Electric Machines Tutorials for NX-Magnetics

Dr. Binde Ingenieure

January 2, 2024

2011-2025 Dr. Binde Ingenieure, Design & Engineering GmbH. All Rights Reserved. This software and related documentation are proprietary to Dr. Binde Ingenieure, Design & Engineering GmbH. All other trademarks are the property of their respective owners.

DR. BINDE INGENIEURE, DESIGN & ENGINEERING GMBH MAKES NO WARRANTY WHATSOEVER, EXPRESSED OR IMPLIED THAT THE PROGRAM AND ITS DOCUMEN-TATION ARE FREE FROM ERRORS AND DEFECTS. IN NO EVENT SHALL DR. BINDE INGENIEURE, DESIGN & ENGINEERING GMBH BECOME LIABLE TO THE USER OR ANY PARTY FOR ANY LOSS, INCLUDING BUT NOT LIMITED TO, LOSS OF TIME, MONEY OR GOODWILL, WHICH MAY ARISE FROM THE USE OF THE PROGRAM AND ITS DOCUMENTATION.

THIS SIMULATION SOFTWARE USES FINITE ELEMENT METHODS. USERS SHOULD BE AWARE THAT RESULTS CAN HAVE UNPREDICTABLE ERRORS IF INPUT DATA IS NOT COMPLETELY CORRECT. THEREFORE ANY DESIGN DECISIONS SHOULD NOT BE BASED SOLELY ON THE SIMULATION. USE ADDITIONAL MEASUREMENTS TO ENSURE THE CORRECTNESS.

Contents

1	Techniques for Motion	4
2	2.1.3 3D Enforced Driver	6 7 14 18 26
3		28 28
4	4.1 Permanentmagnet Synchron Motor Analysis	40 40 49 51 53 55
5		64 64
6	6.1Asynchron (Induction) Motor Analysis6.1.1Basic Model Setup6.1.2Simulation at Fixed Speed6.1.3Sweep over Rotor Speed6.1.4Sweep with two Parameters6.1.5Thermal Analysis6.1.6Transient Run at Fixed Operating Point6.1.7Start of Motor6.1.83D Model of Induction Motor	68 68 69 76 83 85 87 91 97 99 02
7	7.1 Motor (Team30) with Moving Band/Constant Rotor Shape (Conductor Motion) 10 7.1.1 Simulation with Moving Band 10 7.1.2 Post Processing 11 7.1.3 Solution with Conductor Motion 11 7.1.4 Post Processing 11 7.1.4 Post Processing 11	03 03 08 10 11 12 12

8	Futorial 7	119
	3.1 Motor Analysis in 3D by Sliding Motion	119
	8.1.1 Properties for Sliding Surface	120
	8.1.2 Properties for Periodicity Segments	121
	8.1.3 The Rotor Joint Definition	122
	8.1.4 Recommendations for Meshing	124
	8.1.5 Solving and Post Processing	125
9	Futorial 8 0.1 Motor Analysis in 3D by General Motion	127 127
10	Futorial 9	134
	10.1 Electric Motor Analysis for NVH	134
11	References	150

1 Techniques for Motion

The MAGNETICS solver has five different types of motion techniques:

- 'General Motion (GM)': General Motion can be used for any kind of motion, e.g. translation or rotation in 3D, because a remeshing is performed for every step. The process is the following:
 - 1. An initial start solution is solved. The basic result 'Magnetic Vector Potential' is stored in an extra file (restart-file). Following this solution is referenced as step 0.
 - 2. The nodes and elements of the moving body are then moved, e.g. rotated or translated corresponding to the desired displacement step. In 3D scenarios, the air mesh is recreated using function 'Solid from Shell'. This function needs a 2D boundary mesh which typically goes over all parts and also over the boundary of the air volume. Therefore, we don't have 'Mesh Mating Conditions' between air and the parts. In 2D scenarios said 'Solid from Shell' meshing is superfluous, of course.
 - 3. In case of magneto dynamic solutions:

To capture dynamic effects (the time derivative) it is necessary for every solution time step (e.g. step 1) to include results from one time step prior (e.g. step 0), in order to form time derivatives. Therefore, each solution step uses the result of the prior solution (e.g. step 0) as start condition. This is done automatically inside the solver.

4. In case of magneto static solutions:

No time derivative is necessary. Therefore, each time step is a simple magneto static analysis step.

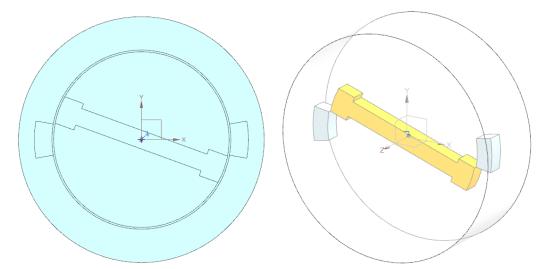
- 5. One time step is computed. Results can be as usual: Magnetic Flux Density, Force, Eddy Currents, Current Losses, Temperature and others. In case of dynamic solutions finally the Magnetic Vector Potential result is again stored in the restart file and referenced as step 0.
- 6. These steps are repeated in a loop to perform the complete movement.
- 'Moving Band (MB)': This is capable for 2D rotation only. The process is the following:
 - 1. The user defines the outside and inside edge of the air gap. The system creates a tri mesh between them and updates this at each step.
 - 2. The nodes and elements of the moving body are then moved, e.g. rotated corresponding to the desired displacement step.
 - 3. These steps are repeated in a loop to perform the complete movement.
- 'Sliding Surface (SM)': Sliding Surface is very similar to 'Moving Band' but it is made for 3D rotation. Here the air gap is made of two surfaces sliding in circular way. The mesh between the two surfaces must be structured such that, if a movement step appears, all nodes can find a new partner node again.
- 'Conductor Motion, const Shape (CM)'. This feature models the effect of moving conductors, such as water. It can also be used for motors, if the rotor shape does not change while rotating. So, there must be no change in geometry to use this feature. This is the case for instance if water flows through a tube or some induction motor types.

• 'Frequency Motion (FM)': This feature is used for the analysis of induction motors. The principle works similar to the 'Conductor Motion, const Shape'.

2 Tutorial 1

2.1 Simple Rotor

In this tutorial we analyse a rotor that is moving between two permanent magnets. First we apply a forced speed of 5.000 turns per minute and then we analyse the free motion behaviour.



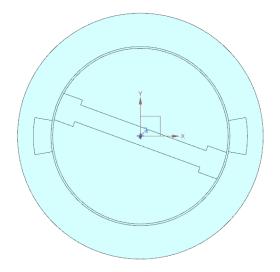
Different techniques are shown here, so we use 2D as well as 3D modeling. And we use two different motion techniques for rotation: 'General Motion', a feature that can be used for 3D rotation and translation. And the 'Moving Band' technique, that is capable for 2D rotation only. Also we show free (dynamic) motion as well as enforced motion. Thus, the tutorial is split into four parts:

- 1. 2D enforced,
- 2. 2D dynamic,
- 3. 3D enforced,
- 4. 3D dynamic.

Two permanent magnets are positioned near the rotor. This leads to time dependent variations of the magnetic field and therefore eddy currents will appear in all electric conducting parts. In reality such eddy losses will lead to temperature rise, an effect that is neglected in this basic tutorial but could be simulated by a coupled thermal solution. Those eddy currents shall be displayed and resulting power loss shall be computed. The enforced analysis shall be done over 45 degrees, the dynamic one over a time period that allows observing the expected oscillations. The CAD models of both the 2D and 3D base on the same 'Skelett.prt' file.

2.1.1 2D Enforced Driver

The following example a pure 2D example; and thus not designed to work in 3D. In this example, an iron rotor rotates between two permanent magnets. The example shows how to perform analysis' that couple electromagnetics and mechanical movement. In the present scenario the motion is simulated with an enforced motion; however, it can also be simulated with dynamic motion. This is done in the next tutorial. Then, the movement would have one degree of freedom in the mechanical sense. The example is executed in 2D because of the shorter solution time but it would be similar in 3D as is shown in a further example.



Main goal is to model the enforced motion effect of the rotor. Other results like eddy currents can be requested additionally if desired.

Estimated time: 45 min.

Follow the steps to reproduce it:

- Download the model files for this tutorial from the following link: https://www.magnetics.de/downloads/Tutorials/6.CouplMotion/6.2SimpleRotor3D. zip
- 2. Open the part file 'rotor2D.prt'.
 - Here, the start position of the rotor is set to -20 degrees.
 - Also, there is a 'Moving Band' visible. This has two circular edges, one connected to the moving part and one to the stator regions. At each time step the solver will rotate the moving regions and create a mesh between the two edges automatically.
- 3. Start Simcenter Pre/Post, create a new Fem and Sim file for 2D Electromagnetics.
 - Switch off 'Create Idealized File',
 - Choose Solver 'MAGNETICS' and Analysis Type '2D or axisym Electromagnetics', OK,
 - Set the 'Analysis Type' to 'Magnetodynamic Transient' and name the solution 'Enforced'.

• In register 'Output Requests', 'Table' activate 'Motion Data' and in 'Plot' activate 'Displacement' as well as 'Magnetic Fluxdensity'. Others can be activated if one is interested.

Solution				
Solution			^	Plot
Name Solver Analysis Type	Enforced MAGNETICS 2D or axisym Electromagnetics Magnetodynamic Transient		*	Table Total Force - virtual Total Moment - virtual Total Lorentz Force
Solution Type Magnetodyna Magnetodynamic Transient			• •	RotorBand Torque - stresstensor RotorBand Force - stresstensor Fluxlinkage - Vectorpotential on Conductors
Output Rec Time Steps Initial Cond Coupled Th 2D	ditions	Plot Magnetic Fluxdensity Magnetic Fieldstrength Current Density Eddy Current Losses Density Magnetic Potential (a-Pot) Nodal Force - virtual Lorentz Force Displacement Material Properties		 □ Electrode Voltage □ Electrode Current □ Electrode Power □ Circuit Voltage □ Circuit Current □ Circuit Power ☑ Eddy Current Losses □ Hysteresis Loss - steinmetz □ Eddy Current Loss - steinmetz □ FFT Tables □ Motor Efficiency
		Table	N	NVH Coupling

• In register 'Time Steps' set the 'Time Increment' to $1/(60^*360/60)$. This formula corresponds to a rotor speed of 60 rpm (1 rps).

Output Requests	Time Step Option	Constant		
Time Steps	Time Increment	1/(60*360/60) s -		
 Initial Conditions 				
- Coupled Thermal	End Time Option	Number of Time Steps		

• In register '2D' set the 'Thickness' to 10 mm.

Output Requests Plane	Plane	Absolute X,Y			
Time Steps	Axisymme	Axisymmetric			
Initial Conditions	Thickness	10			
- Coupled Thermal	THICKNESS	10	mm •		
2D					

- OK.
- 4. Switch to the Fem File.
 - When using the Moving Band technique the band elements should have about the same element size. Therefore, create a 'Mesh Control' , use type 'Size on Edge', select the outer and inner edges of the moving band, set the 'Element Size' to 1.5 mm

and click OK.

Mesh Control		ુ ગ x	
😂 Size on Edge		•	
Selection		^	
✓ Select Targets (16)		÷	
Size on Edge		^	
Location on Edge	Overall	•	
Element Size	1.5 mr	n 🔻	
Auto Size		1	

• Create a Tri-Mesh on the Magnets. Use the suggested element size and assign material 'N30EH at 100C'. The permanent direction of north is X by default, so we can stay with the default for our example. Assign the name 'Magnets'.

	Ģ	PlanePhysical		ა x
		Physical Property Table		^
		Name	Magnets	
		Label	1	
YC		Properties		^
xc		Material	N30EH at 100C	-
		Conductor Model		~
		Rigid Body Motion		~
		CSYS		^
		Material CSYS	Absolute	-
		Output Total Moment CSYS	Absolute	•
	-			

• For the Rotor also create a tri mesh with the suggested size and assign the material 'Iron_Sample1'. Also set the 'Inertia RZ' to '87 Kgmm²'. Assign the name 'Rotor_Iron'.

	PlanePhysical	ა x
	Physical Property Table	^
	Name	Rotor_Iron
ус	Label	2
	Properties	^
XC	Material	Iron_Sample1 👻 🔇
	Conductor Model	v
	Rigid Body Motion	^
	Mass	kg 🔹 👻
	Inertia RZ	87 kg·mm² ▼ ▼
	CSYS	v

• For the inner air (2 faces) create a tri mesh with suggested element size and assign a 'FluidPhysical' with material 'Air'. Name it 'innerAir'.

		FluidP	hysical	ს? X
 Mesh Collector Properties Physical Property 	৩ ? × ^	Physical Name	Property Table FluidPhysical1	^
Type PlanePhysical	FluidPhysical None Kone	Label Propertie	4 es	
Name	InnerAir	Material	Air	•

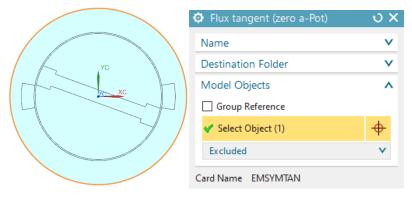
• For the outer air create a tri mesh and assign one more 'FluidPhysical' with material 'Air'. Name it 'outerAir'.

		Fluid	Physical	υx
Mesh CollectorProperties	৩ x ^	Physical Name	Property Table FluidPhysical2	^
Physical Property Type	► FluidPhysical	Label	5	
PlanePhysical	None 🔻 😻 🏷 💌	Properti		^
Name c	outerAir	Material	Air	▼

- Notice: Do NOT mesh the air gap between rotor and stator. When using the MovingBand feature this will be done automatically inside the solver.
- Check that all mesh collectors have a meaningful name. Then, click the button 'Rename Meshes and Physicals from Collectors' from the Magnetics toolbar.



- 5. Switch to the Sim File.
- 6. Assign a constraint of type 'Flux tangent (zero a-Pot)' on the circular edge of the infinity air.



7. Now create a new Simulation Object 'Enforced Motion 2D'. Use the type 'Revolute by Moving Band'. Hint: The type 'Revolute by General Motion' would also work, but this would use a different motion technique.

🖻 📲 Solution 1			
	2	. [
	New Simulation Object	•	💰 Enforced Motion 2D
- 🗸 🗈 Loads	1 Page	•	💰 Dynamic Motion 2D
🗄 🗁 Results		_	🛷 Deactivation Set

• At 'Airgap Rotor Edge' select the 8 inner circle edges as shown below. These must be those edges, that belong to the moving part. Maybe the selection filter with option 'Tangent Continuous Edges' is helpful here.

	Enforced Motion 2D	<u>ა</u>
	鶰 Revolute by Moving Band	-
	Name	v
	Destination Folder	~
	Airgap Rotor Edge	^
	Group Reference	
	< Select Object (8)	\
VC A	Airgap Stator Edge	٨
	Group Reference	
	✤ Select Object (0)	•
	Airgap, auto meshed	^

• At 'Airgap Stator Edge' select the 8 outer circle edges as shown below.

	Airgap Rotor Edge	^
	Group Reference Select Object (8)	•
YC	Airgap Stator Edge	^
	🞸 Select Object (8)	.
	Airgap, auto meshed	^

• Under the box 'Airgap, auto meshed' select from the list the Air, that already resides in the Fem file. Accept the default 1 at 'Number of Air Gaps'.

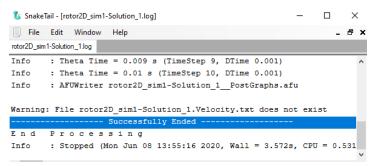
Airgap, auto meshed		~
Number of Air Gaps	1	•
Material	ortor2D_fem1::Air	▼ ⁽ 1)
Rotor Geometry		^
Moving Parts (0)		

• Under 'Rotor Geometry', click 'Create Moving Parts' and select the Rotor and the inner air and add these to the list. Then click 'Close' as shown below. If the names do not appear in the list take care that in the Fem file the Physicals have such

names assigned.

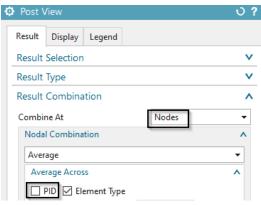
			Physical Property	ty Table M	anager	?	Definition		^
			Create			V	Deminion		
			Filter			v	Rotation Axis	Z	•
Airgap Rotor Edge		v	Selection			^	Stator Shift X0	0 mm •	• •
31 3			Name	Label	Туре		Driver	Angular Step	•
Airgap Stator Edge		V	Rotor	2	PlanePhysical	^	Angular Step	1 • .	• •
Airgap, auto meshed		^	PlanePhysical1	3	PlanePhysical FluidPhysical		Tolerance Factor for Links	1e-05	-
Number of Air Gaps	1	-	A PlanePhysical2	5 6 (B)	PlanePhysical	~			
	O	<u>A</u>		6 43			Periodicity		^
Material	☐ rotor2D_fem1::Air	▼ №3	List			^	Number Poles in Model	1	
Rotor Geometry		^	InnerAir			Ð	Number Poles Total	1	
Moving Parts (2)						Close	Card Name MBEnfRevolute		

- At 'Angular Step' key in 1 deg. Because we have defined 45 time steps we will compute for 45 degrees of motion with these settings. OK.
- 8. Solve the solution. The solution monitor indicates the progress and successful end.



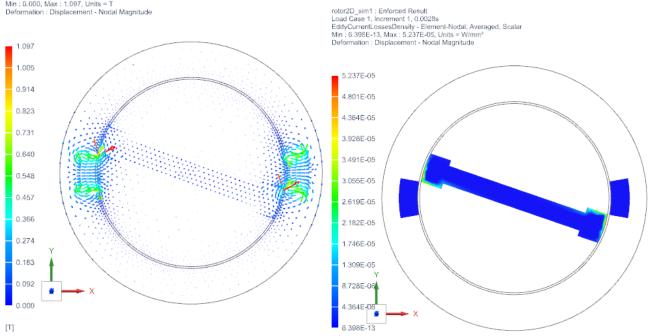
9. Plot the Magnetic Flux Density and observe the result. For a smoother display set the 'Combine At' option at 'Edit Post View' to 'Nodes' and at 'Average Across' deactivate

'Combine At' option at 'Edit Post View' 🚰 to 'Nodes' and at 'Average Across' deactivate 'PID'.

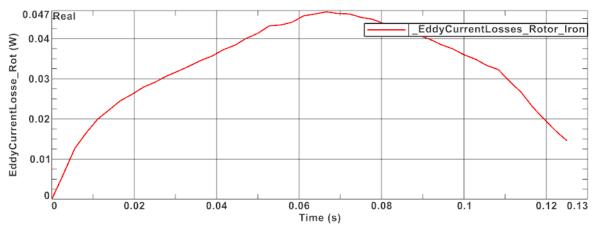


10. The plot with magnetic flux density, arrows and contour, then will look like the following for increment 1.

rotor2D_sim1 : Enforced Result Load Case 1, Initial Time Condition, 0s Magnetic Flux Density - Element-Nodal, Averaged, Magnitude Min : 0.000, Max : 1.097, Units = T Deformation : Displacement - Nodal Magnitude



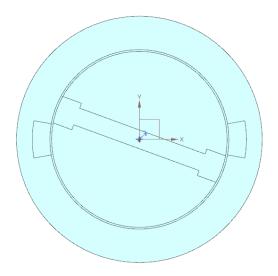
- 11. Cycle through the time steps by using the green button 'Next Iteration' ← →. Alternatively, use the 'Animation' function and set the option 'Animate' to 'Iterations'. If you want to see more movement repeat the solution with more time steps.
- 12. Display the eddy current losses on the rotor. This graph shows nicely the eddy current effect when rotor and magnets move near to each other.



This ends the tutorial.

2.1.2 2D Dynamic Driver

The following example is a pure 2D example; and particularly is intended to be a follow-up example of the previous example 'Simple Rotor, Enforced'. In this example, the rotor oscillates between two permanent magnets. The example shows how to perform an analysis that couples electromagnetics and mechanical movement. In the present scenario the magnet is simulated with a dynamic motion. Here, the movement has one degree of freedom in the mechanical sense. The example is executed in 2D because of the shorter solution time but it would be quite similar in 3D.



Main goal is to find the oscillation behavior of the rotator. Other results like eddy currents can be requested additionally if desired.

