Simcenter 3D Low Frequency EM
Tutorial #3

Linear Actuator

Version: Simcenter 3D 2020.1
# 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

Figure 1: Linear Actuator tutorial model in Simcenter 3D 2020.1 ......................................................... 1
Figure 2: Actuator 3D assembly ................................................................................................................. 2
Figure 3: Create new FEM and Simulation .................................................................................................. 2
Figure 4: Setup for new FEM and Simulation ............................................................................................... 3
Figure 5: Wave link all the bodies .............................................................................................................. 3
Figure 6: Part Navigator ............................................................................................................................ 4
Figure 7: Modeling mode ............................................................................................................................ 4
Figure 8: Sketch Command ....................................................................................................................... 4
Figure 9: Extrude for virtual air box creation .............................................................................................. 4
Figure 10: Edit object display ................................................................................................................... 5
Figure 11: Boolean Subtract ....................................................................................................................... 5
Figure 12: Settings for Subtract Boolean operation .................................................................................... 5
Figure 13: Extrude remesh region .............................................................................................................. 6
Figure 14: Edit object display for remesh region ....................................................................................... 6
Figure 15: Boolean Subtraction for remesh region .................................................................................... 6
Figure 16: Extrude air region ...................................................................................................................... 7
Figure 17: Edit object display for air region ................................................................................................ 7
Figure 18: Boolean Subtract for air region ................................................................................................ 7
Figure 19: Pre/Post command .................................................................................................................. 8
Figure 20: Make fem the work part ........................................................................................................... 8
Figure 21: Edit FEM settings ..................................................................................................................... 8
Figure 22: Physical Properties .................................................................................................................... 9
Figure 23: Create physical property table .................................................................................................. 9
Figure 24: Physical Property table ........................................................................................................... 9
Figure 25: Mesh Mating command ........................................................................................................... 10
Figure 26: Settings for Mesh Mating Condition ......................................................................................... 10
Figure 27: Mesh Mating Condition information ......................................................................................... 10
Figure 28: Find Exterior Faces command ................................................................................................ 10
Figure 29: 3D Tetrahedral command .......................................................................................................... 11
Figure 30: 3D Tetrahedral Mesh Settings ................................................................................................ 11
Figure 31: Group selection ....................................................................................................................... 12
Figure 32: 3D mesh settings for magnets .................................................................................................. 12
Figure 33: Magnets mesh collector ........................................................................................................... 12
Figure 34: 3D Mesh Settings for VA air .................................................................................................... 13
Figure 35: VA air mesh Collector ................................................................................................................ 13
Figure 36: 3D Mesh Settings for Remesh ................................................................................................ 13
Figure 37: Remesh mesh collector ............................................................................................................ 14
Figure 38: 3D Mesh Settings for Coils ...................................................................................................... 14
Figure 39: Coils mesh collector .................................................................................................................. 14
Figure 40: 2D seed mesh Settings for stator tooth ..................................................................................... 15
Figure 41: 2D Seed Mesh Collector .......................................................................................................... 15
Figure 42: 3D Mesh Settings for Outer Stator ......................................................................................... 15
Figure 43: Outer Stator mesh collector ..................................................................................................... 16
Figure 44: 3D Mesh Settings for Inner Stator .......................................................................................... 16
Figure 45: Inner Stator mesh collector .................................................................................................... 16
Figure 46: 3D Mesh Settings for Air box ................................................................................................. 17
Figure 47: Air box mesh collector ............................................................................................................ 17
Figure 48: Set mesh collector color .......................................................................................................... 17
Figure 49: Make Magnets mesh and polygon geometry visible ........................................ 18
Figure 50: Assign Magnets command ........................................................................... 18
Figure 51: Settings for assigning direction .................................................................... 18
Figure 52: Preview the pattern direction ....................................................................... 19
Figure 53: Make sim the work part .............................................................................. 20
Figure 54: create a new solution ............................................................................... 20
Figure 55: Create Transient solution ........................................................................... 20
Figure 56: Make Coil bodies visible only ....................................................................... 21
Figure 57: Launch Create Coil tool ............................................................................ 21
Figure 58: Settings for group Coil1 creation .................................................................. 21
Figure 59: Settings for group Coil2 creation .................................................................. 22
Figure 60: Coil-Pattern folder .................................................................................... 22
Figure 61: Edit coil direction ....................................................................................... 22
Figure 62: Create an excitation from Modeling Objects .............................................. 23
Figure 63: Modeling Objects Manager ..................................................................... 23
Figure 64: Assign current excitation to coil ................................................................. 23
Figure 65: Group selection for Motion Component ....................................................... 24
Figure 66: Motion Component command .................................................................... 24
Figure 67: Setup motion component ......................................................................... 24
Figure 68: Setup source type ...................................................................................... 24
Figure 69: Import table field data .............................................................................. 25
Figure 70: Finish Data point import for Position ......................................................... 25
Figure 71: Motion component in Simulation object container ...................................... 25
Figure 72: Create an expression ............................................................................... 26
Figure 73: Setup parameterization ............................................................................ 26
Figure 74: Edit Parameterization Attributes ............................................................... 26
Figure 75: Switch to Part Navigator in ideal part ............................................................ 27
Figure 76: Move Face command ............................................................................... 27
Figure 77: Settings for move face .............................................................................. 27
Figure 78: Use Interpart Expression ......................................................................... 28
Figure 79: Select Interpart Expression ...................................................................... 28
Figure 80: Move Face feature in Model History .......................................................... 29
Figure 81: Create another move face feature ............................................................... 29
Figure 82: Make sim Displayed & Work part ............................................................... 30
Figure 83: Launch active solution solve ................................................................. 30
Figure 84: Solver monitor ......................................................................................... 30
Figure 85: Launch Results Viewer from Low Frequency EM tab ................................ 31
Figure 86: Datasets and Solutions in Low Frequency EM Results viewer .................. 31
Figure 87: Edit Time instant filter .......................................................................... 31
Figure 88: Edit Problem filter .................................................................................. 32
Figure 89: Edit Object filter ..................................................................................... 32
Figure 90: Show B field plot ..................................................................................... 32
Figure 91: B field plot at different time instant ............................................................ 33
Figure 92: Add new quantity-Force ........................................................................... 33
Figure 93: Force plots for three designs ................................................................. 33
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.

