
<tr>
<td>32</td>
<td>3D mesh settings for magnets</td>
<td>12</td>
</tr>
<tr>
<td>33</td>
<td>Magnets mesh collector</td>
<td>12</td>
</tr>
<tr>
<td>34</td>
<td>3D Mesh Settings for VA air</td>
<td>13</td>
</tr>
<tr>
<td>35</td>
<td>VA air mesh Collector</td>
<td>13</td>
</tr>
<tr>
<td>36</td>
<td>3D Mesh Settings for Remesh</td>
<td>13</td>
</tr>
<tr>
<td>37</td>
<td>Remesh mesh collector</td>
<td>14</td>
</tr>
<tr>
<td>38</td>
<td>3D Mesh Settings for Coils</td>
<td>14</td>
</tr>
<tr>
<td>39</td>
<td>Coils mesh collector</td>
<td>14</td>
</tr>
<tr>
<td>40</td>
<td>2D seed mesh Settings for stator tooth</td>
<td>15</td>
</tr>
<tr>
<td>41</td>
<td>2D Seed Mesh Collector</td>
<td>15</td>
</tr>
<tr>
<td>42</td>
<td>3D Mesh Settings for Outer Stator</td>
<td>15</td>
</tr>
<tr>
<td>43</td>
<td>Outer Stator mesh collector</td>
<td>16</td>
</tr>
<tr>
<td>44</td>
<td>3D Mesh Settings for Inner Stator</td>
<td>16</td>
</tr>
<tr>
<td>45</td>
<td>Inner Stator mesh collector</td>
<td>16</td>
</tr>
<tr>
<td>46</td>
<td>3D Mesh Settings for Air box</td>
<td>17</td>
</tr>
<tr>
<td>47</td>
<td>Air box mesh collector</td>
<td>17</td>
</tr>
<tr>
<td>48</td>
<td>Set mesh collector color</td>
<td>17</td>
</tr>
</tbody>
</table>
1. Introduction
The transverse flux linear oscillatory actuator is a special motor that performs a linear reciprocating motion, with a certain stroke at a specific frequency. With the advantages of having a high transmission efficiency, a compact structure, low noise, etc., this type of motor is more suitable for artificial hearts, pumps, refrigeration, compressors and other similar applications.

Since the actuator’s flux paths and mover displacement involves radial, axial and circumferential directions, 3D simulation is needed for the analysis. This tutorial will study the effect of motor axial dimension, on the thrust force and stroke, by using the geometric parameter feature in the Simcenter Low-Frequency EM environment. It will cover the model’s pre-processing and the post-processing of the typical results; i.e. the flux density distribution and thrust force.

It is important to note that in low-frequency electromagnetic analysis, the device under analysis needs to be enclosed with a fluid material (in most cases, air), which defines the computation domain. Applying the proper boundary conditions to the device provides a unique solution for the problem.

![Figure 1: Linear Actuator tutorial model in Simcenter 3D 2019.2](image)

Estimated duration: 1.5 hours
2. CAD Model Pre-processing
In this section, the CAD model is prepared for finite element discretization and EM simulation setup. This involves adding the enclosing air region.

In the Start model folder, you will find all the part files and one assembly part file “Linear Actuator Tutorial.prt”. We will start from this model.

- Launch Simcenter 3D by double-clicking the desktop icon
- Click File | Open, in Open window, navigate to Start model folder, select Part File(*.prt) as Files of type and select “Linear Actuator Tutorial.prt”, then click OK
- The “Linear Actuator Tutorial.prt” is a full model of the assembly of the actuator as shown below:

![Figure 2: Actuator 3D assembly](image)

2.1. Creating Virtual air region
In this model, the moving parts include six disjoined magnets. For the 3D motion setup, the motion component should be one joined body; this can be done by creating a small virtual air box to enclose all of them. Another benefit of using virtual air is to improve the accuracy of the force calculation.

- Right-click Linear Actuator Tutorial.prt in the Simulation Navigator and select New FEM and Simulation

![Figure 3: Create new FEM and Simulation](image)
In the **New FEM and Simulation** setup window:
- Check the option **Create Idealized Part**
- Set “**Polygon Body Resolution**” to High
- Set **Solver** to Simcenter MAGNET
- Set **Analysis Type** to Electromagnetic 3D

![Figure 4: Setup for new FEM and Simulation](image)

After clicking **OK**, the solution setup window will pop up. We will return to the solution setup later so, for now, click **Cancel**

Double click “**Linear Actuator Tutorial_fem1_i.prt**” in the **Simulation Navigator** to make it the work part

Click “**Wave**” under **Home|Start** tab from the menu ribbon

In the **Wave Geometry Linker** window, use Ctrl-A to select all the bodies, make sure the option **Hide Original** is checked, and then click **OK**

![Figure 5: Wave link all the bodies](image)
Click Part Navigator button from the side tool bar; the Part Navigator appears.