Estimated time: 25 min

Follow the steps to reproduce it:

1. Start from the previous example and clone the solution,

Solution 1	<i>a</i> .
🛨 🖌 🐳 Simulation Objects	J Edit
🛨 🖌 🚽 🛨 Constraints	臣 Edit Solver Parameters
🗸 🖈 Loads	📝 Manage Prerequisite Solu
🕂 🗁 Results	

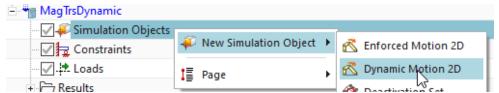
- 2. then remove the enforced joint.
- 3. Name the new Solution 'Dynamic'.
 - Chose Analysis Type 'Magnetodynamic Transient'.
 - In register 'Output Requests', Table activate 'Motion Data' and in Plot activate Displacement as well as 'Magnetic Fluxdensity'. Others can be activated if desired.

Solution			ა x	Plot
Solution			A .	Table
			~	Total Force - virtual
Name	Dynamic			Total Moment - virtual
Solver	MAGNETICS			Total Lorentz Force
Analysis Type	2D or axisym B	Electromagnetics		RotorBand Torque - stresstensor
Solution Type	Magnetodyna	mic Transient		RotorBand Force - stresstensor
				Fluxlinkage - Vectorpotential on Conductors
Magnetodyna	mic Transient		^	Electrode Voltage
Output Req	luests	Plot	^	Electrode Current
Time Steps		Magnetic Fluxdensity		Electrode Power
- Initial Cond		Magnetic Fieldstrength		Circuit Voltage
- Coupled Th	iermal	Current Density		Circuit Current
i 2D		Eddy Current Losses Density		Circuit Power
		Magnetic Potential (a-Pot)		Eddy Current Losses
		Nodal Force - virtual		Hysteresis Loss - steinmetz
		Nodal Moment - virtual		Eddy Current Loss - steinmetz
		Lorentz Force		Excess Loss - steinmetz
		✓ Displacement		Motion Data
		Material Properties		FFT Tables
		Table	v	Motor Efficiency

• Modify the 'Time Increment' to 0.001 and the 'Number of Time Steps' to 100 resulting in a total simulation time of 0.1 sec.

lagnetodynamic Transi	ent			
Output Requests	Time Step Option	Constant		•
Time Steps	Time Increment	0.001	c •	-
Initial Conditions			5 .	_
Coupled Thermal	End Time Option	Number of Time Steps		•
2D	Number of Time Steps	100		-

- 4. Change to the Sim file, if not already there.
 - If a new solution is created, add the constrain 'Flux Tangent' to the solution.
 - Create a 'New Simulation Object' of type 'Dynamic Motion 2D'. Remove the old one, if the solution was cloned.



• Select Type 'Revolute by Moving Band' (the default)

Dynamic Motion 2D		ა? X			
🚯 Revolute by Moving Ba	and	•			
Name		v			
Destination Folder		v	Definition		
Airgap Rotor Edge		^	Rotation Axis Stator Shift X0	Z	mm •
✤ Select Object (0)		↔ …	Tolerance Factor for Links	1e-05	
Airgap Stator Edge		^	Periodicity		
☐ Group Reference ★ Select Object (0)		ф·	Number Poles in Model Number Poles Total	1	
Airgap, auto meshed		^	Initial Conditions		
Number of Air Gaps	1	•	Spring, Damper		
Material	None	-	More		
Rotor Geometry		^	Card Name MBDynRevolute		
Moving Parts (0)				OK Apply	Cano

• Select the 'Airgap Rotor Edge' and the 'Airgap Stator Edge' in the same way as already done in the previous enforced example.

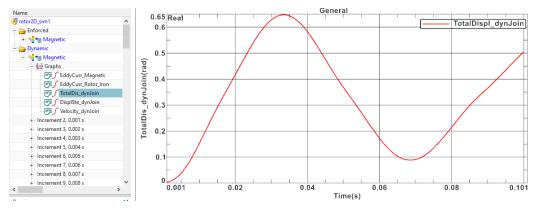
Airgap Rotor Edge	^
☐ Group Reference ✓ Select Object (8)	•
Airgap Stator Edge	^
Select Object (8)	+
Airgap, auto meshed Number of Air Gaps	rotor2D_fem1::Air 👻 🚱

• Again select Air for the material at the Airgap

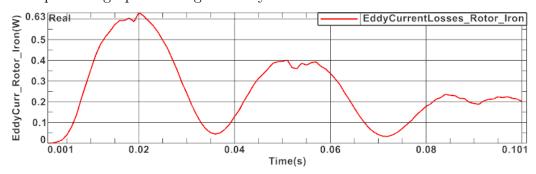
			Selection			^
			Name	Label	Туре	
			Rotor	2	PlanePhysical	^
Airgap, auto meshed		^	PlanePhysical1	3	PlanePhysical	
···· 3-1-			🕞 InnerAir	4	FluidPhysical	
Number of Air Gaps	1	-	PlanePhysical2	5	PlanePhysical	¥
Material	🔒 rotor2D_fem1::Air	-		86	ŧ ■ŧ × 0	<u>a</u>
Rotor Geometry		^	List			^
-			Rotor			Ð
Moving Parts (2)			InnerAir			$\overline{\nabla}$

- At 'Rotor Geometry': Select the Rotor and the inner air.
- All other settings can stay at the defaults. There is no step size necessary because this will be computed from the dynamics of the system: Electromagnetic forces and mass inertia of rotor.

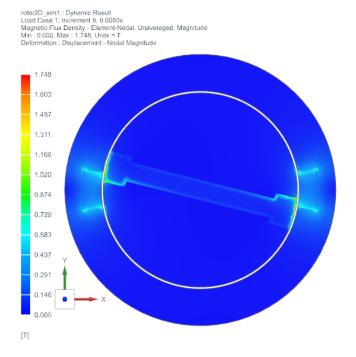
- 5. Solve the Solution. This will take about 1 min.
- 6. Now verify the oscillating behaviour of the rotor. To do so, open the results and display the 'Total Displacement' of the dynamic joint as in the below picture.



7. Also plot the graph showing the eddy current losses on the rotor.



8. Then plot the Magnetic Flux Density.

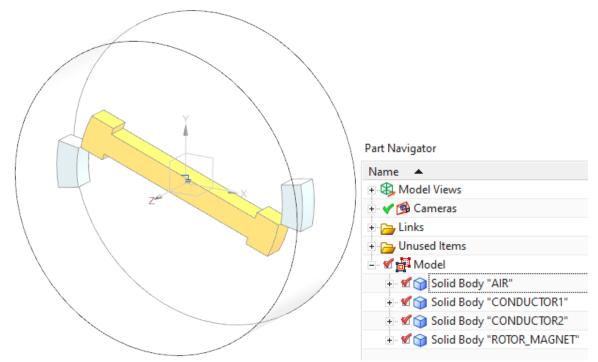


This ends the tutorial.

2.1.3 3D Enforced Driver

In this example we will use an enforced driver and 3D geometry.

Estimated time: 1 h. The following picture shows the geometry and the named CAD bodies.



To set up the model, follow these steps:

- Download the model files for this tutorial from the following link: https://www.magnetics.de/downloads/Tutorials/6.CouplMotion/6.2SimpleRotor3D. zip
- 2. Open the file 'rotor3D.prt'.
- 3. Switch to Simcenter Pre/Post
- 4. Create a new FEM and Simulation. Choose Solver 'MAGNETICS' and Analysis Type '3D Electromagnetics'. Switch off the 'Create idealized Part'.
- 5. Choose Solution Type 'Magnetodynamic Transient'. Name the solution 'Enforced'.
 - In register 'Output Requests' under 'Plot' activate 'Current Density' and 'Eddy Current Losses Density' to enable the calculation of eddy currents. Also enable 'Nodal Force virtual' to check for these results.
 - Under 'Table' activate 'Eddy Current Losses' to enable the calculation of integrated current losses and to get them written into a tabular file. Also enable 'Motion Data' to get results of displacement and velocity of the motion driver.

Solution			υ×.	
Solution			^	Plot
Name	Enforced			Table
Solver	MAGNETICS		•	Total Force - virtual
Analysis Type	3D Electroma	agnetics	-	Total Moment - virtual
Solution Type	Magnetodyn	amic Transient	•	Total Lorentz Force
Magnetodyna	nic Transiant			RotorBand Torque - stresstensor RotorBand Force - stresstensor
Output Req Time Steps Initial Cond Coupled Th Coupled Ela Coupled Pa	uests itions ermal asticity	Plot Magnetic Fluxdensity Magnetic Fieldstrength Electric Fieldstrength Current Density Eddy Current Losses Density Magnetic Potential (a-Pot)	^	Electrode Voltage Electrode Current Electrode Power Circuit Voltage Circuit Current Circuit Power Eddy Current Losses
		Nodal Force - virtual Nodal Moment - virtual Forcedensity - virtual Lorentz Force Poynting Vector Material Properties Table	~	 Hysteresis Loss - steinmetz Eddy Current Loss - steinmetz Excess Loss - steinmetz Motion Data Motor Efficiency FFT Tables 4D Fields

• In register 'Time Steps' set the 'Time Increment' as shown. This corresponds to a rotor velocity of 5.000 U/min if the step size is 1 deg. We will use this later.

Output Requests	Time Step Option	Constant	
Time Steps	Time Increment	1/(5000*360/60)	c -
Initial Conditions			3
Coupled Thermal	End Time Option	Number of Time Steps	
Coupled Elasticity	Number of Time Steps	45	
Coupled Particle			

• In register 'Initial Conditions' accept the defaults. Hint: The option 'Set back after Solve' can be deactivated to allow restarting from a previous run.

Output Requests	Magnetic	Zero Vector Potential	
Time Steps	Initial Time	0	s .
nitial Conditions		•	3
Coupled Thermal	U Output		
Coupled Elasticity	rel. Magnetic Perm	eabililty (mur) in Nonlinear Domain	from File
Coupled Particle	General Motion		1
Coupled Particle	General Motion Current Step	0	

- Click Ok.
- 6. Switch to the Fem file.
- 7. Mesh the rotor:
 - Use (3D) tetrahedral elements and the half of the suggested element size.

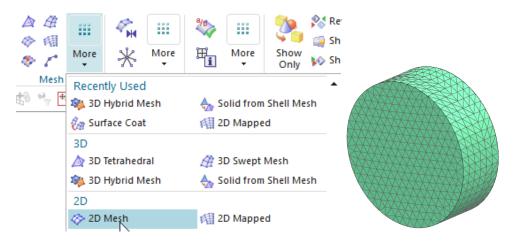
- Assign material 'Iron_Sample1' from the Magnetics material library.
- Key in the value 87 Kg mm^2 for Inertia RZ. This is necessary only in case of dynamic motion of the rotor because the forces acting on the rotor will be transformed into a motion step by Newtons law.
- name the collector 'Rotor_Iron', click OK to finish.

SolidPhysical	ა	X
Physical Property Table		^
Name	Rotor_Iron	
Label	1	
Properties		^
Material	Iron_Sample1 👻 🤇	۵
Conductor Model	· · · · · · · · · · · · · · · · · · ·	×
Rigid Body Motion	/	^
Mass	kg 🔹 👻	·]
Inertia RX	kg-mm² 🝷 🔻	·]
Inertia RY	kg-mm² 🝷 💌	·
Inertia RZ	87 kg·mm² ▼ ▼	
Active in Solution	,	~
CSYS	N	Y

- 8. Mesh the two magnets:
 - Use (3D) tetrahedral elements with half of the suggested element size.
 - Assign material 'N30EH at 100C'. Name the collector 'Magnets'. OK. Hint: For the north direction, we want x. Because x is the default, there is nothing to do now.

SolidPhysical		ა x
Physical Property Table		^
Name	Magnets	
Label	2	
Properties		^
Material	N30EH at 100C	- ¢
Conductor Model		^
Model	Massive	-
Rigid Body Motion		~
Active in Solution		×
CSYS		^
Material CSYS	Absolute	•
Output Total Moment CSYS	Absolute	•

- 9. Mesh (2D) the three outside faces of the body 'AIR'.
 - Use (2D) tri elements with a quarter of the suggested element size.



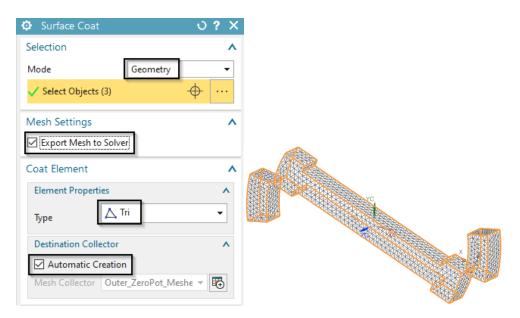
- In the mesh collector of this 2D mesh, set the Type to 'ZeroPotential' and click 'Create Physical...'. and OK. This setting will impose a boundary condition at the mesh level, so there is no need to give a zero potential condition in the Sim file.
- name the mesh collector 'Outer_ZeroPot_Meshes'. OK. Notice: This 2D mesh will be used two times: First it serves as boundary condition and second it is used as border for the following air mesh (Solid-from-Shell Mesh).

Ø Mesh Collecto	r	- ა ? X		
Properties		^		
Physical Property		^	ZeroPotential	ა? X
Туре	ZeroPotential	-	Physical Property Table	^
Shell Property	None 👻 🐳	\$ 🎨 🔻	Name ZeroPotential1	
Name	Outer_ZeroPot_Meshes		Label 4	
	ОК	Cancel	ОК	Cancel

- 10. Now create 2D Surface Coat meshes on the parts:
 - Blank the 2D meshes and also the air body.
 - Choose the meshing function 'Surface Coat',

2D	
🗇 2D Mesh	📢 2D Mapped
💱 2D Dependent	🗞 2D Local Remesh
🚑 Surface Coat	
1D and 0D	

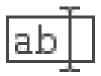
- Select the bodies of the two conductors and the rotor,
- Set the 'Mode' to 'Geometry', activate 'Export Mesh to Solver' and 'Automatic Creation', Check that the 'Type' is set to 'Tri'.



- The system creates by default a physical of type NotToSolver what is correct because we want to use these 2D meshes only as borders for the following air mesh (Solid-from-Shell Mesh).
- Assign the name 'Inner_Coat_Meshes' to this mesh collector.

Mesh Collector		ა? X
Properties		^
Physical Property		^
Туре	NotToSolver	•
✓ Shell Property	NotToSolver2	- & 🔯 -
Name	Inner_Coat_Meshes	

11. We want to have all collector names written to the corresponding physicals and meshes because this is easier to use. Therefore click the button 'Rename Meshes and Physicals from Collectors' from the Magnetics toolbar.



- 12. Next we create a new 3D mesh collector that will later hold the 3D air mesh. We will set up this collector in a way that it automatically updates the air mesh at every new rotor position. It will automatically use the 'Solid from Shell' mesher, therefore it is necessary to have meshed the 2D border of this 3D air.
 - First choose the function to create a new collector. Name this collector 'UpdatingAirMesh'.

	Ø Mesh Collect	or	0? X
	Element Topolog	ах	^
	Element Family	3D	•
	Collector Type	Solid	•
🛨 🗹 💸 2D Collectors	Properties		^
🖙 🐼 🏷 Mesh Controls	Physical Propert		v
- Marcollectore	Auto Mesh	у	v
🕂 🗹 🎼 Rotor, 🍅 New Collector	Name	UpdatingAirMesh	
🗄 🗹 🎼 Cond 🛕 New 3D Mesh 🔸	Warne	opdatingAinviesn	

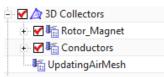
• In box 'Physical Property' set the 'Type' to 'FluidPhysical'. Use the button 'Create Physical' to create a new physical. In the new physical choose the material Air. OK.

r	ა? X		
у	^		
3D	-		
Solid	-	FluidPhysical	ა? X
	^	Physical Property Table	^
FluidPhysical	▲ - ++++++++++++++++++++++++++++++++++++	Name FluidPhysical1 Label 6	
None		Properties	^
UpdatingAirMesh	Create Physical	Material Air	•
	Solid FluidPhysical None	y A 3D J Solid J FluidPhysical Create Physical	y A 3D A Solid A FluidPhysical Air Create Physical y FluidPhysical Property Table Name FluidPhysical 1 Label 6 Properties Material Air

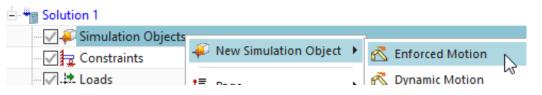
- Back in the mesh collector dialogue expand the box 'Auto Mesh' and set the option 'Activate' to 'Auto Mesh at Solve'.
- Next choose the button 'Shell Boundary for 3D Mesh'. In the following dialogue select the two border physicals and click on 'Add'.
- Then press Close and Ok to close all dialogues.

OMesh Collector		υx				
Element Topology	ý	^	Physical Property Tal	ble Mana <u>o</u>	ger	?
Element Family	3D	•	Create			V
Collector Type	Solid	•	Filter			v
		_	Selection			^
Properties		^	Name	Label	Туре	
Physical Property		^	NotToSolver1	3	NotToSolver	
Туре	FluidPhysical	-	Outer_ZeroPot_Meshes	4	ZeroPotential	
🗸 Solid Property	FluidPhysical1 👻 😓 🗞	-	Inner_CoatMeshes	5	NotToSolver	
Auto Mesh	,	^	Ŀ			b,
Activate	Auto Mesh at Solve	-	List			^
Shell Boundary fo	r 3D Mesh (0)		Outer_ZeroPot_Meshes			Ð
Element Type	Tetra4 Create Shell Boundary for	3D Mesh	Inner CoatMeshes			×
Name	UpdatingAirMesh				Cl	ose

• The navigator now shows the new collector. Notice that there is no mesh in it. This mesh will be created at solve time automatically through the function 'Solid from Shell Mesh'. The mesh will also be updated at every motion step.



- 13. Switch to the Sim-file.
- 14. Create 'Simulation Object' for the rotor movement:



• Choose the function 'Enforced Motion'. The following dialogue appears

Enforced Motion		<u> </u>				
饞 Revolute by General N	Notion	•	Physical Property T	able Man	ager ?	
Name		v	Create		N	1
Destination Folder		v	Filter		N	/
Rotor Geometry		^	Selection		/	
Moving Parts (0) Airgap for Torque Out	put		Name Rotor_Magnet	Label 1 2	Type SolidPhysical SolidPhysical	
Definition		^	🔒 UpdatingAirMesh	6	FluidPhysical	
Axis Driver Angular Step	Z Angular Step	▼	B		• × 0]
Angular Step	ų	• • •	List		/	1
Initial Conditions Angle	0	∧ ▼ ▼ °	Rotor_Magnet		Ð	Į
Card Name GMEnfRevolu	te				Close	

- At 'Rotor Geometry' click on 'Create Moving Parts' III. In the following dialogue select the physical Rotor_Magnet and click Add, Close.
- Notice that the 'Angular Step' is set to one degree for each step. This is already what we want so there is no change necessary. Leave all other settings at their defaults. OK.
- Hints: By this procedure the MAGNETICS Solver will rotate the 3D Rotor mesh at each timestep for the desired amount and will automatically create 'Solid from Shell' meshes for the air using the previously created 2D boundary meshes.
- 15. Model setup is now done. Save your parts, blank the meshes (maybe you leave the rotor mesh visible because this will nicely show the motion). Decrease the window size, because it will pop up to foreground at every time step.

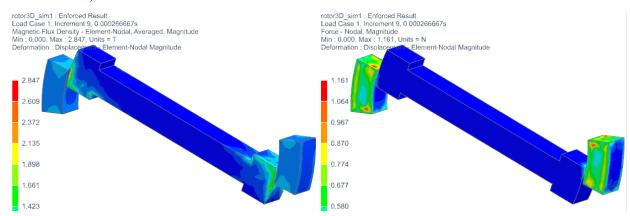
16. Solve the solution. This will take 5-10 minutes because of the 45 steps to run. A progress bar at the bottom of the Simcenter window shows the current and the remaining steps.

Step 4 of 45, 8%

17. The finishing of the solve is shown by the information window as below



- 18. After the solve finishes you can post process the results.
 - Open the plot results. All motion steps are stored in the result file. This allows running animations over the iterations and creating movies if desired.
 - The following picture shows the magnetic flux density result (left) and the nodal forces (right) at time step 9. (Results are set to averaged at nodes and element edges are blanked).