![Linear Actuator tutorial model in Simcenter 3D 2020.1](image)

*Figure 1: Linear Actuator tutorial model in Simcenter 3D 2020.1*

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 Finish to complete the sketch.

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 = -7 mm, and End Distance = 62 mm.

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|Object tab from the menu ribbon. Set Translucency in the Edit Object Display window for a better view.

![Edit Object Display](image)

**Figure 10: Edit object display**

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.

![Boolean Subtract](image)

**Figure 11: Boolean Subtract**

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.

![Settings for Subtract Boolean operation](image)

**Figure 12: Settings for Subtract Boolean operation**

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. Remember to click Finish to complete the sketch.
- 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= -8mm and End Distance= 103 mm, and then click OK.

![Extrude remesh region](image1)

*Figure 13: Extrude remesh region*

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

![Edit object display for remesh region](image2)

*Figure 14: Edit object display for remesh region*

- 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 (4) as the Tool body. Ignore the alter message. Check the option Keep Tool and click OK.

![Boolean Subtraction for remesh region](image3)

*Figure 15: Boolean Subtraction for remesh region*

- 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 100 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.

- 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.

- 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.
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, select Solid from the Type drop down list.
- 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.

- 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.

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

- 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.

- 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.

![Figure 29: 3D Tetrahedral command](image)

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.

![Figure 30: 3D Tetrahedral Mesh Settings](image)
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.

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

Select Body Group #1 and click 3D Tetrahedral. Set Element Size=1.25 mm and keep all other settings. Create a mesh collector “Remesh” with Air as Solid Property.
Click OK. After finishing the mesh processing, the Remesh collector is listed under 3D Collectors in Simulation Navigator.

Double-clicking Body Group #5 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=2.5 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 #4** 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 the faces highlighted in the image below. Set **Element Size=1.25 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.

![Figure 40: 2D seed mesh Settings for stator tooth](image)

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.

![Figure 41: 2D Seed Mesh Collector](image)

Click **3D Tetrahedral** and select the two outer stators. Set **Element Size=3 mm** and keep all other settings. Create a mesh collector **“Outer Stator”** with **CoreSteel** as **Solid Property**.

![Figure 42: 3D Mesh Settings for Outer Stator](image)
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=3 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=12.5 mm and keep all other settings. Create a mesh collector “Air box” with Air as Solid Property.

Click 3D Tetrahedral. Select the air enclosure body. Set Element Size=12.5 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 **M 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 Time Step=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 #5..” (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

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

- 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**.

- In the **Position** tab, select **Position Based** type, and click **Table Constructor** button from the drop-down list beside **Specify Field**.
Click Next to accept all defaults until the Definition table appears.

In the Table Field/Definition window, click Import from Text File.

In Table Data Import window, navigate to Start model folder and select Position.txt.

Click OK twice to get back to the Table Field window.

The data are imported and shown in the Table Field/Definition 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.

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

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.

![Figure 72: Create an expression](image)

- 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.

![Figure 73: Setup parameterization](image)

- 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 74: Edit Parameterization Attributes](image)
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 (4)" and select Make Current Feature. Then, click Move 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 (5) will be inserted in Model History.

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 (5) and Move Face (6), are listed under Model History.

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.

![Simulation File View](image)

*Figure 82: Make sim Displayed & Work part*

- 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.

![Solver Options](image)

*Figure 83: Launch active solution solve*

- 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.

![Solver Monitor](image)

*Figure 84: Solver monitor*
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.

5.1. B field Result

Note In version 2020.1 of the Results Viewer:
- TreeView panel has been renamed Datasets and Solutions.
- Component has been renamed Object.
  Functionality has not changed.

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 Datasets and Solutions, the default datasets are listed: Solution mesh and the B field.

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.
- 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**.

![Figure 88: Edit Problem filter](image1)

- Right-click **Object** filter under **B field**, and select **Properties**. In the properties window, keep **Exclude** for **Selection mode**, then from the **Selected objects** drop-down list, uncheck **VA/3d, Remesh/3d** and **Air box/3d** meshes.

- Click **OK**.

![Figure 89: Edit Object filter](image2)

- Checking **Object** 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](image3)
Right-click Time instant filter under B field, and select Properties. In the properties window, select 10000 ms for Time instant Value, then click OK. The view will update the B field.

Figure 91: B field plot at different time instant

5.2. Thrust force Results

Right-click Motion magnetic 3D in Datasets and Solutions. Select New Quantity and then Motion magnetic force.

Figure 92: Add new quantity-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.

Figure 93: Force plots for three designs