Figure 6: Part Navigator

Click Modeling from Application|Design tab from the menu ribbon; the menu ribbon switches to the modeling mode.

Figure 7: Modeling mode

Click Sketch from Home|Direct Sketch and click OK in the Create Sketch window. Draw two circles centered at origin with radii: 35.6667 mm and 39.3333 mm.

Figure 8: Sketch Command

Click Extrude from Home|Feature tab; the two circles drawn in last step are automatically choosen as Select Curve in the Extrude window. Set Start Distance= -7mm, and End Distance= 62mm.

Click OK.

Figure 9: Extrude for virtual air box creation
The extruded body is shown under the Model History list. Select it and click Edit Object Display in View | Visualization tab from the menu ribbon. Set Translucency in the Edit Object Display window for a better view.

For the purposes of meshing, Simcenter 3D does not allow any overlapping between bodies, so a Boolean subtraction is required here. Select Subtract from Home | Feature tab.

In the Subtract window, select Extrude (7) from the Part Navigator as the Target body, and Linked Body (5) as the Tool body. Check the option Keep Tool and then click OK.

Subtract (8) is listed under the Model History list. The virtual air region has now been created.
2.2. Creating Remesh region
For the motion analysis, a remesh region is needed as the interface between the stationary and moving parts. It will be re-meshed when the motion component moves. In this section, a remesh region is created for this model.

- Make a new sketch following the same steps. Draw two circles centered at origin with radii: 35.3333 mm and 39.6667 mm.
- Click Extrude from Home|Feature tab; the two circles drawn in last step are automatically choosen as Select Curve in the Extrude window. Set Start Distance= -8mm and End Distance= 103 mm, and then click OK.

![Figure 13: Extrude remesh region](image)

- Make the extruded body transparent to obtain a better view.

![Figure 14: Edit object display for remesh region](image)

- Again, a Boolean subtraction is needed. Select Subtract from Home|Feature tab.
- In the Subtract window, select Extrude (10) from Part Navigator as the Target body, and Subtract (8) and Linked Body (5) as the Tool body. Ignore the alter message. Check the option Keep Tool and click OK.

![Figure 15: Boolean Subtraction for remesh region](image)

- Subtract (11) is listed under Model History list. The remesh region has now been created.
2.3. Creating Air enclosure region

The last procedure in this section is to create a big air region that will enclose the whole model.

- Make a new sketch following the same steps. Draw a circle centered at origin with radius 96.2 mm. Click Extrude from Home | Feature tab; the two circles drawn in last step are automatically chosen as Select Curve in the Extrude window. Set Start Distance= -34 mm, and End Distance= 129 mm and then click OK.

![Figure 16: Extrude air region](image)

- Make the extruded body transparent to obtain a better view.

![Figure 17: Edit object display for air region](image)

- Again, a Boolean subtraction is needed. Select Subtract from Home | Feature tab.
- In the Subtract window, select Extrude (13) from the Part Navigator as the Target body, and use Ctrl+A select all the other bodies as the Tool body. Ignore the alter message. Check the option Keep Tool and click OK.

![Figure 18: Boolean Subtract for air region](image)

- Subtract (14) is now listed under Model History list. The air enclosure region has now been created.
3. Finite Element model

In Simcenter 3D, all the operations related to mesh and materials are done in the fem file. In this section, we will create the meshes for each part of the actuator and assign the materials to the corresponding mesh collectors. Since there are six permanent magnets, the direction of the magnetization will also have to be defined.

◇ Click Pre/Post from Application | Simulation tab to exit the modeling mode.

Figure 19: Pre/Post command

◇ The Part Navigator will switch back to the Simulation Navigator. Right-click “Linear Actuator Tutorial _fem1_i.prt” in the Simulation Navigator, select Display FEM and then “Linear Actuator Tutorial _fem1.fem”, making the fem the work part.

Figure 20: Make fem the work part

◇ Right-click “Linear Actuator Tutorial_fem1.fem” in the Simulation Navigator, then select Edit. In the Edit FEM window, check option Edit Bodies to Use and set All Visible.