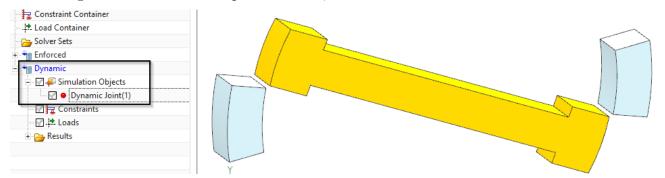


19. After this General Motion run the moving meshes are set to 'Locked'. If an update of the meshes to the original situation is desired, unlock the moving meshes and perform a mesh update.

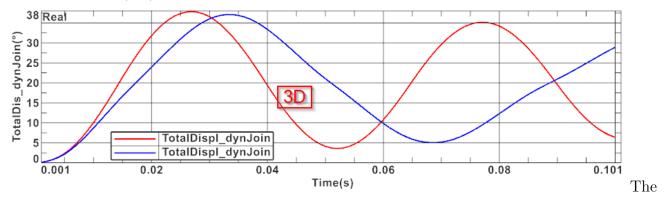
The tutorial is complete.

2.1.4 3D Dynamic Driver

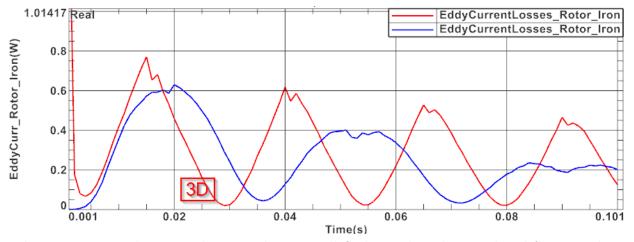
The completed Sim file, that resides in the tutorial folder, contains also a solution (named 'Dynamic') with a dynamic joint. This can be used for further studies. (Set the 'Number of Time Steps' to 100 before solving the complete period.) The following result graphs are made with a larger number of time steps. Of course, that solution needed much more time.



The following picture shows the resulting displacement of the previous 2D dynamic (blue) and the 3D dynamic (red) rotor.

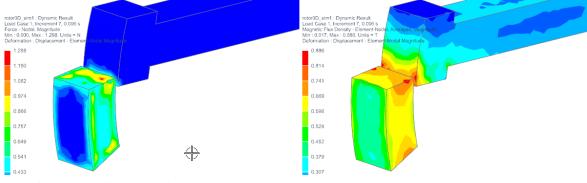


next picture shows the resulting eddy current losses of the 2D dynamic (blue) and the 3D dynamic (red) rotor.



When comparing the 2D and 3D results one will find out that the simulated forces and torques are higher in the 3D case, even if the 2D is set to the same z thickness. Therefore, also the dynamic rotation speed becomes higher in 3D. This effect can be explained as follows: The magnetic forces appear mainly on the border faces between air and magnetic material. Because the 3D model has additional side faces, and these are used in the simulation, the forces here are

higher. The following picture illustrates this: It shows the force distribution on the 3D rotor. It can be seen, that forces are not homogeneously distributed. At the side faces they are higher. Also the magnetic flux density result shows this effect. This is one reason why 3D simulations can be more realistic.



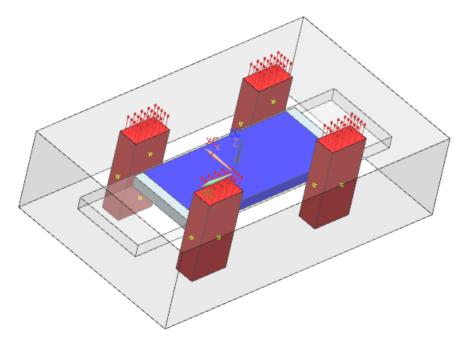
The tutorial is complete.

3 Tutorial 2

3.1 Relay 3D, Translational Motion

In this example we show the usage of the 'Dynamic General Motion' technique in the context of a 3D Relay.

Estimated time: 1 h



Follow the steps:

This example is shown using 3D, but the technique is possible in 2D very similar. The corresponding model in 2D is stored at the same folder location, so it can be seen there how it is set up.

Estimated time: 1 h

Follow the steps to reproduce it:

- Download the model files for this tutorial from the following link: https://www.magnetics.de/downloads/Tutorials/6.CouplMotion/6.3Relay.zip
- 2. Open the part file 'relay3D.prt'.
- 3. Start Simcenter Pre/Post, create a new Fem and Sim File for 3D.
 - Switch off 'Create Idealized File',
 - Chose Solver 'MAGNETICS' and Analysis Type '3D Electromagnetics',
 - Choose Analysis Type 'Magnetodynamic Transient'.

• In register 'Output Requests', 'Table', activate 'Motion Data'. Others can be activated as desired.

Solution			υx	
Solution			^	
Name	Dynamic			
Solver	MAGNETICS			
Analysis Type	3D Electromagne	etics		
Solution Type	Magnetodynami	ic Transient		
Magnetodynan	nic Transient		^	Plot
Output Requ Time Steps Initial Condit Coupled The Coupled Elas Coupled Part	ions II rmal II ticity II icle II II II II II II II II II II II II II	Magnetic Flux density Magnetic Fieldstrength Electric Fieldstrength Current Density Eddy Current Losses Density Magnetic Potential (a-Pot) Nodal Force - virtual Nodal Moment - virtual Forcedensity - virtual Lorentz Force Poynting Vector Material Properties e Fields Fields		Table Total Force - virtual Total Moment - virtual Total Lorentz Force RotorBand Torque - stresstensor Electrode Voltage Electrode Voltage Circuit Voltage Circuit Voltage Circuit Power Eddy Current Losses Hysteresis Loss - steinmetz Ekcess Loss - steinmetz Motion Data

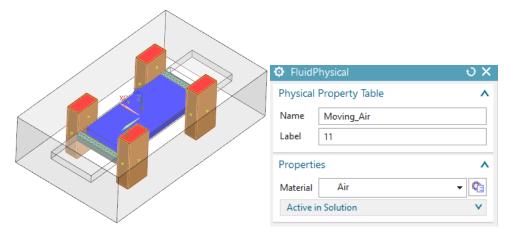
• In register 'Time Steps', select a time increment of 0.1 sec and 10 time steps.

Magnetodynamic Trans	sient			^
Output Requests	Time Increment	0.1	s	• •
Time Steps	Number of Time Steps	10		
Initial Conditions	Number of Time Steps	10		
Coupled Thermal				
Coupled Structural				

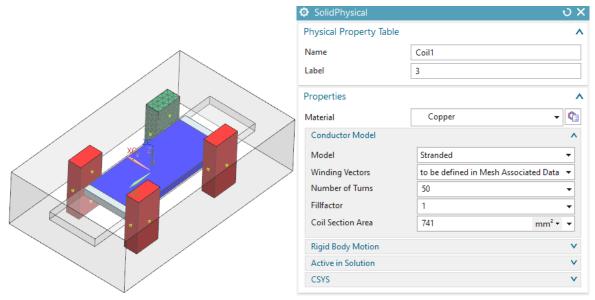
- 4. Change to the Fem file.
- 5. In the following steps we create all 3D Meshes (all but the air gap). This will be done by tetrahedral elements and appropriate mesh collectors.
 - First, create mesh mating conditions. So, make all polygon bodies visible, click on 'Mesh mating Condition', drag a window over the geometry, set the 'Mesh Mating Type' to 'Glue-Coincident' (default), click 'OK'. Check, there will be 22 mesh matings created.
 - Then, mesh the Plunger with one fourth of the suggested mesh size. Assign the material 'Iron_Sample2' as well as a 'Rigid Body Motion' mass of 0.28 Kg.

	SolidPhysical	ა x
	Physical Property Table	^
	Name	Plunger
	Label	1
	Properties	^
xe	Material	Iron_Sample2 🔹 🔇
	Conductor Model	v
	Rigid Body Motion	٨
	Mass	0.28 kg • •
	Inertia RX	kg·mm² 🝷 💌
	Inertia RY	kg·mm² ▼ ▼
	Inertia RZ	kg·mm² ▼ ▼
	Active in Solution	v
	CSYS	۷

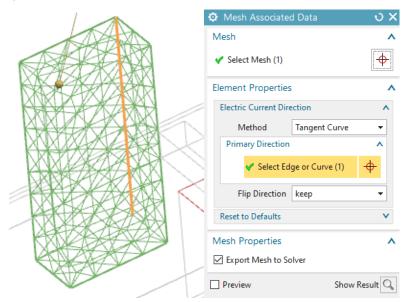
• Mesh the (two) moving air stripes with one fourth of the suggested element size. Use a 'FluidPhysical' and assign material 'Air'.



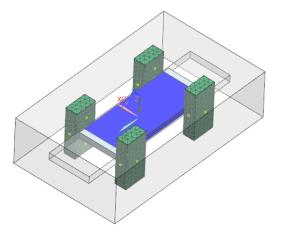
• Mesh the first Coil with the suggested mesh size and assign material 'Copper'. Also, set the 'Conductor Model' to 'Stranded' with 50 turns, a fill factor of 1 and a coil section area of 741 mm^2 .



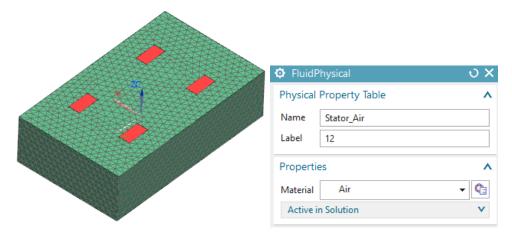
• Edit the 'Mesh Associated Data' of the coil mesh and define the direction of current (use one of the z-dir edges. Take care that the polygon body is shown.).



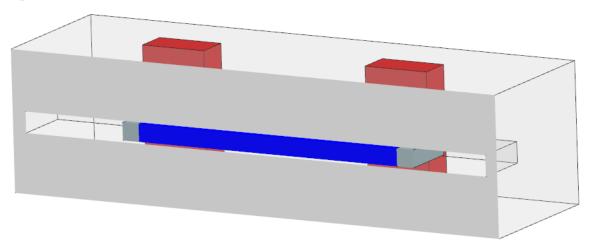
• Proceed in the same fashion with the other three coils (meshing, material and current direction).



• Finally, create a mesh with one third of the suggested mesh size for the stator air and set the small feature tolerance to 2%. Again, use the type 'FluidPhysical' for this.

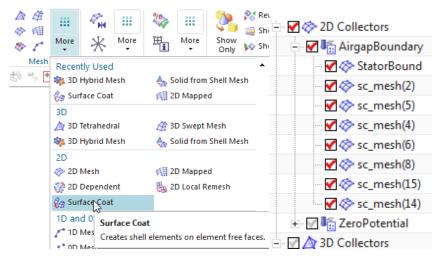


6. Following we do preparations for the 3D air gap mesh. E.g. the mesh that will update with each movement step. Following picture shows the plunger and how the updating air gap surrounds it. So, if the plunger moves and the air gap is updated, a new simulation step can be done.



This updating air gap mesh will be of type 'Solid from Shell Mesh' and therefore needs 2D boundary meshes from which it depends. Therefore, NX Magnetics provides a feature that allows to recreate that mesh prior to each solve. We will set up this feature now.

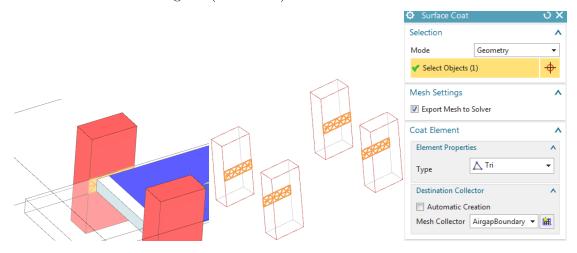
• Following you create all boundary meshes of the updating air gap. Therefore, you create surface coat meshes (picture below left) on all faces that belong to the boundary of the air gap. The right side picture below shows how the navigator looks after all the boundary meshes are there.



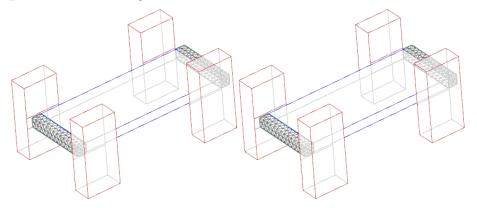
• To collect the following meshes, first create an 2D mesh collector, name it 'Airgap-Boundary'. Use a physical of type 'NotToSolver' because these meshes will act as borders only for the 3D air gap mesh.

	Ø Mesh Collector		ა x		
	Properties		^		
	Physical Property		^		
🝘 relay3D_fem1.fem	Type	NotToSolver	-	NotToSolver	ა x
🗇 relay3D.prt	Shell Property	AirgapBoundary 👻 😓 🎨		Physical Property Table	• •
🗉 🖌 🗁 Polygon Geometry	• onen roperty	• • • • • • • • • • • • • • • • • • • •		Name AirgapBoundary	
🕶 🛃 Mesh Controls	Name	AirgapBoundary		Label 7	
		ОК Са	ncel	OK Car	ncel

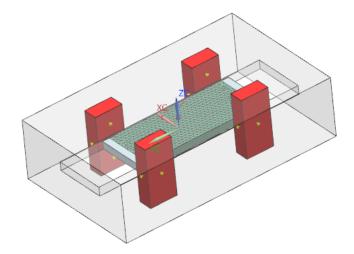
• First, create a 'surface coat' on the 4 inner side sections of the four coils that are in contact with the moving air (see below).



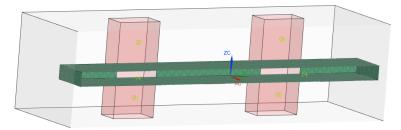
• Next create surface coat meshes for the left (e.g. green mesh) and the right surface parts of the moving air.



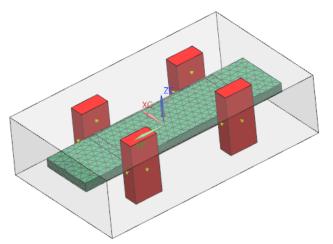
• Then, create a surface coat mesh for the moving air.



• Finally, create a surface coat mesh stator bound air (that provides the extend over which the air can move). For easier selection of these interior faces of the 'Stator_Air', use the helpful 'Clip Section' feature from toolbar 'View'. Select all inside faces of the air as shown in the picture below (Note: You have to invert the clip section to select the faces on the opposite side).



• When all boundaries are visible, it should look like the following picture.



- 7. With these air gap boundaries being completed we now set up the automatically updating 3D air gap mesh:
 - Create a 3D mesh collector.

🗄 🕜 🖄 3D Collectors (
🕂 🗹 🎼 Plunger	🍅 New Collector
	A Now 20 Mach

• Name the collector UpdateAirGap, set the physical type to FluidPhysical, click Create Physical and set the material to Air. Click Ok.

Mesh Collecto	or	<u> ७ ×</u>	FluidPhysical	ુ v x
Properties		^	Physical Property Table	^
Physical Property	r	^	Name FluidPhysical1	
Туре	FluidPhysical	•	Label 14	
Solid Property	None	- 😻 -	Properties	^
Solid from Shell M	Mesh Update	Create Physical.	Material Air	- (
Name	UpdateAirGap		Mechanical Deformation	V

• Back in the mesh collector dialogue expand the box 'Solid from Shell Mesh Update' and set the option Activate to 'Create/Update Mesh at Solve'.

Mesh Collector		ੁ v x	nections	▼ U
Properties		^	° _\ ⊕ -	°¢
Physical Property		^	fem 🖸	×
Туре	FluidPhysical	•		
🗸 Solid Property	FluidPhysical1 👻 🐥 ŧ	> 💌		
Solid from Shell Me	sh Update	^		
Activate	Create/Update Mesh at Solve	-		
Boundary (1)				
Name	UpdateAirGap	Create	e Bounda	ry
	ок с	Cancel		

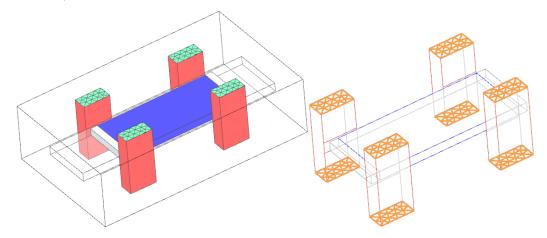
• Click on 'Create Boundary' and in the following dialogue select the previously created physical AirgapBoundary that holds the 2D boundary.

Physical Proper	rty Table N	×			
Create			v	-	
Filter		v			
Selection			^		
Name	Label	Туре			🖃 🗹 🗇 2D Collectors
AirgapBoundary	7	NotToSolver	Π.		🕀 🖌 🎼 AirgapBoundary
ZeroPotential1	10	ZeroPotential		Ξ	🛨 📝 🎼 ZeroPotential
					🖃 🗹 🖄 3D Collectors
					🕂 🗹 🎼 Plunger
se an		= X i	0		🗉 🗹 🎼 Moving_Air
					🗉 🗹 🎼 Coil1
List			^	_	🕂 📝 🎼 Coil2
AirgapBoundary			⁺♠		🗉 🗹 🎼 Coil3
		F	- vr	4	🕀 📝 🎼 Coil4
			Add	ł	🗉 📝 🎼 Stator_Air
		Cl	ose	ľ	UpdateAirGap

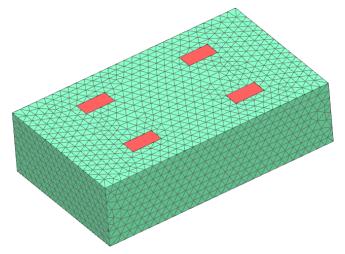
- Click Close, Ok and the mesh collector is created. Notice that it contains all necessary information to create a Solid from Shell Mesh, but the mesh itself will be created at solve time. (Alternatively you can create the mesh manually now, it makes no difference).
- 8. Now create all outer 2D surface coat meshes (equivalent as in the last step). However, in contrast to the last step, use a physical property of type ZeroPotential. Hint: We do this procedure in order to fix the Boundary condition (i.e. ZeroPotential) directly in the .fem file. However, this could also be done in a later step in the .sim file.

Mesh Collector	t	УX				🖃 🗹 🗇 2D Collectors
Properties		^	🙃 Zerc	Potential	υx	🛨 📝 🎼 AirgapBoundary
Physical Property		^			0 ~	🖃 🇹 🏣 ZeroPotential
			Physica	al Property Table	^	🖌 🏹 🛷 sc_mesh(9)
Туре	Zeror otentiar	•	Name	ZeroPotential1		🗹 🛷 sc_mesh(10)
🗸 Shell Property	ZeroPotential1 🔻 🔑 🎨 🔻	•				🗹 📀 sc_mesh(13)
Name	ZeroPotential		Label	10		🖌 🎻 sc_mesh(11)
						🗹 🗇 sc_mesh(16)
	OK Cance	el		OK Car	ncel	🖃 📝 🆄 3D Collectors

• First, create surface coat meshes for the top and bottom sides of all four coils (see picture).



• Then create a surface coat mesh for the outer air.



9. Meshing is done, change to the Sim file.

- 10. Create a Dynamic Motion to define the movement of the plunger.
 - Create a simulation object of type 'Dynamic Motion'. Activate 'Slider by General Motion'.

🗄 📲 Dynamic				Dynamic Motion	ა? X
+ ₩ ₩ Simulation Ob	Select All			除 Slider by General Motion	-
	Remove All Simulation Objects			Name	v
🕂 📇 Results	🔑 New Simulation Object	₽	Enforced Motion	Destination Folder	v
	i Information		Dynamic Motion	Slider Geometry Moving Parts (0)	

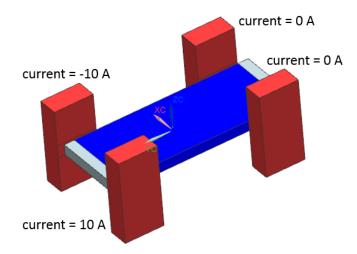
• In the dialogue first define the Slider Geometry. Click on 'Create Moving Parts' and add the two physicals Moving_Air and Plunger that will move to the list. Click Close.

	ુ v x			
	v			
	v			
	v			
	^			
	^		ty Tabl	e Manager
Y	•	Filter		
Force - virtual	-	Selection		
				Type FluidPhysical
	V	A Plunger	1	SolidPhysical
		🔒 Stator_Air	12	FluidPhysical
	V	A Undate∆irGan	12	FluidPhysical
	v	List		
	Y Force - virtual	 ✓ 	 ✓ ✓	✓ ✓ ✓ ✓ ✓ ✓ ✓ Ý Force - virtual ✓

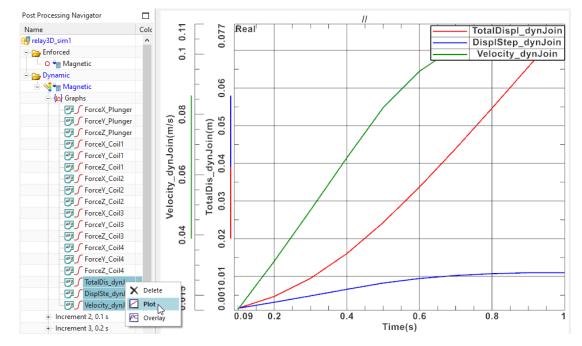
• Back in the Dynamic Motion dialogue, set the Direction to Y and accept all other defaults with Ok.

Joint Definition	^	
Direction	Υ -	
Magnetic Load	Force - virtual 🔹	

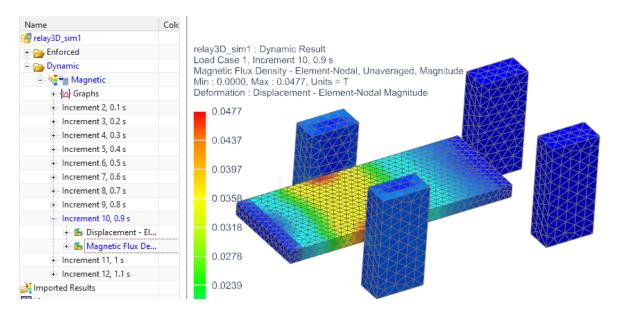
11. Define the electric currents on the four coils: Define zero amperes on the two right coils (see picture) and define 10 and -10 amperes on the two left coils. This will result in a Y force on the plunger.



- 12. Solve the solution.
- 13. Post processing.
 - We first check the motion data result. This contains Total Displacement, Displacement Step and Velocity.
 - Open the associated result file in the post processing navigator. A list of all tabular results appears above the plot results.
 - plot the three results as shown below.



- If acceleration is needed this can be derived by differentiation using the mathematical operations for afu graphs.
- 14. Next we check the plot results. After 10 time steps the Magnetic Flux density field on the slider should look as follows

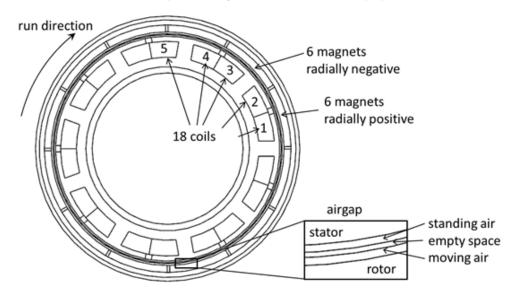


This ends the tutorial.

4 Tutorial 3

4.1 Permanentmagnet Synchron Motor Analysis

In this tutorial a permanent magnet electric motor is analyzed for torque. Notice that there is also a video of this tutorial in https://magnetics.de/index.php/learn.



4.1.1 Model Setup by AutoFEM

For this model setup we want to use the feature 'AutoFEM' that allows to perform this process highly automatized. Some preparations in the CAD (already done here) are the following. The faces of the CAD model have names. When building the fem model with 'AutoFEM' we will take advantage of these preparations. To assign names to a CAD face you can select a face and then go to RMB properties. Follow the steps:

- Download the model files for this tutorial from the following link: https://www.magnetics.de/downloads/Tutorials/6.CouplMotion/6.4ElectricMotor. zip
- 2. Open part 'ElectricMotor.prt'.
- 3. Start Simcenter and the Pre/Post application and chose function 'New FEM and Simulation'.
- 4. Switch off 'Create Idealized Part', set the Solver to 'MAGNETICS' and 'Analysis Type' to '2D or axisym Electromagnetics'.
- 5. Choose the solution type 'Magnetodynamic Transient'.
- 6. In the dialog Solution set the settings in registers 'Output Requests' and '2D' as shown in the next picture. OK.

					Plot		
Solution			t) ? X	Table		
Solution				^	Total Force - virtual		
Name	Solution 1				Total Moment - virtual		
Solver	MAGNETICS			•	Total Lorentz Force		
Analysis Type	2D or axisym El	ectromagnetics		•	RotorBand Torque - stresstensor		
Solution Type	Magnetodynan			•	RotorBand Force - stresstensor		
Solution type	Iviagnetodynam				Fluxlinkage - Vectorpotential on Conductors		
Magnetodyna	mic Transient			^	Electrode Voltage		
···· Output Red	quests PI	-+		^	Electrode Current		
- Time Steps				^	Electrode Power		
- Initial Cond		Magnetic Fluxdensity			Circuit Voltage		
- Coupled Th	hermal	Magnetic Fieldstrength			Circuit Current		
2D		Current Density			Circuit Power		
		Eddy Current Losses Dens	·		Eddy Current Losses		
] Magnetic Potential (a-Po	t)		Hysteresis Loss - steinmetz		
		Nodal Force - virtual			Eddy Current Loss - steinmetz		
		Nodal Moment - virtual			Excess Loss - steinmetz		
		Lorentz Force			Motion Data		
		Displacement			FFT Tables		
		Material Properties			Motor Efficiency		
	Та	ble		×	NVH Coupling		
Magnetody	namic Transie	ant			٨		
Magnetody							
Output P	Requests	Plane	Absolute X,Y		•		
- Time Ste	eps	Axisymmetri	c				
- Initial Co	onditions	Thickness					
- Coupled	l Thermal	Inickness	100		mm 👻 👻		
2D							

- 7. Change the displayed part to the Fem file.
- 8. As a prerequisite step add the required Materials to the local libraries; to do so follow the workflow below.

Hint: This prerequisite step must be done for an AutoFEM simulation with automatic Material assignments; more precisely said materials must be added to the 'local materials library' first, in order to be properly assigned later by the AutoFEM function. Note that this must be done before the usage of the AutoFEM function.

• First, Select the 'Manage Materials' button

File	Home	2	Nodes and Elements Res		Vie	w
6	2	C É	Kerge Face	. 🌳		4
Chan Displayed Conte	d Part 🔻	F	Manage Materials Defines and manages materia	ls. try	More •	4

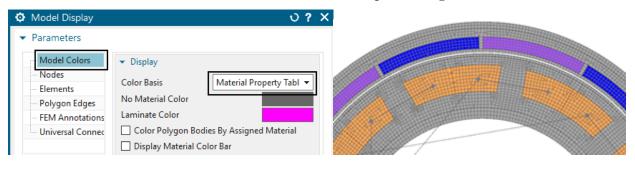
• Then, right click on the material 'ElectroSheet_Sample2' from the Magnetics Library and select 'Load Library Material', to load this Material into the local material Library.

🧿 Manage Materials	4	Configure Library Materia					
Material List	Material List						
R Library Materials	0	Re-label					
Libraries	0	Show Usages					
Default Material Library	MØ	Show Body(s) Without ar					
User MatML Library	걊	Remove Usages					
C:\Program Files\Siemens\NX1899\MAGNETI	CS\Magnetics	Materials x	ml	Ç	Update Material		
	6	Load Library Material					
Materials		\times	Delete				
Name 🔺	Used	Category	Тур	і ц	Delete Orphaned User M		
ElectroSheet_Sample1		other, li	lsot	R	Delete User Material (UM		
ElectroSheet_Sample2		other, n	lsotr	tropic iviag			
Gray iron, as cast		Metals	lsotr	opic	Magne		

• Repeat the last step for the other four required materials (see picture below). After completion the Local Materials List looks as shown below.

Manage Materials								
Material List								
💫 Local Materials								
 Materials 								
Name	Туре	Label	Library	Magnetic relative Permeabilit	Magnetic Remanent Fieldstrengt	Electric Conductivity		
Air	Fluid	1	Magnetics_Materials.xml	1	0 A/m	0		
Copper simple	Isotropic	2	Magnetics_Materials.xml	1	0 A/m	58000000		
ElectroSheet_Sample2	Isotropic	3	Magnetics_Materials.xml	1500	0 A/m	5800000		
N30EH at 100C	Isotropic	4	Magnetics_Materials.xml	1.07653	761621 A/m	625000		
N30EH at 100C (neg)	Isotropic	5	Magnetics_Materials.xml	1.07653	-761621 A/m	625000		

9. For easier display of colors: RMB on the Fem file, choose 'Edit Model Display...', register 'Element' and set the 'Color Basis' to 'Material Property Table'. This will later make the meshes look in different colors as shown in the below picture right side.



10. Choose the function 'Auto FEM' if from the Magnetics toolbar and choose the register '2D'. Activate the settings as shown in the picture.

🗘 Auto FEM	ა? X	Mesh		^	
Model Type	2D or axisym 🔻	✓ Mesh Groups Group Filter			
Face Options	^	Element Type Attribute	Tri	•	
✓ Named Faces Filter Colored Faces		Material		^	
Network	nes 🔨	Attribute ElectroSheet_Sample2 Copper Air	MAT	^	
Filter Length	COIL	N30EH at 100C		~	· · ·
Import Network		Assign	Value	•	
\6.4ElectricMotor\cc	omplete\Network.txt	Value	1.5 mm	•	
File Format Help File Format Help	^	Division Factor Attribute [mm]	1	*	
# These lines explain # <	n the syntax for the N 💊	MAGNETICS specific		^	
Mesh	^	Mesh Creation Order	Element Size	•	L.

Hints:

- 'Named Faces': All faces that have names in the CAD part will be found and there will be groups created for them in the Fem part.
- 'Connectors on faces / Filter': All faces that have a name starting with 'COIL' will be found. A connector element will be created on those faces. Connector elements are necessary to connect 1D circuit networks to the finite-element mesh. The length of those connector elements will be 20 mm. This setting only plays a role for visibility effects. Each connector element will have two points (A and B) which can be understood as the coil ends.
- 'Import Network': The file 'Network.txt' that is chosen contains definitions for 1D circuit network elements. Those elements contain the winding scheme in our case. Take a look into this text-file to see the definitions.
- 'Mesh Groups': This option leads to a meshing of all groups that correspond to the value in the 'Group Filter'. If the 'Group Filter' is empty as in our case simply all groups will be meshed.
- 'Element Type': The meshing will be done using the given element type.
- 'Material': The option 'By Attribute' will search for the given attribute ('MAT' in this case) on faces of the geometry to select a material.
- 'Element Size': The size for the elements. Hint: You can use expressions here. This makes it easy to change the mesh sizes later.
- 11. Press OK in the Auto FEM dialog. The network and meshes are created.
- 12. Edit the physical properties of the coils:

• First, edit the physical of the first coil (i.e. COIL01). Apply the settings for 'Material' and 'Conductor Model' as shown below.

PlanePhysical		ა? X
Physical Property Table		^
Name	COIL01	
Label	23	
Properties		^
Material	Copper	-
Conductor Model		^
Model	Stranded	•
Number of Turns	50	•
Fillfactor	1	•
Rigid Body Motion		~
CSYS		~

• To copy the properties of this first coil to all other coils you select the button 'Multi Physical Edit, from the Magnetics toolbar. Insert the name of the physical to copy from 'COIL01' and 'COIL' as a filter for the physicals to copy to, OK.

Multi Physical Edit	ა? X
Physical Property Tabl	e to copy from 🔥 🔨
Solver	MAGNETICS -
Туре	PlanePhysical 👻
Name	COIL01
Physicals to paste to	^
Filter	COIL

- Maybe you check some other coils to verify they now all have the same properties.
- 13. Edit the magnets:
 - Modify the physicals of the magnets 'MAGNET_NEG1' and 'MAGNET_POS1'. Set the 'Material CSYS' to 'Cylindrical' and choose for the 'Material Orientation' the

absolute \leq coordinate system. See the settings as shown in the next picture.

PlanePhysical	ა ? X	PlanePhysical	ა? X
 Physical Property Table 	le	 Physical Property Tab 	le
Name	MAGNET_NEG1	Name	MAGNET_POS1
Label	32	Label	34
✓ Properties		 Properties 	
Material	N30EH at 100C (neg) 🔻 🔇	Material	N30EH at 100C 👻 🗘
Material CSYS	Cylindrical	Material CSYS	Cylindrical 🗸
Material Orientation		Material Orientation	
▼ Electromagnetic Solution	ons	▼ Electromagnetic Solution	ons
Active		✓ Active	
Conductor Model	Massive 🔻	Conductor Model	Massive 💌
Thermal Solutions		Thermal Solutions	
Elasticity Solutions		Elasticity Solutions	
 Motion Solutions 		Motion Solutions	
Post Processing		Post Processing	

• Again use the 'Multi Physical Edit' to copy these properties to all other 'MAG-NET_NEG' and 'MAGNET_POS' magnets, respectively.

Multi Physical Edit		ა? X	Ø Multi Physical Edit		0?>	<
Physical Property Tabl	e to copy from	^	Physical Property Tab	le to copy from	^	
Solver	MAGNETICS	•	Solver	MAGNETICS	•	
Туре	PlanePhysical	•	Туре	PlanePhysical	-	
Name	MAGNET_NEG1		Name	MAGNET_POS1		
Physicals to paste to		^	Physicals to paste to		^	
Filter	MAGNET_NEG		Filter	MAGNET_POS		

14. Edit the Iron: The Rotor and Stator should be set to 'Conductor Model' 'Laminated'. Otherwise the default model 'Massive' would be active and this would result in strong eddy current effects (if the solution is dynamic). So, modify the two physicals as shown in the following picture.

PlanePhysical	ა? ×	PlanePhysical	ა? ×
 Physical Property Table 		 Physical Property Table 	
Name	ROTOR	Name	STATOR
Label	4	Label	5
✓ Properties		✓ Properties	
Material	ElectroSheet_Sample2 👻 🔇	Material	ElectroSheet_Sample2 👻 🕼
Material CSYS	Absolute 👻	Material CSYS	Absolute 👻
▼ Electromagnetic Solutions		▼ Electromagnetic Solutions	
Active		Active	
Conductor Model	Laminated 👻	Conductor Model	Laminated 👻
Thickness of one Sheet	0.3 mm • •	Thickness of one Sheet	0.3 mm • •
EM Shape Functions	First Order 💌	EM Shape Functions	First Order 💌
Thermal Solutions		Thermal Solutions	
Elasticity Solutions		Elasticity Solutions	
Motion Solutions		Motion Solutions	
Post Processing		Post Processing	

15. Edit the resistors in the 1D network: Edit the mesh 1D collector 'ResistorsFromFile'. Create a new physical there and insert a resistance value of 2 ohm. Hint: This value only influences results if the motor is driven by voltage.

		PRESISTOR	ა? X
		Physical Prope	rty Table
		Name	PRESISTOR1
		Label	36
ElectricMotor_fem1.fem		Properties	^
ElectricMotor.prt		Resistance per	Element A
🗉 🗹 🗁 Polygon Geometry		Туре	Given 🔻
···· 🖌 🤠 Mesh Controls	Mesh Collector	Resistance	2 Ω • •
🖃 ✔ 🖍 1D Collectors	Properties	A Resistance	
🕂 🖌 📻 ConnectorsOn_COIL	Physical Property	^ Inductance per	Element A
🕂 🗸 📻 ResistorsFromFile 🔯 Select All	Type PRESISTOR	Inductance	0 H • •
	Crod Property None	 ✓ 2 100 	
🗉 🗹 🔆 Connection Collector: 🔑 Edit.	Name ResistorsFromF	ile Network ID	v

- 16. Switch to the Sim file.
- 17. Create parametric expressions: Create the following expressions. To create them, either use the dialogue (Menu, Tools, Expressions). Alternatively run the journal 'CreateExpressions.vb' in the part directory to automatically create them. (Menu, Tools, Journal, Play, Browse and select the vb file, Run)

	1 Name	Formula	Value	Units	Dimensionality	Туре
1	✓ Default Group					
2				mm 💌	Length 🔹	Number
3	CurrentAmplitude	6	6 A	A 🔹	Electric Current 💌	Number
4	CurrentFrequency	(6*SpeedMech/360)*(180.0/pi())	0.1 Hz	Hz 🔹	Frequency 🔹	Number
5	Poles	6	6		Unitless 🔹	Number
6	RotorStep	-1	-1 °	• 👻	Angle 🔻	Number
7	SpeedMech	1	1 rev/min	rev/min 💌	Angular Velocity 🔻	Number
8	TimeIncrement	-1*RotorStep/SpeedMech	0.1666666667 s	s 🔹	Time 🔹	Number

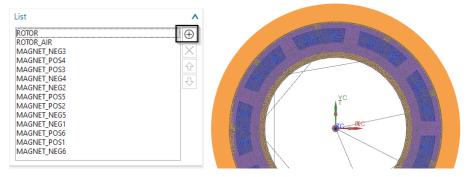
- 18. Edit the solution and set the
 - 'Number of Time Steps' to 90
 - 'Time Increment' to the newly created expression 'TimeIncrement'.

Magnetodynamic Transien	t		^
Output Requests	Time Step Option	Constant	•
Time Steps	Time Increment	TimeIncrement	c v v
 Initial Conditions 			5 • •
- Coupled Thermal	End Time Option	Number of Time Steps	•
2D	Number of Time Steps	90	•

- 19. Define the rotor motion:
 - Create a simulation object of type 'Enforced Motion 2D'. Accept the default type 'Revolute by Moving Band'.
 - Select the 'Airgap Rotor Edge' (outer) and the 'Airgap Stator Edge' (inner).

Enforced Motion 2D	ა? X	
🙀 Revolute by Moving Bar	nd 🔻	
Name	v	
Destination Folder	v	
Airgap Rotor Edge	٨	
Group Reference		
✓ Select Object (1)	Φ	
Airgap Stator Edge	٨	
Group Reference		
Select Object (1)	⊕ …	
Airgap, auto meshed	^	
Number of Air Gaps	1 •	
Material	🔒 ElectricMotor_fem1::Air 💌 🜊	
Rotor Geometry	٨	
Moving Parts (14)		
Definition	٨	
Rotation Axis	Ζ -	airgap edges:
Stator Shift X0	0 mm • •	Airgap Stator Edge
Driver	Angular Step 👻	stator
Angular Step	RotorStep ° 👻 🔻	Airgan Potor Edga
Tolerance Factor for Links	1e-05 👻	rotor Airgap Rotor Edge
Periodicity	v	
Card Name MBEnfRevolute		

- Assign Air for the material at the air gap.
- Also, at 'Rotor Geometry' select the 'Moving Parts'. These are ROTOR, ROTOR_AIR and all Magnets as shown in the below picture.



- Enter the expression 'RotorStep' in the field 'Angular Step'.
- Click OK to create the joint feature.

20. Define current for phase u:

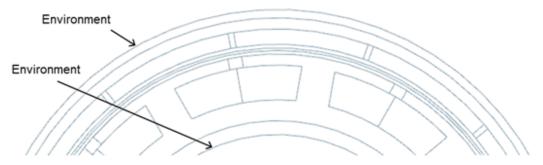
- blank all 2D meshes.
- Create a Load 'Current 2D' with Type 'On 1D-Circuits' and set the Method to 'Harmonic'.
- For 'Primary Node', select the point that is marked in the picture as 'U phase'. (Hint: The node is on the z=0 plane)
- For 'Secondary Node' select the starpoint.
- For 'Electric Current Amplitude', key in the expression 'CurrentAmplitude'.
- For 'Frequency' key in 'CurrentFrequency'.
- The 'Phase Shift' stays zero for phase u.
- Assign name 'u'. OK.

Current 2D	ა? ×
E On 1D-Circuits	•
Name	^
u Description	×
Destination Folder	×
Primary Node	^
✓ Select Object (1)	ф·…
Secondary Node	^
🗸 Select Object (1)	\oplus …
Magnitude	^
Method	Harmonic 🔹
Electric Current Amplitude	CurrentAmplitude A 🔹 🗸
Frequency	CurrentFrequency Hz • •
Phase Shift	0 • •
Network ID	v
Card Name CurrentOnCircu	

- 21. Define current for phase v and w:
 - Repeat the last steps with exception of the 'Primary Node' selection and 'Phase Shift'. For phase v use 120 degrees and for phase w use 240 degrees.