◇ Click OK.

Figure 21: Edit FEM settings
3.1. Prepare Materials

The materials in the model are listed below:

- Air, remesh region: Air
- Virtual air region: Virtual air
- Coil material: Copper 5.77e7
- Core material: M-19 26 Ga
- Magnet material: Neodymium Iron Boron: 36/26

Since different mesh collectors might have the same material, the Physical Properties table will be used to better organize the material selection.

- In Home | Properties commands group, click Physical Properties.
- In the Physical Property Table Manager window, set the name to Air and click Create.
- In Solid window, click Choose material button beside Material cell.
- In the Material List window, select AIR and click OK

- In Solid window, Click OK.
- The Air physical property is listed under Selection in Physical Property Table Manager window. Follow the same steps to create the other physical properties that are required (i.e. Virtual air region, Coil material, Core material and Magnet material).
3.2. Mesh Mating

For an EM analysis, a conformed mesh is needed. For 3D problems, Mesh Mating should be always applied before starting mesh.

- Click Home | Connection | Mesh Mating.

- Figure 25: Mesh Mating command

- Change the selection filter to Polygon Body, use Ctrl+A select all bodies as Selection in the Mesh Mating Condition window, making sure that the mesh mating type is Glue-Coincident. Keep all other settings and click OK.

- Figure 26: Settings for Mesh Mating Condition

- After finishing the process, 390 Glue Coincident MMC are created.

- Figure 27: Mesh Mating Condition information

- To check if all the MMCs are applied properly, click Find Exterior Faces command from the Low Frequency EM ribbon. This will group all the exterior faces under Groups in the Simulation Navigator. Click Exterior Faces group; the view window will update to show all the exterior faces. If there are no interior faces, this means that all MMCs have been applied properly.

- Figure 28: Find Exterior Faces command

- Click File | Save | Save all to save the files.
3.3. Generate Mesh

The Group Similar Geometry tool, found in the Low Frequency EM tab, will be useful for the bodies’ selection. Click it to group the similar geometries under Groups in Simulation Navigator.

In the 3D Tetrahedral Mesh window, the following selections (highlighted below) are required settings for EM models:

- **For 3D case**, the only available mesh type is **Linear Tetrahedron**.
- **Internal Mesh Gradation** should be always set to 1
- **Small Feature Tolerance** should be always set to 0
Due to Simcenter 3D’s meshing paradigm, the order of the mesh assignments is important. Where faces/edges meet, it is recommended that you start the mesh from the one that has the smaller mesh element size.

- Select Body Group #6 (note: your # may differ – ensure that you’ve selected the magnets) and click 3D Tetrahedral from Home | Mesh tab.

- Set Element Size=1 mm and keep all the other settings. Create a mesh collector “Magnets” with PM as the Solid Property.

- Click OK. After finishing the mesh processing, the Magnets collector is listed under 3D Collectors in the Simulation Navigator.
Select Body Group #3 and click 3D Tetrahedral. Set Element Size = 1 mm and keep all other settings. Create a mesh collector “VA air” with VA as Solid Property.

Figure 34: 3D Mesh Settings for VA air

Click OK. After finishing the mesh processing, the VA air collector is listed under 3D Collectors in the Simulation Navigator.

Figure 35: VA air mesh Collector

Select Body Group #1 and click 3D Tetrahedral. Set Element Size = 1.26 mm and keep all other settings. Create a mesh collector “Remesh” with Air as Solid Property.

Figure 36: 3D Mesh Settings for Remesh
Click **OK**. After finishing the mesh processing, the Remesh collector is listed under **3D Collectors** in Simulation Navigator.

Double-clicking **Body Group #4** will display this group and hide all of the others.

Click **3D Tetrahedral**. Use Ctrl+A to select all of the coil bodies. Set **Element Size**=2.69 mm and keep all other settings. Create a mesh collector “**Coils**” with Copper as **Solid Property**.

Click **OK**. After finishing the mesh processing, the **Coils** collector is listed under **3D Collectors** in Simulation Navigator.
Double-clicking Body Group #5 will display this group and hide all of the others (note: your # may differ – ensure that you’ve selected the two outer stator bodies). Since the stator teeth are also critical regions, a 2D seed mesh will be used here for local mesh refinement.

Click 2D Mesh. Select all two bodies. Set Element Size=1.26 mm and keep all other settings. Create a mesh collector “Tooth Seed”.