Current 2D		0? X	Current 2D	ა? ×
E On 1D-Circuits		•	≨ On 1D-Circuits	•
Name		^	Name	^
v			w	
Description		~	Description	V
Destination Folder		V	Destination Folder	v
Primary Node		^	Primary Node	^
Select Object (1)	[ф···	✓ Select Object (1)	$ \Rightarrow \cdots$
Secondary Node		^	Secondary Node	^
✓ Select Object (1)		• ···	✓ Select Object (1)	+
Magnitude		^	Magnitude	^
Method	Harmonic	•	Method	Harmonic 🔹
Electric Current Amplitude	CurrentAmplitude	Α • •	Electric Current Amplitude	CurrentAmplitude A 🔹 👻
Frequency	CurrentFrequency	Hz 🔻 🔻	Frequency	CurrentFrequency Hz 🔹 🖛
Phase Shift	120	• • •	Phase Shift	240 ° • •
Network ID		v	Network ID	v
Card Name CurrentOnCircu	it		Card Name CurrentOnCircu	uit

- 22. Define the environment condition:
 - Create a constraint of type 'Zero Potential Flux tangent' on the two circular edges (outside and inside) shown in the picture.



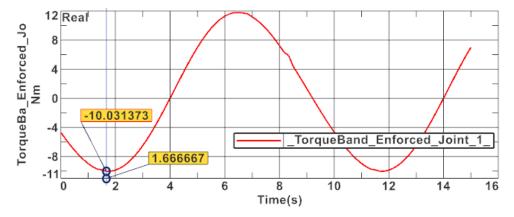
4.1.2 Find the Rotor Start Position for Maximum Torque

The model is now ready to solve, but in a first step we should find the correct rotor position. Normally for synchronous motors, this is the position of maximum torque. So, we have to find this and then set the rotor to that specific angle. One way how to find this is to run the motor without motion and check for torque. Because the rotor moves in negative direction we have to find the max negative value, e.g. the minimum.

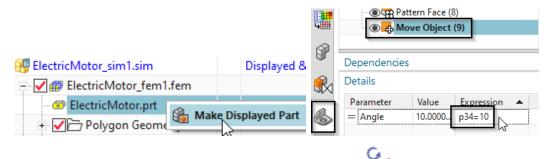
1. Edit the enforced motion joint and set the 'Angular Step' (temporary) to 0. We want to fix the rotor and only turn the currents to find the maximum torque angle.



- 2. Solve the solution. (this takes about 3 minutes because of the nonlinear materials)
- 3. Show the torque result graph and indicate the first minimum.



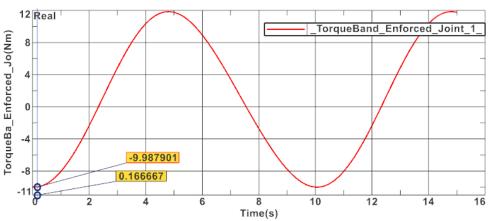
- 4. The torque shows a first minimum at 1.667 sec. Let's transfer this time to the corresponding rotor angle: Because 15 sec represent 90 degrees the minimum is at 1.667 * 90/15 = 10 degrees.
- 5. Change the rotor to minimum torque start position:
 - Set the displayed part to the CAD master 'ElectricMotor.prt'
 - Expression p34 is responsible for the rotor angle. Change this from zero to 10 degrees.



- Change to the Fem part. Press the Update button
- Change to the Sim file.
- 6. If the model does not converge, change the numerical settings to help the solution to converge better. We use a fixed relaxation instead of an adaptively controlled. Therefore, use right mouse button on the solution and 'Edit Solver Parameters'. In register 'Numeric'at box 'Nonlinear Magnetic Material' switch from 'Program Controlled' to 'Advanced Scheme (Default)'. Set the 'Relaxation Type' to 'Fixed Value'.

		Solver Parameters		ა [.]
		Solver		
		MAGNETICS		
		Parameters		
		General Numeric Cluster Parameter Sweep User Defined	Nonlinear Magnetic Material Newton-Raphson Method Max. Number Iterations Absolute Tolerance	Advanced Scheme (Default)
Solution 1 + √ ↓ Simulation Objects	Bedit		Relative Tolerance Convergence Check Error Norm	0.0005 a-Field in whole Domain Mean L2 Norm
T ♥ ➡ Constraints	Edit Solver Parameters		BH-Curve Interpolation Relaxation Type	Akima

7. Rerun the solve and display the torque graph again. Verify that it now starts with minimum torque.



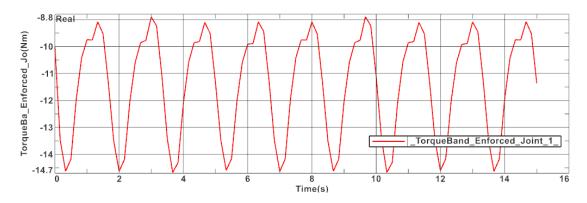
8. Close the graph display.

4.1.3 Analyze for Torque with Rotation

- 1. In the Enforced Motion, set the 'Angular Step' back to expression 'RotorStep'.
- 2. For a higher accuracy result of torque, decrease the tolerance in the Newton scheme: Go to 'Edit Solver Parameters', switch to register 'Numeric' and change the 'Absolute Tolerance' and 'Relative Tolerance' both from 5e-4 (default) to 5e-5.

General	Nonlinear Magnetic Material			
Numeric	Nextee Berkers Mathed	Adversed Coheren (Defer	(0. (
- Cluster	Newton-Raphson Method	Advanced Scheme (Default)		
Parameter Sweep	Max. Number Iterations	30		
- User Defined	Absolute Tolerance	5e-05	•	
	Relative Tolerance	5e-05		

3. Solve the solution and display the torque graph again. It should look similar as the next picture.



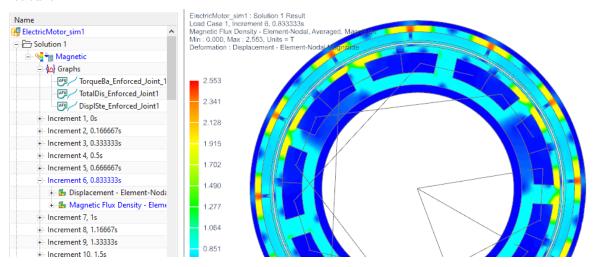
- 4. Close the AFU file.
- 5. Check for torque ripple: Use right mouse button on the torque graph and use the function 'Information'. In the information window there are minimum and maximum values as well as the mean value of the graph shown. The ripple is the difference between min and max.

Information
AFU File Name:\ElectricMotor_sim1-Solution_1PostGraphs.afu
Record Name: TorqueBand_Enforced_Joint_1_Function
Min, Max = -14.6737, -8.89469
RMS = 11.5771 Mean = -11.4119
Standard Deviation = 1.95946

6. Display rotation: To show the motion of the rotor, first display a plot result. In 'Post View', register 'Deformation', activate 'Deformation'.



and then use the green buttons 'Next Iteration' $\Leftarrow \Rightarrow$. Alternatively, use the animation feature. $\Leftarrow \Rightarrow$.



4.1.4 Analyze for Voltage in Phases

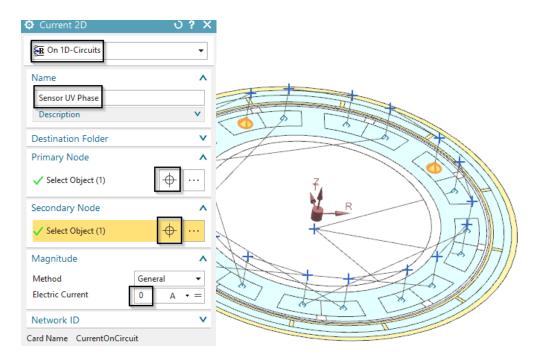
- 1. We now want to run the motor with 1000 turns per minute. So, change the expression 'SpeedMech' to 1000 rev/min. (Menu, Tools, Expressions)
- 2. To analyse for voltage, we want to run the motor with zero currents. So change also the expression 'CurrentAmplitude' to 0 A.

	1 Name	Formula	Value	Units	Dimensionality	
1	Y Default Group					
2				mm 👻	Length 🔹	I
3	CurrentAm	0	0 A	Α 🔻	Electric Current 🔻	I
4	CurrentFr	(Poles*SpeedMech/360)*(180.0/pi())	100 Hz	Hz 🔻	Frequency -	I
5	Poles	6	6		Unitless	I
6	RotorStep	-1	-1 °	• 🗸	Angle 🔻	I
7	SpeedMech	1000	1000 rev/min	rev/min 🔻	Angular Velocity 🔻	I
8	TimeIncre	-1*RotorStep/SpeedMech	0.000166666666	s 🔻	Time 🔻	1

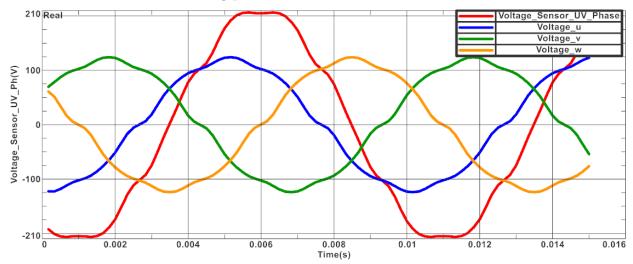
- 3. We do not want to overwrite the existing solution. Thus, clone the solution (right mouse button on solution 1, 'Clone') and rename the new one to 'MagDyn1000Umin'.
- 4. Set the output requests as shown in the picture. The 'Circuit Voltage' request will calculate voltages in all circuit elements.

Solution		0?×	
Solution		∧ Plot	
Name Name Nolver N Analysis Type 2	MagDyn1000Umin MAGNETICS D or axisym Electromagnetics Magnetodynamic Transient	Table Total Force - virtual Total Moment - virtual Total Lorentz Force RotorBand Torque - stresstensor RotorBand Force - stresstensor	
Output Reques Time Steps Tinitial Conditio Coupled Therm 2D	ns Plot Magnetic Fluxdensity	 Fluxlinkage - Vectorpotential on Conductor Electrode Voltage Electrode Current Electrode Power Circuit Voltage Circuit Current Circuit Power Eddy Current Losses Hysteresis Loss - steinmetz Eddy Current Loss - steinmetz Eddy Current Loss - steinmetz Excess Loss - steinmetz Motion Data FFT Tables Motor Efficiency 	s

5. For the measurement of voltage between two phases you create a new current of type 'On 1D-Circuits' with zero amperes between the desired network points. Use for instance the two points U and V. Name this load 'Sensor UV Phase'.

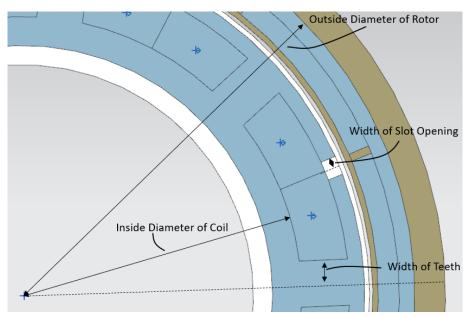


- 6. Solve the new solution
- 7. Display the graph results for the voltages in the three phases and the new sensor. This should look like in the following picture.



4.1.5 Parameter Optimization with HEEDS

In the last part of this tutorial, we intend to optimize the permanent magnet synchronous motor. Therefore, we use the additional program Design Space Exploration, which is a limited version of HEEDS, that can be used only with Simcenter 3D. The goal of this optimization is to maximize the RMS of the Torque and to minimize the Ripple. We thus adjust the Outside Diameter of the Rotor, the Inside Diameter of the Coil, the Width of the Slot Opening and the Width of the Teeth.



Before setting up the Optimization in HEEDS, we need to make sure that all necessary results are available.

- 1. Prepare the Model in NX
 - Download the model files for this tutorial from the following link: https://www.magnetics.de/downloads/Tutorials/6.CouplMotion/6.4ElectricMotor. zip
 - Open the file 'ElectricMotor_sim1.sim'. Hint: You may also use the solution that was created in the previous section.
 - Change to the Part file and set the expression p18 to 220 mm.
 - Switch to the Sim file and set the 'Result Graphs (afu)' within the 'Solver Parameters' to 'Create, keep txt files' and activate 'Solve in Foreground'.

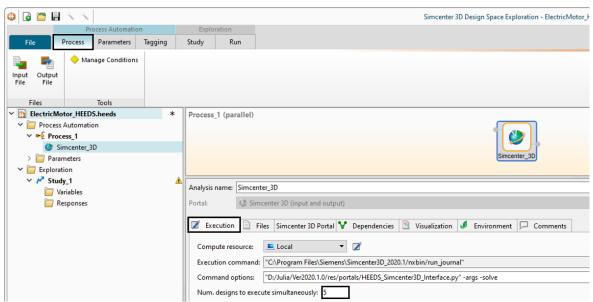
Solver Parameters			ა ?
Solver			
MAGNETICS			
Parameters			
General	Solver Version	1058, Build Date 2022/02/03 12:4	43:06
- Numeric	Description		Lê
Cluster			L+
Parameter Sweep	Result Tables (txt)	Overwrite	•
Parameter Import	Result Graphs (afu)	Create, keep txt Files	•
User Defined	Result Plot File	Create	•
	Result File Type	binary	•
	Logfile Verbosity	1	
	Solution Monitor		
	Keep Solution Mor	nitor	
	Keep Scratch (.pre,	.res)	
	Solve in Foregroun	nd	

- Solve the Solution once to create the txt files.
- Now import three Parameters in the 'Solver Parameters' menu.
- Choose for every Parameter the txt file 'ElectricMotor_sim1-MagDyn1000Umin.RotorTorque.txt' and copy it into the window.
- Set up the Parameter Imports as shown in the picture below.
- Hints: The Torques are shown in the fourth column of the file. As the lines in this case are negative, we set the Line Option for the maximum Torque to the Minimum Value.

		୦ ? ୪
✓ Import Expressions from Result Ta	ables (txt)	
Perform	Three Parameters Import	•
	· ·	
np	Set Solve in Foreground' and 'keep txt Files'	
✓ One Parameter Import		
File containing Parameter	ElectricMotor_sim1-MagDyn1000Umin.RotorTorque.txt	
Position in File - Column Number	4	
Position in File - Line Option	Min Value over Lines	•
Expression Name to create/update	Torque_max	
▼ Two Parameter Import		
File containing Parameter	ElectricMotor_sim1-MagDyn1000Umin.RotorTorque.txt	
Position in File - Column Number	4	
Position in File - Line Option	Max Value over Lines	•
Expression Name to create/update	Torque_min	
File containing Parameter	ElectricMotor_sim1-MagDyn1000Umin.RotorTorque.txt	
Position in File - Column Number	4	
Position in File - Line Option	RMS Value over Lines	•
Expression Name to create/update	RMS_Torque	
	Perform Tip One Parameter Import File containing Parameter Position in File - Column Number Position in File - Line Option Expression Name to create/update Two Parameter Import File containing Parameter Position in File - Column Number Position in File - Line Option Expression Name to create/update Three Parameter Import File containing Parameter Position in File - Column Number Position in File - Line Option	Tip Set 'Solve in Foreground' and 'keep txt Files' • One Parameter Import File containing Parameter File containing Parameter ElectricMotor_sim1-MagDyn1000Umin.RotorTorque.txt Position in File - Column Number 4 Position in File - Line Option Min Value over Lines Expression Name to create/update Torque_max • Two Parameter Import ElectricMotor_sim1-MagDyn1000Umin.RotorTorque.txt Position in File - Column Number 4 Position in File - Column Number Forque_min • Three Parameter Import File containing Parameter File containing Parameter ElectricMotor_sim1-MagDyn1000Umin.RotorTorque.txt Position in File - Column Number 4 Position in File - Column Number 4<

- Solve the Model again and save it before changing to Design Space Exploration (HEEDS).
- 2. Set up the HEEDS run.

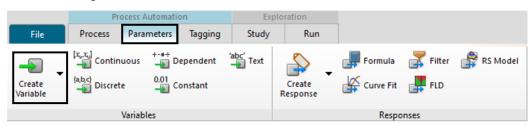
- Open HEEDS and save your HEEDS-project in the same folder, were you also have saved the Sim file. Name it 'ElectricMotor_HEEDS'.
- In order to minimize the solution time, set 'Num. designs to execute simultaneously' to five.



• Input the Sim file which you have prepared in advance. HEEDS will then automatically upload the Fem File, the prt file and also create the output file. After loading, continue without 'Auto Tagging'. Click on 'Process' and 'Input File'.

🙂 🖬 🔚 🔨 🔨								
	Pro	cess Automati	on	Explo	ration			
File	Process	Parameters	Tagging	Study	Run			
Input File	🔶 Man	age Condition	5					
Files		Tools						

• The next step is to create the 'Variables'. Click on 'Parameters' and 'Create Variable'.



- Create the variables as shown in the picture below. HEEDS will adjust these variables to optimize the objectives. In this example we have chosen geometric variables (see below fig.).
- Hint: You always define a minimal value, a maximum value and the baseline value, which is set up in your NX model and defines the first Design, the so called 'Baseline Design'.

📲 Variables 📑 Responses					
√ Variable Name ⁽¹⁾	~ Туре	Min	Baseline	Max	Resolution
1 🗹 语 Outside_Diameter_of_Rotor	Continuous 🔹 🔻	200	204	210	101
2 🗹 🏝 Inside_Diameter_of_Coil	Continuous 🔹	68	72,5	75	101
3 🗹 🏝 Width_of_Teeth	Continuous 🔹	3	6	10	101
4 🗹 🏝 Width_of_Slot_Opening	Continuous 🔻	0,5	2,5	5	101

• The next step is to create the responses. Therefore, click on 'Parameters' and 'Create Responses'.

	Pro	ocess Automa	Exp	Exploration		
File	Process	Parameters	Tagging	Study	Run	
->	$\begin{bmatrix} x_{u}, x_{u} \end{bmatrix}$ Contin	_	Dependent	'abc' Text	₽ .	
Create Variable	{a,b,c}	e 0.01	Constant		Create Response	
	Variables					

- Create the 'Responses' as shown in the picture below.
- Hints: In this example the minimal and the maximal torque is printed out with a negative value. To calculate the torque-ripple we want to use the value of these responses by creating new responses and calculating the values. Then we will use the type 'Formula' and input the calculation. By clicking 'Apply' we add the formula to the response.

- Variables 🕒 Responses]	
য় Response Name স	Source	Properties of Amount_of_Torque_max
	Γag ▼ Γag ▼	Specify the source analysis:
3 🚅 RMS_Torque T	ſag →	Simcenter_3D Set design to ERROR if value is outside:
4 🚮 Amount_of_Torque_max F	ormula 👻	Minimum:
		Maximum
		Data type:
		<pre>sqrt((Torque_max)*(Torque_max))</pre>

- Create a formula also for the ripple. Hint: one response of the solution must be added to generate results. Therefore, the Flux Density is also created as a response.
- Your list of responses should look like the one in the picture below.

🖶 Variables 🖶 Responses								
ম্ Response Name	⊽ ₊ Source	Formula	Comment					
1 📑 Torque_min	Tag 🔻							
2 📑 Torque_max	Tag 👻							
3 📑 RMS_Torque	Tag 👻		1. Objective: Maximize					
4 📑 Amount_of_Torque_max	Formula 🔹	sqrt((Torque_max)*(Torque_max))						
5 📑 Amount_of_Torque_min	Formula 🔹	sqrt((Torque_min)*(Torque_min))						
6 📑 Ripple	Formula 🔹	Amount_of_Torque_max-Amount_of_Torque_min	2. Objective: Minimize					
7 📑 Flux_Density	Tag 👻							

• Now follows the part of 'Tagging'. Here, we will connect the variables and the responses to our Input File and Output File. First, choose the input file by clicking on the button in the picture below.

		Pro	Process Automation						
	File	Process	Parameters	Tagging					
		File:		▼ 7,					
4	-10	Variable:		-					
	Portal	Response:	📑 Flux_Den	sity 🔻 📑					
	Mode								

- Choose the Sim file marked green as input file and 'Tag' the variables with the 'CAD_Expressions'.
- Hints: To 'Tag' a variable choose it from the list of variables (top), click on it in the list of expressions and tag it with clicking on the button 'Tag'.

		Pr	ocess Automati	on	Explora	tion			
	File	Process	Parameters	Tagging	Study	Run			
		File:	ElectricM	otor_sim 🔻	7. 8	🚫 Multi-Tag	🖌 Update	Print format:	
		Variable:	- Outside_[Diameter 🔻				Default	
•	Portal	Response:	- gouside_t		tag ⊒∔	😽 Auto Tag	Remove Ta	#.## Manage Pr	int Format
	Mode					Tagging			
~	ni Elect ✓ Pr	File: D:/Julia	MAGNETICS/	Schulungsunt	erlagen HEEDS/	/ElectricMotor_sim1.s	im		
	✓ ••{	Input Type	[📔 🏣 Part		Expre	ession	Value	
	> 1	CAD_Expres	sions	-> Elect	tricMotor	Autside_Diam	eter 1	78	
	~ 🚞	CAE_Express	sions			Band_Inside_D	iameter 1	80.5	
	>	FEM_Express	sions			Band_Outside	Diameter 1	82.25	
	>					Inside_Diamet	er 1	30	
`	~ 🧧 Ex					Inside_Diamet	er_of_Coil 1	45/2	
	~ ~					Inside_Diamet	er_of_Magnets 1	85	
				r 2		Inside_Diamet	er_of_Rotor 1	96	
						Outside_Diam	eter_of_Rotor 2	04	
						Outside_Radiu	s_of_Coil 1	70/2	
						Width_of_Slot	Openings 5	/2	
						Width_of_Teet	h 1	2/2	

• After tagging every variable the list should look like the one in the picture below.

	Parameter	Mode	Data		
1	- Outside_Diameter_of_Rotor	Portal	CAD_Expressions.ElectricMotor.Outside_Diameter_of_Rotor	Default 👻	
2	- Inside_Diameter_of_Coil	Portal	CAD_Expressions.ElectricMotor.Inside_Diameter_of_Coil	Default 👻	
3	- Width_of_Teeth	Portal	CAD_Expressions.ElectricMotor.Width_of_Teeth	Default 👻	
4	- Width_of_Slot_Opening	Portal	$CAD_Expressions.ElectricMotor.Width_of_Slot_Openings$	Default 🔹	

- Do the same with the responses. Choose the Sim file marked blue as Output File and tag the Torque_max, the Torque_min and the RMS_Torque from the list of 'CAE_Expressions'.
- Choose the Flux_Density in the list of the solution and set the filter to 'Max' to get the maximum value.

Output Type	III ĭ≣	Output		Component	Properties	
CAD_Expressions		Time		х	Filter:	Max 🔻
CAE_Expressions		Displacement - Element-Nodal		γ	C	
FEM_Expressions		Magnetic Flux Density - Element-Nodal	\rightarrow	Z	System:	AbsoluteRectangular 🔹
CAD_MassProperties				Magnitude	Step ID (1-91):	All
CAE_MassProperties						O Specify:
FEM_MassProperties						
Solutions				r 7	FL	0
CircuitVoltage	\rightarrow			2 2 2 2	Element IDs:	
					r + 	O Specify:

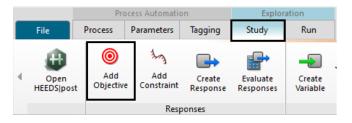
• Afterwards, the list of tagged responses should look like the one in the picture below.

Parameter	Mode	Data		
1 📑 Torque_min	Portal	CAE_Expressions.ElectricMotor_sim1.Torque_min	Default	•
2 📑 Torque_max	Portal	CAE_Expressions.ElectricMotor_sim1.Torque_max	Default	•
3 📑 RMS_Torque	Portal	CAE_Expressions.ElectricMotor_sim1.RMS_Torque	Default	•
4 📑 Flux_Density	Portal	Solutions.CircuitVoltage.Magnetic Flux Density - Element-Nodal.Magnitude (Filter=3 System=AbsoluteRectangular iter=all id=all #System=AbsoluteRectangular)	Default	τ.

- The part of tagging is now done and we will set up a study in the 'Study' tab.
- Choose the 'Multiple objective tradeoff study (Pareto Front)' as 'Hybrid Adaptive Method' and set up the 'Method Properties' as shown in the picture below.
- Hint: We choose a minimum value of the evaluations and the archive size to reduce the time of solving the run to a minimum. For the creation of more designs you can choose a higher number.

	Pro	ocess Automatio	n	Explor	ation			•
File	Process	Parameters	Tagging	Study	Run			0
Open HEEDS/pos	Add Objectiv	Add e Constraint	Create Response	Evaluate Responses	Create Variable	[x,,x,] → Continuous + (a,b,c) → Discrete		Create Study
		Res	onses			Variables		Study
✓ <a>E <a>F <a>Pr <a>F		/ contains setu ie details and		errors before	e running th	he study.		
✓ ••€ ✓ •• ✓ •• ✓ •• ✓ •• ✓ •• ✓ ••	Study name Process: Study type:	► Proce	ess_1 neter Optimi	zation	•	Saved Designs Success designs: Last best desig ▼ Error designs: Keep ▼	Run Options Save restart data after Do not stop HEEDS fo More options	
>		daptive Method	l f all objective	5	_ [Method Properties Max number of evaluations: Archive size:	18 6	

• In the next step we will define the two objectives.



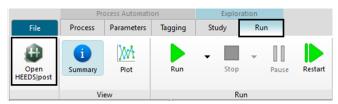
- Choose the response RMS_Torque as first objective and maximize the value. The second objective will be the ripple. Choose it out of the list and set the option to 'Minimize Value'.
- Afterwards it should look like in the picture below.

Study name:	Study_1				Saved Designs	Run Options
Process:	► Proce	ess_1		•	Success designs: Last best desig ▼	Save restart data after each evaluation Do not stop HEEDS for a design-based error
Study type:	n Paran	neter O	ptimization	•	Error designs: Keep 🔹	More options
📝 Methods	x Variable	es 🗳	Responses	Relationships	Comments	
∀_ Available	Responses	7	Туре	Analysis	Objective	Option
🖂 💕 Torque	min	Tag		Simcenter_3D	🗹 🎯 RMS_Torque	Maximize Value 🔻
🖂 💕 Torque		Tag		Simcenter_3D	🖂 🧿 Ripple	Minimize Value 🔻

- The setup of the study in HEEDS is done. Safe your project.
- Run the study by clicking on the button 'Run' in the run tab.

	Pro	ocess Automati	on	Explo	ration
File	Process	Parameters	Tagging	Study	Run
Open HEEDS post	i Summary	Plot	Run	▼ Stop	→ Pause
	Vi	ew		R	un

- It will take HEEDS about 45 min to create the designs.
- We then want to do some post-processing to evaluate the results and to choose the best design. To this end open 'HEEDS post'.



• In order to find the best design, we create a 'Design Table'.

File	Home				
Return to HEEDSjmdo	Refresh Study Data	i Summary	2D Relation Correlation Plot Design Table	Create Plot	•
	Study		Plots		

• Choose all variables, the responses ripple and the RMS_Torque from the list (picture below).

Design Table

Displays the design data in a spreadsheet format

Design Set:	🔍 All Designs	•
Select one or more v	variables and responses to display in the table.	
	Available parameters	7
	🗌 💕 Torque_min	
	🔲 💕 Torque_max	
	🗹 💕 RMS_Torque	
	🔲 💕 Amount_of_Torque_max	
	🔲 💕 Amount_of_Torque_min	
	🗹 📑 Ripple	
	🗌 📑 Flux_Density	
	🗌 💕 performance	
	C Toutside_Diameter_of_Rotor	
	🗹 韇 Inside_Diameter_of_Coil	
	🗹 🏝 Width_of_Teeth	
	✓ ▲ Width_of_Slot_Opening	

- Create the table with 'Finish'.
- Out of the 18 designs created, we choose the one design with the smallest ripple value and the highest RMS_Torque. With design 6 we can reduce the ripple by 25 % and

Study_1: De	signTable_4					
Design ID	RMS_Torque	Ripple	Outside_Diameter_of_Rotor	Inside_Diameter_of_Coil	Width_of_Teeth	Width_of_Slot_Opening
1	11,5613	6,19362	204	72,5	6	2,5
2	15,9914	18,611	209,1	72,48	6,85	5
3	10,2333	1,0946	203,6	69,26	8,11	0,905
4	14,3251	10,1646	210	71,15	7,48	1,715
5	8,98279	3,52498	201,8	68	5,52	3,785
6	11,9023	3,99088	207,3	74,37	4,26	2,12
7	9,36315	2,23324	200,9	73,74	6,15	1,31
8	13,631	10,874	205,5	75	8,74	2,975
9	12,0358	6,78733	206,4	68,63	9,37	4,19
10	11,185	8,66632	202,7	73,11	4,89	4,595
11	8,42361	1,77224	204,5	70,52	3	0,5
12	11,8624	7,24391	208,2	69,89	3,63	3,38
13	11,9988	4,66885	207	74,16	4,4	2,345
14	9,81763	1,95061	202,6	69,68	7,34	0,68
15	9,55337	1,28707	203,3	70,8	4,05	1,04
16	9,26617	3,17615	201,8	68,28	5,52	3,785
17	15,6225	16,4483	210	71,15	7,48	3,965
18	15,7937	17,1555	210	70,94	7,76	4,19

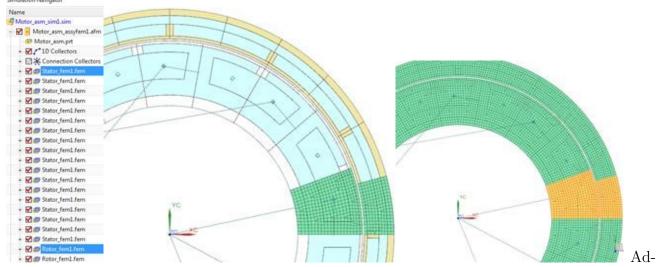
the RMS_Torque increases slightly compared to the Baseline Design. Thus, the motor model definitely is optimized.

The tutorial is completed.

5 Tutorial 4

5.1 AFEM Modeling of Motors

This tutorial shows how to model electric motors using the NX assembly fem method. This method allows to create meshes only for parts of the whole motor and then use those meshes multiple times in an assembly (AFEM). The next picture shows this situation: The file Stator_fem1.fem is created only once, but used several times in the AFEM file Motor_asm_assyfem1.afm.

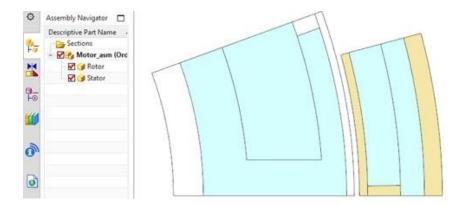


vantages of this method are

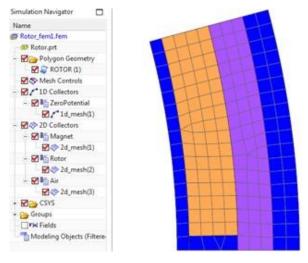
- The overall mesh becomes perfectly regular because the same mesh is used for each segment. Because of this there will also be more precise results.
- Less meshing effort
- Less data volume

You can find the already completed model of the simple motor in this tutorial folder. Go through the following steps to become familiar with the main features:

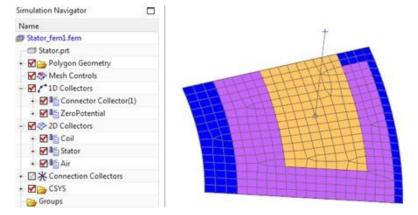
- Download the model files for this tutorial from the following link: https://www.magnetics.de/downloads/Tutorials/6.CouplMotion/6.5ElectricMotorAFEM. zip
- 2. Open the file Motor_asm_sim1.sim.
- 3. Make the CAD assembly part Motor_asm.prt the displayed part.
- 4. Check in the assembly navigator the structure of this CAD assembly. It contains one segment of the rotor and one of the stator.



5. Change the window to the file Rotor_fem1.fem. Check the contents of this file.

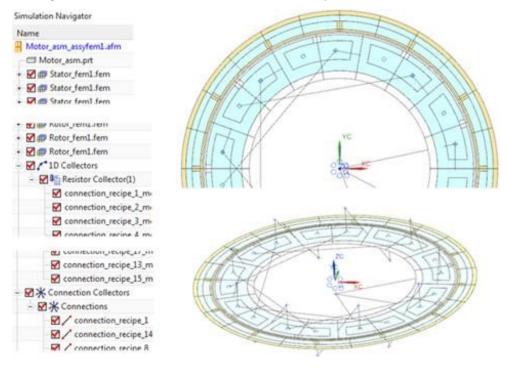


- There is a 1D mesh at the outer border. This is applied to a Physical of type ZeroPotential. Therefore no additional constraint of this type is necessary anymore in the Sim file.
- The Magnet has permanent magnet material and a cylindrical coordinate system as usual. Later in the Sim file we will see that every second magnet physical is overwritten and because of the need for opposing magnet directions.
- 6. Change the window to the file Stator_fem1.fem. Check the contents of this file.

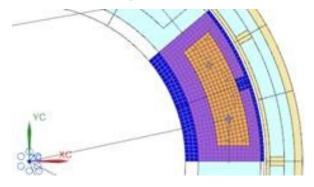


• Again there is a 1D mesh at the inner border with Physical of type ZeroPotential. Therefore no additional constraint of this type is necessary anymore in the Sim file.

- There is a 1D Connector mesh. This is later used to implement the network. The advantage is that this Connector must only be created once.
- 7. Change the window to the file Motor_asm_assyfem1.afm. Check the contents of this file



• Notice that there are 18 Stator_fem1.fem files included in the assembly fem file. For best exploitation of symmetry every second of those meshes is positioned upside down. This method has the advantage of perfectly matching nodes at borders. Next picture shows such a couple of two meshes.



- Notice the 1D resistor elements. They form the circuit network to couple the coil connectors.
- Although nodes at borders are perfectly matching, they must be merged. In this model nodes are already merged, but in a new model you must check for duplicate nodes using the function Duplicate Nodes .
- Like always in NX assembly fem you must use the Assembly Label Manager to resolve conflicts in labeling. Simply press 'Automatically Resolve' in the Assembly Label Manager.
- 8. Change the window to the Sim file Motor_asm_sim1.sim. Check the contents of this file.

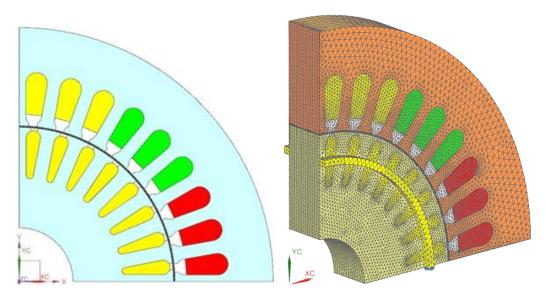
• Check the solutions in the Sim file. There is one static and one transient solution included. Loads and simulation objects are set up in the same way as without assembly fem. There is nothing special for afems now anymore.

The tutorial is complete.

6 Tutorial 5

6.1 Asynchron (Induction) Motor Analysis

In this tutorial an induction or asynchronous motor is analysed. In part one we use a frequency domain analysis in order to provide quick results. Such frequency domain analysis provides settled situations of dynamic problems with harmonic behaviour. However, the frequency domain analysis also has some disadvantages, such as: nonlinear material properties can be simulated only with reduced accuracy; and same-wise also no permanent magnets are possible to be simulated.



In a second part we will analyse the temperatures in the motor. Herein, we will perform a sweep over the full operating range.

In a third part we will use a time domain analysis to solve this problem at one operating point. We will see that the resulting torque agrees nicely to the prior frequency analysis. In time domain we can include accurate nonlinear material effects; also the inclusion of permanent magnets would be possible, but is not needed here. The time domain analysis works quite similar as the prior examples of this document.

After that we want to use time domain analysis to investigate the starting behaviour of the motor. We find that the resulting velocity after some time agrees to the synchronous velocity.

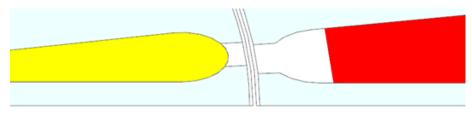
Finally we show a 3D model of the induction motor with simulation in frequency domain. This simulation shows very similar results as the 2D simulation of part one but it would additionally allow to study end effects and other 3D specific effects.

In the frequency domain solutions results come out without rotor motion. Therefore they depend also on the relative position of rotor to stator bars. To check for this influence one could simulate at different rotor angle positions. The 2D CAD model has an expression to control that.

6.1.1 Basic Model Setup

For the basic setup follow these steps:

- Download the model files for this tutorial from the following link: https://www.magnetics.de/downloads/Tutorials/6.CouplMotion/6.6InductionMotor3kW. zip
- 2. Open the file InductionMotor_3kW.prt.
- 3. Notice the air gap in the next picture: There is one layer for the inner mesh, one for the outer mesh and the middle layer is without geometry. Here the solver will create a Moving Band mesh by himself.



- 4. Create a new Sim and Fem file.
- 5. Use solver MAGNETICS, Analysis type 2D and Solution Type 'Magnetodynamics Frequency'.
- 6. Name the solution 'MagDynFreq1'.
- 7. Set the Output Requests as shown below.

Solution			υx	
Solution			^	
Name Solver Analysis Type Solution Type	MagDynFreq1 MAGNETICS 2D or axisym Elect Magnetodynamic	-		Table Ø RotorBand Torque - stresstensor Fluxlinkage - Vectorpotential on Conductors Electrode Voltage Ø Electrode Current Ø Electrode Power
Magnetodyna Output Rec Initial Conc Frequency Coupled Th 2D	litions	Plot Magnetic Fluxdensity Magnetic Fieldstrength Current Density Eddy Current Losses Density Magnetic Potential (a-Pot) Error Estimation Material Properties		 Circuit Voltage Circuit Current Circuit Power Eddy Current Losses Steinmetz Hysteresis Loss Steinmetz Eddy Current Loss Steinmetz Excess Loss Ohm Resistance Coil Inductivity Phase Shift [Efficiency]

- 8. Some remarks to the requested outputs:
 - RotorBand Torque stresstensor: This is the usual result for the appearent torque in the air gap. It is computed by the Maxwell Stresstensor method.

• Electrode Current:

The resulting electric current on voltage loaded faces (electrodes).

• Electrode Power:

This result will compute voltage times electric current on all faces with loads (complex multiplication). The real part of this result is the active power and the imaginary part is the reactive power.

• Eddy Current Losses:

This result will compute power losses that result from eddy currents and ohmic resistance. Also lamination effects will be taken into acount.

• Steinmetz Hysteresis Loss:

Based on the material property Kh, the frequency and the magnetic induction b this result computes losses on all parts with Kh>0 using the Steinmetz formula.

• Steinmetz Eddy Current Loss:

Similar as above, but using the material property Kc and the corresponding Steinmetz formula. This result is alternative to the above Eddy Current Losses.

• Phase Shift:

This result computes the angle between electric current and voltage on each load. This is also known as Power Factor.

• Efficiency:

The Efficiency result computes output power divided by input power. Output power is computed by the resulting torque times rotor speed. Input power is computed by the active power on electrodes or circuits.

9. Set the thickness as shown in the '2D' options.

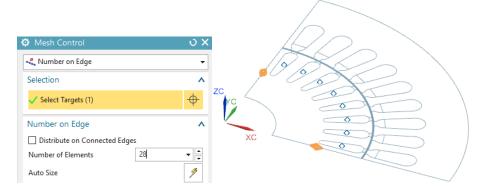
Magnetodynamic Free	quency				^
- Output Requests	Plane	Absolute X,Y			F
- Frequency	Axisymr	netric			-1
- Coupled Thermal	Thickness	127			1
2D	Thickness	127	mm	• •	1

10. And the options for Frequency Domain: The forcing frequency defines the velocity of the rotating field of the coils. 50 Hz divided by the number of poles (4) gives defines Hint: The Setting 'Conductivity Type': 'Slip Dependent' is basis for the used induction motor analysis method.

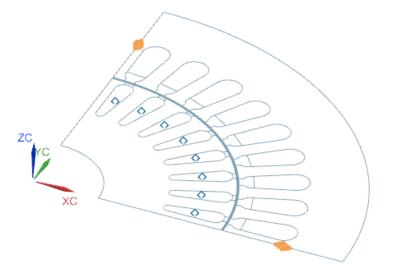
				Forcing Frequence	ies 📢	ა? X
				Modeling Object		v
Magnetodynamic Fre	quency		^	Properties Description		^
···· Output Requests	 Forcing Frequencies 	ForcFreq	- 7 📇 -	Frequency List		^
Initial Conditions	· · · ·	-	332)	Frequency List Form	Individual Frequencie	es 🔻
Frequency	Conductivity Type	Slip Dependend	•	Frequency List	Hz	•
- Coupled Thermal				Frequency List (1)		
2D				50		*

11. Click Ok.

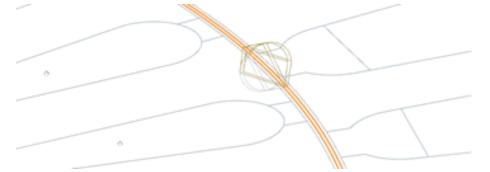
- 12. Switch to the FEM file
- 13. Create Mesh Controls 💎:
 - Assign a number of 28 elements on the shown two linear edges:



• Assign a number of 20 elements on these two edges:



• Assign a number of 180 elements on the two edges of the Moving Band.