Note: In 3D analysis, the 2D mesh is only used as a seed for local mesh refinement or for periodic matching boundary. It will not be exported to solver; therefore, it does not require a physical property.

Click OK. After finishing the mesh processing, the Tooth Seed collector is listed under 2D Collectors in Simulation Navigator. Note that under the Status column, the text Not exported to solver is shown. Uncheck Tooth Seed to make it invisible.

Click 3D Tetrahedral and select the two outer stators. Set Element Size=2.96 mm and keep all other settings. Create a mesh collector “Outer Stator” with CoreSteel as Solid Property.
Click **OK**. After finishing the mesh processing, the **Outer Stator** collector is listed under **3D Collectors** in the **Simulation Navigator**. Note that the tooth surfaces have a smaller element size due to the 2D seed mesh.

![Figure 43: Outer Stator mesh collector](image)

Double-clicking **Body Group #7** *(note: your # may differ – ensure that you’ve selected the inner stator bodies)* will display this group and hide all of the others.

Click **3D Tetrahedral**. Select the inner stator bodies. Set **Element Size**=2.96 mm and keep all other settings. Create a mesh collector “**Inner Stator**” with **CoreSteel** as **Solid Property**.

![Figure 44: 3D Mesh Settings for Inner Stator](image)

Click **OK**. After finishing the mesh processing, the **Inner Stator** collector is listed under **3D Collectors** in the **Simulation Navigator**.

![Figure 45: Inner Stator mesh collector](image)
Uncheck Polygon Geometry and check it again, making sure that all polygon bodies are shown in view window.

Click 3D Tetrahedral. Select the air enclosure body. Set Element Size=12.6 mm and keep all other settings. Create a mesh collector “Air box” with Air as Solid Property.

Click OK. After finishing the mesh processing, the Air box collector is listed under 3D Collectors in the Simulation Navigator.

Clicking Home | Utilities | More and selecting Model Display Preferences provides a better view of the mesh. In Model Display window, under Element page, select Mesh Collector for Color Basis and click Set Mesh Colors button.

Click File | Save | Save all to save the files.
3.4. Define Magnetization direction

In this model, there are six magnets, whose direction will be defined using the pattern feature.

- Check **Polygon Geometry** to make all polygon bodies visible. Uncheck all other 3D mesh collectors except for the **Magnets** collector, making the magnets the only visible components.

![Figure 49: Make Magnets mesh and polygon geometry visible](image)

- From the Low Frequency EM menu ribbon, click **Assign Magnets**.

![Figure 50: Assign Magnets command](image)

- From the **Assign Magnets Orientation** window, select the mesh from the view window, as shown below. Select **–XC** as Specify Vector, set pattern type to **Circular**, using the **Point Dialog** method to set origin as **Specify Point**, Select **Count and Span** for **Spacing** method, then set **Count** to 6, **Span** to 360, and check the option **Alternate Orientation**.

![Figure 51: Settings for assigning direction](image)
Click **Preview Pattern** to check the direction assignment. Click **OK** to finish the assignment.

![Preview Pattern](image)

*Figure 52: Preview the pattern direction*

Click **File | Save | Save all** to save the files.
4. Simulation Set-Up

The simulation set-up involves creating coils, circuits, motion objects and assigning boundary conditions, which can only be done in the sim file.

Right-click “Linear Actuator Tutorial_fem1.fem” in Simulation Navigator, select Display Simulation and then “Linear Actuator Tutorial_sim1.sim”. This will make the sim the work part.

4.1. Create Coils

Coils are implemented as simulation objects and are associated to a solution. Before creating coils, at least one solution should be created.

Right-click “Linear Actuator Tutorial_sim1.sim” in the Simulation Navigator, select New Solution.

In the Solution window, set Solution Type to Transient. Set Stop Time=40 s and TimeStep=1 s. In addition, select First order for Time Step Order.

Click OK. The new solution will be listed in the Simulation Navigator

Note: For motion analysis, the time step is, by default, Second Order, which takes into account transient effects such as eddy currents. In this demo, the purpose is to study the static thrust force, so First Order is ok and it can save considerable solving time.
To obtain a better view, double-click “Body Group #4..” (note: your # may differ – ensure that you’ve selected the Coils) from Groups in the Simulation Navigator to make the polygon bodies of the Coils the only visible components.

Click Create Coil on the Low Frequency EM ribbon.