14. Mesh the inner part of the air gap (element size 0.5 mm). Create a FluidPhysical and assign 'Air' from the Magnetics material library as shown in the picture.

Mesh Collector		૨ ×	
Properties		^	
Physical Property		^	
Туре	FluidPhysical	-	<i>[</i>]// /
🗸 PlanePhysical	Air_Rotor	- & 🗞 -	
Name	Air_Rotor		
			A//
	0	K Cancel	B//
		K	
			A// /
		A	\$//~/
		- Al	

15. Mesh the outer part of the air gap (0.5 mm) and also use FluidPhysical and 'Air'.

Physical Property Table	^
Name Air_Stator	
Label 19	
Properties	^
Material Air	- (
ОК	Cancel
OK	Cancer

16. Mesh the coil faces (1 mm) for phase U and use the shown settings.

PlanePhysical		υx
Physical Property Table		^
Name	PhaseU	
Label	4	
Properties		^
Material	Copper 👻	¢.
Conductor Model		^
Model	Stranded	•
Number of Turns	102.0000	=
Fillfactor	1	•
Rigid Body Motion		×
CSYS		V
	OK Can	cel

17. Mesh the coil faces for phase V and use these settings.

PlanePhysical		υx				
Physical Property Table		^			_	
Name	PhaseV		1990 A			
Label	5					
Properties		^			\frown	
Material	Copper 🗸	¢	THE X	9	$/ \cap$	
Conductor Model		^	0/0//	\mathcal{P}	$\left(\right)$	\sim $^{\prime}$
Model	Stranded	•		$/ \land$	\swarrow)
Number of Turns	102. <mark>0000</mark>	=		^/	X	\sim
Fillfactor	1	•			\sim	\bigcirc
Rigid Body Motion		×		~ <		
CSYS		×	YC	Ć	0	L
	OK Cano	cel	ZC XC	0	•	\square

18. Mesh the coil faces for phase W and use these settings.

PlanePhysical		υx
Physical Property Table		^
Name	PhaseW	
Label	6	
Properties		^
Material	Copper	-
Conductor Model		^
Model	Stranded	•
Number of Turns	102.0000	=
Fillfactor	1	•
Rigid Body Motion		v
CSYS		×
	ОК	Cancel

19. Mesh the bars (1 mm). Put them all in one physical or even in one mesh. This will simulate the effect of connections at top and bottom. So, in this case there is no need for an additional circuit network to couple the bars. Use the shown settings. Create a new material for the bars. Name it 'RotorBarsAlu'. Use the properties as shown and described following.

PlanePhysical		υx
Physical Property Table		^
Name	BARs	
Label	11	
Properties		^
Material	RotorBarsAlu	-
Conductor Model		^
Model	Massive	-
Rigid Body Motion		~
CSYS		~
	ОК	Cancel

- 20. Some words about the properties of material Aluminum_Sample1:
 - Mass Density (RHO): 7.7e-6 Kg/mm3. This property must be given if transient thermal is coupled with the electromagnetic solution.
 - Electromagnetics: Relative Permeability (mur): 1 With mur = 1 the material has the same magnetic characteristic as air.
 - Electromagnetics: Electric Conductivity (sigma) : 25380710 S/m. This property is very important for induction effects. The appearance of eddy currents strongly depends on this.
 - Thermal/Electrical: Thermal Conductivity (K): 117 W/m K Basic characteristic for thermal conduction effects. This property must be applied if thermal analysis is coupled with electromagnetics.
 - Thermal/Electrical: Specific Heat (CP): 900 J/Kg K The CP property is responsible for transient thermal effects. It must be applied if any transient thermal is coupled with electromagnetics.
- 21. The next picture shows the material dialog of Aluminum_Sample1.

Properties				^
Mass Density (RHO) 0			kg/mm³	Ð
Mechanical	Magnetic			^
Strength	Permeability	Linear		
Durability	Permeability	Linear		
Formability	relative Permeability (mur)	1		Ð
Thermal				
Electromagnetic	Permanent Magnet			^
Creep	Remanent Magnetic Fieldstrength X	0	A/m	Ð
 Viscoelasticity 	Demagnetization Fieldstrength		A/m	A
···· Viscoplasticity	2 cm agnetization r relasti engli		Aviii	<u> </u>
Damage	Electric			~
- Other Physical Properties				
Miscellaneous	relative Permittivity (epsr)	1		₽
Electromagnetic MAGNETICS	Conductivity (sigma) [S/m]	2.67e7		Ð

Mechanical	Thermal			~
- Strength	Temperature (TREF)		**	Δ
Durability	Temperature (Ther)		°C	0
Formability	Thermal Expansion Coefficient (A)		°C ⁻¹	⋳
Thermal	Thermal Conductivity (K)	117000	µW/(mm·°C)	Ð
- Electromagnetic	Specific Heat (CP)	9e8	μ/(kg·K)	A
- Creep			1002.0	

22. Mesh the rotor (1 mm) and use the shown settings. Use the library material 'ElectroSheet_Sample1' from the Magnetics library. Notice the setting of the 'Conductor Model'. The 'Laminated' model simulates the behaviour of eddy currents and the corresponding magnetic field in thin laminated sheets. The larger the value for 'Thickness of one Sheet' the more eddy currents and losses will appear.

PlanePhysical	ა x
Physical Property Table	^
Name	Rotor
Label	7
Properties	^
Material	ElectroSheet_Sample1 👻 🔇
Conductor Model	٨
Model	Laminated 👻
Fillfactor	1 🔹
Thickness of one Sheet	0.3 mm • •
Rigid Body Motion	v
CSYS	v
	OK Cancel

- 23. Some words about the used material properties in 'ElectroSheet_Sample1':
 - Mass Density (RHO): Needed only for transient thermal effects.
 - Electromagnetics: Relative Magnetic Permeability (mur): 1500 This property is used in case of frequency domain analysis or if there is no BH curve given.
 - Electromagnetics: Electric Conductivity (sigma): 5800000 S/m
 - Electromagnetics: Core Loss Hysteresis Coefficient (Kh): 650 W/m^3 . This value must be given by measurements or by the material supplier.
 - Electromagnetics: Core Loss Eddy Current Coefficient (Kc): $1.2W/m^3$. This value can be found by test simulations or is also given by the material supplier.
 - Thermal: Thermal Conductivity (K): 35 W/m C
 - Thermal: Specific Heat (CP): 460 J/Kg K
- 24. Mesh the stator (2 mm) and use these settings:

PlanePhysical	ა x		
Physical Property Table	٨		***
Name	Stator		
Label	8		
Properties	^		
Material	ElectroSheet_Sample1 👻 🔇		
Conductor Model	^		
Model	Laminated 🔹		\mathcal{T}
Fillfactor	1 •		$\langle \rangle$
Thickness of one Sheet	0.3 mm • •		//
Rigid Body Motion	v		
CSYS	v	YC	_
	OK Cancel	ZC XC	\leq

25. All necessary work in the Fem file is done. Switch the displayed part to the Sim file.

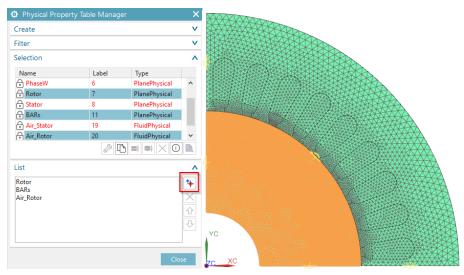
6.1.2 Simulation at Fixed Speed

In this section we will use a frequency domain solution to find motor characteristics at one operating point or rotor speed.

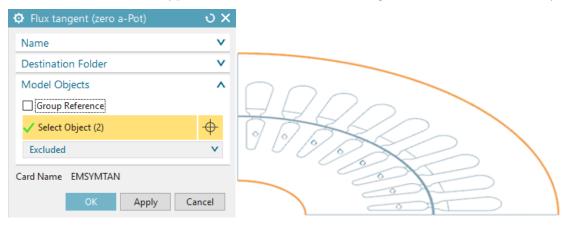
- 1. Create a simulation object 'Enforced Motion 2D Frequency':
 - Select the 'Airgap Rotor Edge' and 'Airgap Stator Edge'.
 - As a preparation for the following parameter sweep, instead of using a fixed value for the 'Angular Velocity', you may easily create an expression named 'vel' and couple this with the value of 800 rev/min. You do this by simply writing vel=800 into the 'Angular Velocity' field. Later this expression can be used in a parameter sweep.
 - Key in 4 in the field 'Number of Poles Total'.
 - Select 'Air' in the material field in box 'Airgap, auto meshed'.

Difference Motion 2D Free	quency ປ 🗙		
Name	×		
Destination Folder	~		
Airgap Rotor Edge	^	Rotor Geometry	Â
Group Reference		Select Physicals (3)	
✔ Select Object (1)	+	Definition	^
Airgap Stator Edge		Rotation Axis	Z
Group Reference		Stator Shift X0	0 mm • •
 Select Object (1) 		Driver	Angular Velocity
		Velocity	vel=800 rev/min • •
Airgap, auto meshed	^	Periodicity	
Number of Air Gaps	1 -	Number Poles in Model	1
Material	Air 👻 🕼	Number Poles Total	
Rotor Geometry		Tolerance Factor for Links	1e-05 -
Select Physicals (3)	I	Card Name MBEnfRevolute	Frequency

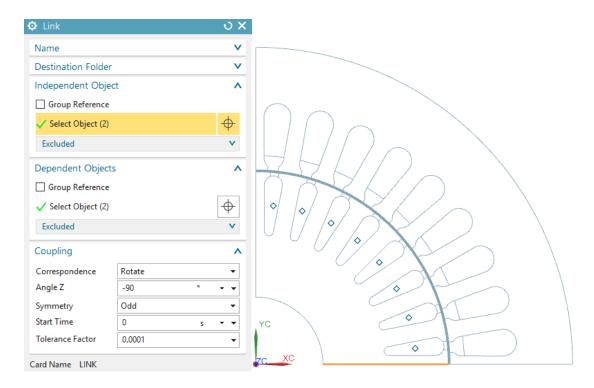
• In 'Rotor Geometry' select the three Rotor Parts, hit the 'Add' button and 'Close'.



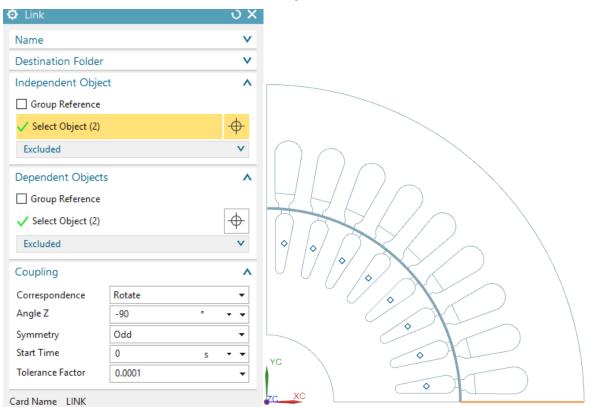
- Click OK and the simulation object is created.
- 2. Create a constraint of type 'Zero Potential Flux tangent' on the two boundary faces:



3. For periodicity conditions create two constraints of type 'Link'. The selection should be done in mathematical positive direction, e.g. against the clock sense. So the 'Independent Object' must be the highlighted (orange) edge in the next picture. The 'Dependent Object' will then be the corresponding edge (vertical), rotated about 90 degrees in positive direction. Use the settings as shown. Select also the small edges of the air gap.



4. Create a second Link constraint for the edges of the stator:



5. Create the following 'Voltage Harmonic 2D' loads: Hint: We apply only 1/4th of the full voltage load because the model contains only 1/4th of the full motor. So the formula we use for the voltage is: 220 V * sqrt(2) /4. Here 220 is the effective voltage. Don't forget the phase shifts in the following three dialogs.

• For phase U:

Voltage Harmonic 2D		ა x
\sim On Physical		-
Name		^
U		
Description		~
Destination Folder		v
Primary Direction +Z		^
Туре	PlanePhysical	-
 Select Physical 	🔒 PhaseU	- 🌽 😻 🗸
Back Direction -Z (option	nal)	v
Magnitude		^
Electric Voltage Amplitude	-220*sqrt(2)/4	V • •
Phase Shift	0	• • •
Card Name VoltageFreqOnF	hysical2D	
Card Name Voltagerregon	nysical2D	

• For phase V (phase shift -120):

Voltage Harmonic 2D		ა x
\sim On Physical		•
Name		^
V		
Description		~
Destination Folder		v
Primary Direction +Z		^
Туре	PlanePhysical	•
Select Physical	🔒 PhaseV	- & 😻 🔻
Back Direction -Z (option	onal)	v
Magnitude		^
Electric Voltage Amplitude	-220*sqrt(2)/4	V • •
Phase Shift	-120	° • •
Card Name VoltageFreqOr	Physical2D	

• For phase W (positive value, phase shift -240):

Voltage Harmonic 2D		ບ X
\sim On Physical		•
Name		^
W		
Description		~
Destination Folder		v
Primary Direction +Z		^
Туре	PlanePhysical	-
 Select Physical 	🔒 PhaseW	- 🌽 😻 🗕
Back Direction -Z (option	nal)	v
Magnitude		^
Electric Voltage Amplitude	220*sqrt(2)/4	V • •
Phase Shift	-240	• • •
Card Name VoltageFreqOnP	hysical2D	

• Create one additional voltage load on the rotor bars. Name this load 'FixedVoltage on free Bars' and apply zero voltage.

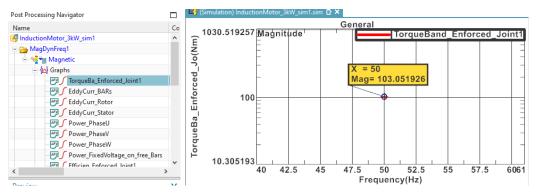
Voltage Harmonic 2D		ა x
\sim On Physical		•
Name		^
FixedVoltage on free Bars		
Description		v
Destination Folder		~
Primary Direction +Z		^
Туре	PlanePhysical	•
 Select Physical 	🔒 BARs	- & 😻 🔻
Back Direction -Z (optio	nal)	~
Magnitude		^
Electric Voltage Amplitude	0	V • •
Phase Shift	0	• •
Card Name VoltageFreqOnl	hysical2D	

6. Solve the solution.

Hints: In some cases there may appear an error message (see picture) saying that one of the link constraints didn't find its corresponding nodes. This message tells you that there are different numbers of nodes on both edges. You can fix this problem by adjusting (probably one element more or one less) the mesh control on one of the edges.

File Edit Window Help	a	×
nductionMotor_3kW_sim1-MagDynFreq1.log		
Solve		•
<pre>Info : Started (Thu Apr 02 13:24:26 2015, CPU = 0s, Mem = 4.19922Mb) P r e - P r o c e s s 1 n g Error : Constraint Link: bad correspondance of number of Nodes (181, 180)</pre>		
End		
•	,	

- 7. The solution will give the result for an operating point with 800 rev/min. To find results for other velocities change the velocity in the motion joint and solve again or run a parameter sweep as shown in the next section 'Sweep over Rotor Speed'.
 - The resulting torque is 103 Nm. It can be found in the Post Processing Navigator at 'Graphs', see picture below.



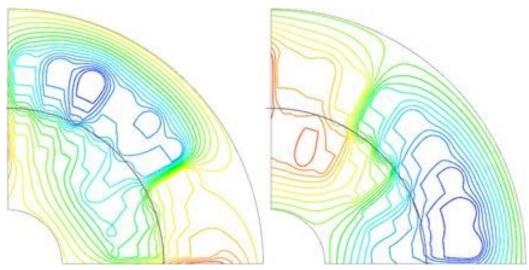
- Magnetic Flux Density (also called 'Induction'): Use the 'Set Result' button to choose between Amplitude, Phase or Real and Imaginary part.
- Hints to complex results: Because this is a frequency domain solution, all loads and results are assumed to be harmonic. Therefore all results are complex, described by real and imaginary parts. Amplitude results (the geometric addition of real and imaginary part) represent the peak values of such harmonics. Other interesting results can also be extracted from the complex results.

Name	
InductionMotor_3kW_sim1	InductionMotor 38 V_sim1 : MagDynFreq1 Result Load Case 1, Frequency 1, 50Hz
⊡ 🗁 MagDynFreq1	Magnetic Flux Density - Element-Nodal, Unaveraged, Magnitude Complex Option : Amplitude
🗄 🗞 🧤 Magnetic	Min : 0.009, Max : 5.856, Units = T
	5.856
TorqueBa_Enforced_Joint1	5.369
	4.882
🕀 🏪 Current Density (Area) - Elemen	4.395
🗄 🏪 Magnetic Flux Density - Elemen	
🛃 Imported Results	3.907
Viewports	3.420
Contour Plots	2.933
+ 🏟 Post View 1	2445
Templates	2.440
	1.958
	1.471
	0.983
	Y A
	0.496
	X+-e0.0
< >>	

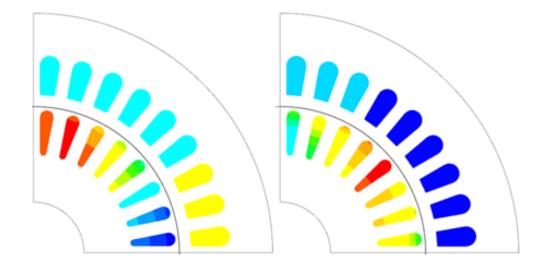
• To study animations over time set the option 'Complex' to 'At Phase Angle' and run the 'Animation'.

	Banded Plot	X		
	Frequency 1, 50 Hz 💌			
	Magnetic Flux Density - Element-Nodal 💌			
	Magnitude 👻			
B	Complex At Phase Angle • 0.000		K	

• The Vector potential z direction result: (left: Real part, right: Imaginary part). This result shows the magnetic field lines.



• The Current Density result: (left: Real part, right: Imaginary part)



6.1.3 Sweep over Rotor Speed

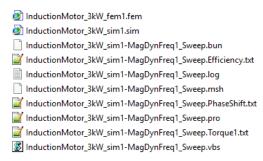
Now we will use a parameter sweep to step over the whole speed range. By this way we will find the characteristic curves describing the motor.

- 1. Clone the Solution 'MagDynFreq1' and rename the Clone 'MagDynFreq1_Sweep'. Open the 'Solver Parameters' of solution MagDynFreq1_Sweep and change to register 'Par.Sweep, SolveAgain'.
- 2. Set the 'Perform' option to 'One Parameter Sweep' and set the settings as shown.

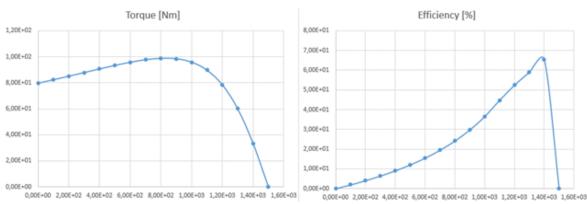
Solver Parameters			υ×	
Solver			^	
MAGNETICS				
Parameters			^	
General	Parameter Sweep / Solve A	Again	^	
Numeric Par.Sweep, SolveAgain	Perform	One Parameter Sweep	•	D
User Defined	Expression One to Sweep	Emerging Original Surger		🗁 Solver Sets 🐂 MagDynFreq1
osci Denned	Expression One to Sweep		^	+ 💭 Simulation Objects
	Location of Expression	Sim Part	-	🕀 🏣 Constraints
	Expression Name	vel		🛨 🕂 Loads
	Expression Name	VEI		🛨 🗁 Results
	Start	0	▼ 5	MagDynFreq1_Sweep
	Stop	1500	•	
				Enviced Joint(1)
	Step	100	-	 FluxTangent(1)
	0.10.1		^	- 🗸 😐 Link(1)
	Post Processing	Post Processing		
	Keep all AFU Files			E V Loads
	Keep all Plot Files			🗸 😐 U
				√ ● V
	Complex Option	Magnitude	•	 W FixedVoltage on free Ba
	Open Charts in Excel			+ P Results

- 3. Some words about the settings:
 - Accept the default 'Location' for the expression: 'Sim Part' because the expression 'vel' resides in the Sim part.
 - Expression Name: This must be an existing expression. To check for expressions you can use Tools, Expressions.