In the Coil window, select Body Coil method. Then set the name as Coil 1. Select the coil body shown in the image as Select Object. An arrow will show the current flow direction. For this case, keep the direction as shown. Then modify the following settings:

- In Coil Type section, select Stranded as Type, set Number of Turns=350.
- In Pattern Definition section, select Circular as Type, set ZC as Specify Vector and set (0,0,0) as Specify Point. Select Count and Span as Spacing method, set Count=6 and Span=360. Click Preview Pattern to show all coils.
Click **Apply** to finish the coil definition and keep the window open with the same settings. Change the name to **Coil 2**, and select the coil body shown in the image below as **Select Object**. Click the Reverse Direction button in Specify Vector to change the current flow direction, keep all other settings, and then click Preview Pattern button to show all coils.

![Figure 59: Settings for group Coil2 creation](image)

Click **OK** to finish the coils definition. In the Simulation Navigator, two folders will be listed under Simulation Object Container. To display the coils, click Display Coils on the Low Frequency EM ribbon.

**Note:** the coils are active and automatically added to the “active” solution that was created earlier.

![Figure 60: Coil-Pattern folder](image)

The coils need to be wound in alternating directions, along the circumferential direction. This can be done in two ways by either making the proper connection in a circuit or editing the coil without creating a circuit. For this example, we will use the second method.

Right-click **Coil 1(2)** and select **Edit**. Click the Reverse Direction button in Body Coil window. Then click **OK**.

![Figure 61: Edit coil direction](image)

Follow the same steps to reverse the direction for **Coil 1(4)**, **Coil 1(6)**, **Coil 2(2)**, **Coil 2(4)**, and **Coil 2(6)**.

Click **File | Save | Save all** to save the files.
4.2. Create excitation

The excitation can be set to Coils directly in the Body Coil window, instead of in a circuit. In this demo, the excitation is a DC source with 1 A for all the coils. So creating one current source excitation will be enough.

❖ Right-click Modeling Objects and select Modeling Objects.

❖ In the Modeling Objects Manager window, select Coil Excitation-Current from the Type drop-down list. Set Name as DC-1A and click Create. In the Coil Excitation-Current window, change the type to DC and set 1 for the value. Click OK. Selection table in Modeling Objects Manager window list the excitations. Click OK to finish the current excitation creation.

❖ Right-click Coil 1 and select Edit. In the Body Coil window, select Current Driven from the Type drop-down list in the Excitation section. Then, select DC-1A for Coil Excitation.

❖ Follow the same steps to set all the other coils.

❖ Click File|Save|Save all to save the files.
4.3. Setup Motion component

The motion component is made up of the six magnets and the virtual air box enclosing the magnets.

- Select **Body Group #3 and #6** *(note: your # may differ – ensure that you’ve selected the six magnets and the virtual air box)* from Groups in the Simulation Navigator.

![Group selection for Motion Component](image)

- Click **Motion Component** from Home | Loads and Conditions tab. The Motion Component window appears.

![Motion Component command](image)

- Select **Linear Component** type at the top cell. Body group #3 and #6 are automatically added as **Select Object**. Set **ZC** as **Specify Vector** and **Source Type** to **Velocity Driven**.

![Setup motion component](image)

- In the **Position tab**, select **Position Based** type, and click **Table Constructor** button from the dropdown list beside **Specify Field**.

![Setup source type](image)
In the Table Field window, click Import from Text File button in the Data Points section. In the Table Data Import window, navigate to Start model folder and select Position.txt file. Click OK twice to get back to the Table Field window.

![Figure 69: Import table field data](image)

The data are imported and shown in the data points table. Click OK to go back to the Motion Component window. Note that a green mark is shown beside Position. Click OK to finish the creation of the motion component.

![Figure 70: Finish Data point import for Position](image)

The newly created Motion component is now listed under Simulation Object Container in the Simulation Navigator.

![Figure 71: Motion component in Simulation object container](image)

Click File | Save | Save all to save the files.
4.4. Setup Geometric Parameterization

In this study, the effect on the stroke and thrust force, due to varied length of permanent magnet, will be investigated. To parameterize the length of the permanent magnet, the synchronous modeling technique, with interpart expressions, is used.

- **Use Ctrl+E to open the Expressions window. Create a length expression Delta with initial value of 1.5mm.**

- **Right-click Solution 1 and select Edit. In the Solution window, change the solution Name to Solution 1-PM Length Parameterization. In the Parameterization tab, set Parameterization Method=Single Parameter and click Create Modeling Object button beside Parameterization Attribute cell.**

  **Note:** Currently, only single parameterization is available.

- **In the Parameterization Attribute window, select Delta from Expression drop-down list and type -0.5, 1.5 and 3.5 as shown below. Check the option Geometric Parameter and click OK twice to finish the solution settings. Three Steps are shown under solution.**

  ![Figure 77: Parameterization Attribute](image-url)
Double-click "Linear Actuator Tutorial_fem1_i.prt" to make it Displayed & Work part. Click Part Navigator button to switch the Simulation Navigator to the Part Navigator.

Right-click "Linked Body (5)" and select Make Current Feature. Then, click Move Face from Home|Synchronous Modeling tab.

In the Move Face window, select the back faces of the 6 magnets, set Motion to Distance and Specify Vector to –ZC axis.
From the drop-down list in Distance cell, select Reference and then Interpart Expression.

In the Create Single Interpart Expression window, select Linear Actuator Tutorial_sim.sim file. All the expressions in the sim will be listed in Source Expression table. Select Delta from the table and click OK.
In Distance cell, **Delta** is set. Click **OK** and **Move Face (6)** will be inserted in **Model History**.

![Figure 80: Move Face feature in Model History](image)

Follow the same steps, selecting the front six faces of the six magnets, and move them in ZC axis direction with **Delta** as distance. Two new features, **Move Face (6)** and **Move Face (7)**, are listed under **Model History**.

![Figure 81: Create another move face feature](image)

- Right-click “**Subtract (16)**” and select **Make Current Feature**.
- Click **File** | **Save** | **Save all** to save the files.
4.5. Solve

- Click Pre/Post from Application | Simulation tab to exit the modeling mode.
- Double-click “Linear Actuator Tutorial_sim” in Simulation File View to make it the Displayed & Work part.

![Figure 82: Make sim Displayed & Work part](image)

- Right-click the active solution Solution 1-PM Length Parameterization, and select Solve. In the window, click OK to start the solve. Note: clicking Edit Solver Parameters and unchecking the option Run Solver in Foreground will allow users to continue working on the sim during the solving process.

![Figure 83: Launch active solution solve](image)

- Since there are three steps in the solution, SC will update the fem based on the Delta value. This will take a while and when starting a solve, a solver monitor will pop up to show the solving progress. Note that the number of problems solving is 3.

![Figure 84: Solver monitor](image)
5. Results
The post-processing of results is available through a separate tool (i.e. Low Frequency EM - Results Viewer) that can be launched directly from Simcenter.

☐ Click Open Results Viewer from the Low Frequency EM ribbon. This will open the results of the active solution.

![Launch Results Viewer from Low Frequency EM tab](image)

5.1. B field Result

☐ From the Simulation Navigator, double-click “Solution 1-Parameterization PM Length” to make it the active solution, then click Open Results Viewer from the Low Frequency EM ribbon. The results viewer will popup. In TreeView, the default datasets are listed: Solution mesh and the B field.

![TreeView in Low Frequency EM Results viewer](image)

☐ Right-click Problem filter under B field, and select Properties. In the properties window, check Include for Selection mode, then from the Selected problem drop-down list, select Step2.

☐ Click OK.

![Edit Problem filter](image)
Right-click Component filter under B field, and select Properties. In the properties window, keep Exclude for Selection mode, then from the Selected components drop-down list, uncheck VA/3d, Remesh/3d and Air box/3d meshes.

Click OK.

Figure 88: Edit Component filter

Right-click Time instant filter under B field, and select Properties. In the properties window, select 2000 ms for Time instant Value, then click OK.

Figure 89: Edit Time instant filter

Checking Vector magnitude will display the B field plot in the View window. To obtain the below view, right-click the Field View and select 3D Views/Isometric.

Figure 90: Show B field plot
Right-click Time instant filter under B field, and select Properties. In the properties window, select 4000 ms for Time instant Value, then click OK. The view will update the B field.

5.2. Thrust force Results

Right-click Results in TreeView. Select New Quantity and then Motion magnetic force.

Check Motion magnetic force and Chart view to show the force plots for the three steps. To obtain the below view, right-click the Chart View and select Show Curve Name to remove the names.

Note: The results show that the three cases have almost the same stable force from 1000ms to 3000 ms, which covers the design stroke -10mm to 10 mm. The red plot design has the shortest material length but the stable force range is almost exact from 1000 ms to 3000 ms. Outside this range, the force changes significantly; so, there is no safe margin. Even the third design (blue) has a large stable force range, but it exceeds the design requirement a little too much. Considering the material cost and the reliability, the second design will be the best one.