- Start, Stop and Step: This defines the sequence of simulation steps.
- 'Complec Option': This setting allows the selection or either 'Magnitude' or the real, im parts of the complex result.
- 'Open Charts in Excel': The parameter sweep results will be written into a vbscript file that can be pushed to MS Excel. Excel must be installed on the computer for that.
- 4. Solve the solution. Notice the progress bar at the bottom of the NX window. It shows the percentage progress of the steps. Also notice the vbscript (vbs) file in the working folder. It contains all requested tabular results.

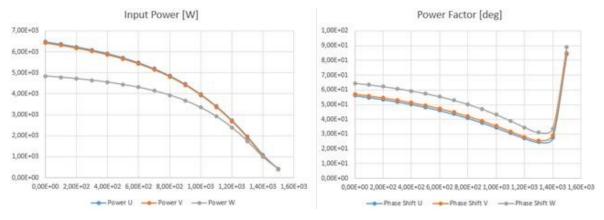


5. After solve has finished Excel starts and shows the requested results as curves over the given parameter.

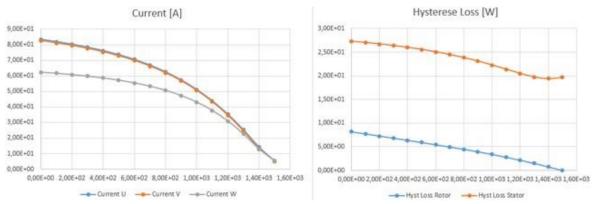


• Torque and Efficiency over speed:





• Current and Hysteresis Loss over speed. Be aware that the computed hysteresis loss in frequency domain is not very accurate, because only one frequency is taken into consideration. The later time domain analysis will show higher hysteresis losses.



6.1.4 Sweep with two Parameters

In many cases it is of interest sweeping over two parameters. For this motor we want to know for instance how torque and efficiency changes with the speed but also with the electric conductivity of the bars. The result will be shown as a surface graph in excel.

- 1. Clone the Solution 'MagDynFreq1_Sweep' and rename the Clone 'MagDynFreq1_TwoSweep'. Open the 'Solver Parameters' of solution MagDynFreq_Sweep and change to register 'Par.Sweep, SolveAgain'.
- 2. Set the 'Perform' option to 'Two Parameters Sweep' and set the settings as shown. There will be a parameter 'CondScale' in the Fem part. This parameter will scale the conductivity of the bars.

Solver Parameters			υ×
Solver			^
MAGNETICS			
Parameters			^
General	Parameter Sweep / Solve A	Again	^
Numeric Par.Sweep, SolveAgain	Perform	Two Parameters Sweep	•
User Defined	Expression One to Sweep		^
	Location of Expression	Sim Part	-
	Expression Name	vel	
	Start	0	-
	Stop	1500	•
	Step	100	-
	Expression Two to Sweep		^
	Location of Expression	Fem Part	-
	Expression Name	CondScale	
	Start	0.6	-
	Stop	1.4	-
	Step	0.2	•
	Post Processing		^
	Keep all AFU Files		
	Keep all Plot Files		
	Complex Option	Magnitude	-
	Open Charts in Excel		

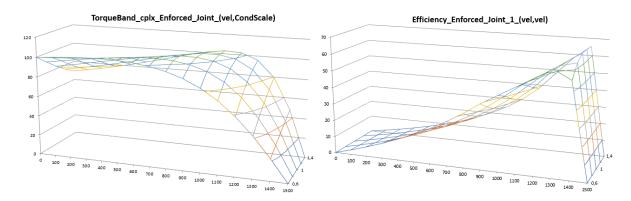
3. Change to the Fem part and create an expression (shortcut Strg+e). Name this 'CondScale' and set it to 'Unitless'.

ø	O Expressions						
		1 Name	Formula	Value	Units	Dimensionality	Туре
	1	Y Default Group					
	2	CondScale	1	1		Unitless 🔹 💌	Number
	-		0 -				

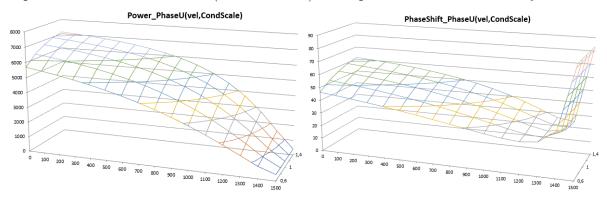
4. Edit the material properties of the bars and key in 'CondScale*2.67e7' for the electric conductivity.

1 2		
- Damage	Electric	
Other Physical Properties		
- Miscellaneous	relative Permittivity (epsr)	1
Electromagnetic MAGNETICS	Conductivity (sigma) [S/m]	CondScale*2.67e7

- 5. Change the displayed part to the Sim file and solve the new solution. The resulting graphs will look as shown below.
- 6. Torque and Efficiency over speed and bar conductivity:



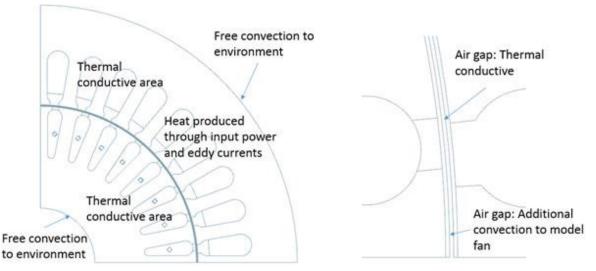
7. Input Power and Phase Shift (Power Factor) over speed and bar conductivity:



6.1.5 Thermal Analysis

In this section, we will include simple 2D thermal analysis methods for the induction motor. In more detailed simulations this must be done in 3D to capture end effects and the motor housing. In this example, we will directly use the power losses from the electromagnetic analysis as input loads for a thermal solve. We include the effects of thermal conduction in all parts. Also, we include thermal convection effects with given fixed coefficients to model water cooling. The air gap is modeled as a thermal conductive area, so heat produced in the rotor can travel through the air gap into the stator. To capture the effect of cooling by a fan we add additional convection to the air gap.

In this case we use a set of given convection coefficients. Of course, these values strongly influence the thermal results of the simulation. To find realistic values for convection coefficients simulations should be calibrated with experiments. Another way to find values is through analytical considerations by use of convection formulas that model shear flow between two moving plates. Such formulas can be found in standard literature like 'VDI Wärmeatlas'.



We first run an analysis at one operating point and afterwards a sweep over the full range. We choose a speed of 1400 rev/min because of its best efficiency as we have found in the previous section. Since the material properties already have all information for this thermal analysis we can continue with solution settings. Remember the necessary material data that is needed for this is only Thermal Conductivity (K). In case of transient analysis the Thermal Capacity (CP) would be needed.

- 1. Clone the solution MagDynFreq1_Sweep and rename it to MagDynFreq1_Thermal_Sweep.
- 2. First switch off the parameter sweep (in 'Solver Parameters'). We will activate it later.
- 3. Edit the expression vel and set it to 1400 rev/min (Tools, Expressions ...).
- 4. Edit the solution and change to register Coupled Thermal.
 - Change the Thermal Solution to Steady State. This will compute the settled thermal situation while running at the given operating point.
 - Activate the Output Requests as shown.

Output Requests	Thermal Solution	Steady State	•
Frequency	Initial Condition		/
Coupled Thermal	D.C. H.T.		
- 2D	Default Temperature	20	°C••
	Output Requests, Plot	1	
	✓ Temperature		
	Temperature Gradi	ent	
	Temperature Cond	uctive Flux	
	Output Requests, Tabl	e	1
	Temperature Maxin	mum	
	Temperature Minir		

- 5. We want to model the effect of a water cooled outside stator face.
 - Create a constraint of type 'EM Thermal Constraints' and set the Type to 'Free Convection'.
 - Select outside edge.

• Key in the values as shown below. Notice the unit of the convection coefficient: W/m2 C.

EM Thermal Constraints	ა x
🚀 Free Convection	-
Name	v
Destination Folder	V
Model Objects	^
Group Reference	
< Select Object (1)	+
Excluded	×
Properties	^
Ambient Temperature 20	°C
Convection Coefficient 5000 W/(m ² ·	°C) - =
Card Name EMConvection	

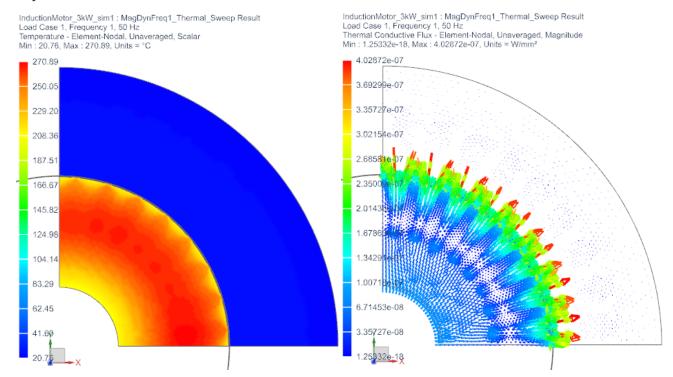
- 6. Similar we model a cooling inside the rotor with smaller coefficient.
 - Create a constraint of type EM Thermal Constraints and set the Type to Free Convection.
 - Select inside edge.
 - Key in the values as shown below

EM Thermal Constra	ints	ა x
🚀 Free Convection		•
Name		v
Destination Folder		v
Model Objects		^
Group Reference		
< Select Object (1)		+
Excluded		~
Properties		^
Ambient Temperature	20	°C • •
Convection Coefficient	400 W/(m ² ·°C)	- =
Card Name EMConvectio	n	

7. Next create a Free Convection constraint on the rotor edges. This will simulate the effect of cooling air from a fan flowing through the air gap.

EM Thermal Construction	aints	ა x
🚀 Free Convection		•
Name		v
Destination Folder		V
Model Objects		^
Group Reference		
🗸 Select Object (16)		+
Excluded		~
Properties		^
Ambient Temperature	20	°C • •
Convection Coefficient	1000 W/(m ² .°C)	- =
ard Name EMConvection	n	

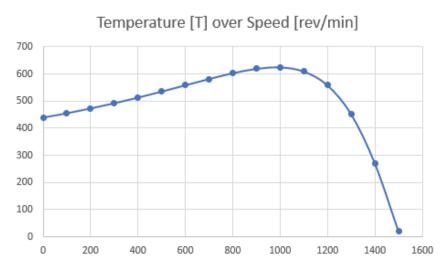
- 8. Switch off the parameter sweep (if not already done) and solve the solution.
- 9. The resulting temperature field at 1400 rev/min has a maximum value of 270 C as shown in the next picture (left side). On the right side, there is the thermal conductive flux displayed with vectors. The thermal flux result helps understanding how the thermal energy is produced and how it moves.



10. Activate the one parameter sweep. Because in this software version the thermal sweep results are not written to the excel file we must view them in text file. Therefore we set the solver parameter 'Result Tables (txt)' to 'Append'. Solve the solution. After finish there appear two additional results (maximum and minimum temperature) as text files.

🔚 InductionMotor_3kW_sim1-MagDynFreq1_Thermal_Sweep.TempMax.txt 🔀							
1	TempMax_Unit_C:	ThermalRegion:	0	439.8023940652081			
2	TempMax_Unit_C:	ThermalRegion:	0	455.8457067658979			
3	TempMax Unit C:	ThermalRegion:	0	473.4293889484283			
4	TempMax_Unit_C:	ThermalRegion:	0	492.6128765987489			
5	TempMax_Unit_C:	ThermalRegion:	0	513.3756689640212			
6	TempMax_Unit_C:	ThermalRegion:	0	535.5532495316708			
7	TempMax_Unit_C:	ThermalRegion:	0	558.6757263245247			
8	TempMax_Unit_C:	ThermalRegion:	0	581.7901774075916			
9	TempMax_Unit_C:	ThermalRegion:	0	603.0779086201599			
10	TempMax_Unit_C:	ThermalRegion:	0	619.2456065884987			
11	TempMax_Unit_C:	ThermalRegion:	0	624.5251846792693			
12	TempMax_Unit_C:	ThermalRegion:	0	609.2750884987115			
13	TempMax Unit C:	ThermalRegion:	0	558.4672222399635			
14	TempMax_Unit_C:	ThermalRegion:	0	451.8323471079671			
15	TempMax Unit C:	ThermalRegion:	0	270.8942707537752			
16	TempMax Unit C:	ThermalRegion:	0	20.22403008602081			
17							

11. In an excel graph this gives the information about how temperature behaves over the speed range:



6.1.6 Transient Run at Fixed Operating Point

Now we do a precise solve for the detailed computation of losses. Because of the transient solution it is possible also to accurately compute a nonlinear BH curve. Because of simplicity reasons we don't use nonlinear material for this and we also don't analyse for temperature. We choose 1400 rev/min as operating point.

- 1. Create a new solution of type 2D Magnetodynamic, Transient. Rename the solution 'Mag-DynTime1_Speed1400'.
- 2. Set the Output Requests as shown.

				Plot	
				Table	
				Total Force - virtual	
				Total Moment - virtual	
				Total Lorentz Force	
Solution			υx	RotorBand Torque - stresstensor	
Solution			^	RotorBand Force - stresstensor	
Name	MagDynTrar	ns1_Speed1400		Fluxlinkage - Vectorpotential on Conductors	
Solver	MAGNETICS			Electrode Voltage	
				Electrode Current Electrode Power	
Analysis Type		n Electromagnetics			
Solution Type	Magnetodyr	namic Transient		Circuit Voltage	
Magnetodyna	mic Transient	t	^	Circuit Current	
···· Output Req	worte	Plot	^	Circuit Power	
Time Steps	00303		~	Eddy Current Losses	
Initial Cond	itions	Magnetic Fluxdensity		Steinmetz Hysteresis Loss	
Coupled Th	ermal	Magnetic Fieldstrength		Steinmetz Eddy Current Loss	
2D		Current Density Eddy Current Losses Density		Steinmetz Excess Loss	
		Magnetic Potential (a-Pot)		✓ Motion Data	
		Nodal Force - virtual		FFT Tables	
		Nodal Moment - virtual			
		Lorentz Force		NVH Coupling	
		✓ Displacement		4D Fields	
		Material Properties		Restart Data	
		Table	~	More	

- 3. Some words to table results:
 - Motion Data: This option writes out the displacement step and total displacement of the motion joint at each time step. That is quite important, because it allowes creating graphs over the rotation angle.

Dist

- FFT Tables: This option enables the fourier transformation of the active table results. In box More there are some options about the fourier transformation process but the defaults are good for now.
- 4. In register Time Steps:
 - Set the Time Increment to 0.000119048 sec. With a rotor speed of 1400 rev/min this corresponds to 1 degree per time step.
 - Set the Number of Time Steps to 1800. This corresponds to 5 times 360. So we will analyze 5 rotor turns. This should be enough to decay the transient that we expect due to sudden start.

gnetodynamic Trans	sient	
Output Requests	Time Step Option	Constant 🗸
Time Steps	Time Increment	0.000119048
Initial Conditions		
Coupled Thermal	End Time Option	Number of Time Steps 🔹
- 2D	Number of Time Steps	1800

5. In register 'Initial Conditions': Set the 'Magnetic' option to 'Zero Vector Potential'. This is the best starting condition for this voltage driven system.

Magnetodynamic Trans	ient					
Output Requests	Magnetic	Zero Vector Pote	ntial 🔹			
Time Steps	Initial Time	0	c v v			
Initial Conditions		Ŭ	, .			
Coupled Thermal	 Output rel. Magnetic Permeability (mur) in Nonlinear Domain from 					
2D	General Motior	• • •	v			

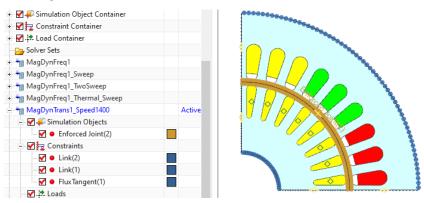
6. In register 'Coupled Thermal': Use the default option None. Because this simulation will cover only a short time period there are no meaningful thermal results expected.

Magnetodynamic Transie	nt		^
···· Output Requests			
Time Steps	Thermal Solution	None (Init Materials only)	•
Initial Conditions	Initial Condition		^
Coupled Thermal			
2D	Default Temperature	20	°C

7. In register '2D': Key in the thickness.

Magnetodynamic Transier	nt		A
···· Output Requests	Plane	Absolute X,Y	•
Time Steps	Axisymme	etric	
Initial Conditions	Thickness		
- Coupled Thermal	Inickness	127	mm 🔻 🔻
2D			

8. Drag all but the thermal constraints into the new solution. You can simply reuse the existing ones.



9. The loads must be created newly because they are different from the frequency solution loads. Create 3 new loads on the corresponding coil faces. Use type 'Voltage 2D', Type 'on Physical' with Method 'Harmonic'. Take care of the correct signs of the values (see picture below).

Ø Voltage 2D	ა	? X	Voltage 2D	ν)	? X	Voltage 2D		ა? X
🞇 On Physical		•	🗃 On Physical		•	On Physical		
Name		^	Name		^	Name		^
U2			V2		_	W2		
Description		~	Description		~	Description		v
Destination Folde	er	×	Destination Folder		v	Destination Fold	er	~
Primary Direction	1 +Z	^	Primary Direction +Z		^	Primary Direction	n +Z	^
Туре	PlanePhysical	•	Туре	PlanePhysical	•	Туре	PlanePhysical	
 Select Physical 	🕞 PhaseU 🔻 🌽 🔯	•	Select Physical	🔒 PhaseV 👻 🌽 🔯	•	Select Physical	-	- & 😻 🗸
Back Direction -Z	(optional)	×	Back Direction -Z (option	nal)	v	Back Direction -Z	(optional)	v
Magnitude		^	Magnitude		^	Magnitude		^
Method	Harmonic	•	Method	Harmonic	•	Method	Harmonic	•
Electric Voltage	-220*sqrt(2)/4 V	• •	Electric Voltage Amplitude	-220*sqrt(2)/4 V	• •	Electric Voltage	220*sqrt(2)/4	V
Frequency	50 Hz	• •	Frequency	50 Hz	• •	Frequency	50	Hz 🔹 💌
Phase Shift	0 •	• •	Phase Shift	-120 °	• •	Phase Shift	-240	• • •
Card Name Voltage	OnPhysical2D		Card Name VoltageOnPhysi	cal2D		Card Name Voltage	On Physical 2D	

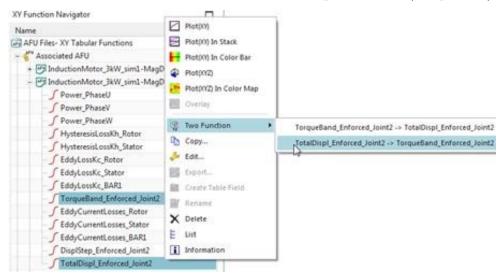
10. Also the load with zero voltage on the Bars must be newly created.

Voltage 2D		ບ x	Name	000000
			InductionMotor_3kW_sim1.sim	and the second s
🞇 On Physical		_	■ Maintent Motor_3kW_fem1.fem	
Chi Physical		•	CSYS	
Name		^	· Croups	
			Fields	
FixedVoltage on free B	ars 2		Modeling Objects	
Description		×	Regions Grad Action Object Container	
Description			+ M = Simulation Object Container	
Destination Falsion		N/	Constraint Container	
Destination Folder		•	Solver Sets	
Primary Direction +Z		^	HagDynFreq1	
			+ MagDynFreq1_Sweep	
Туре	PlanePhysical	•	MagDynFreq1_TwoSweep	
🗸 Select Physical	A BARs	- L to -	MagDynFreq1_Thermal_Sweep	
V Select Physical	0 DAIG	• • • •	🚽 📲 MagDynTrans1	
			💿 🖅 🚑 Simulation Objects	
Back Direction -Z (op	otional)	~	• • • • • • • • • • • • •	
Magnitude		^	- 🗹 🛤 Loads	- Silver - Silver
wagnitude		X	🗹 😐 U2	
Method	General	•	• 🗹 😐 V2	
			• W2	
Electric Voltage	0	v - =	 FixedVoltage on free Bars 2 	
			🗄 👝 Results	
Card Name VoltageOnP	hysical2D		< >	
3				

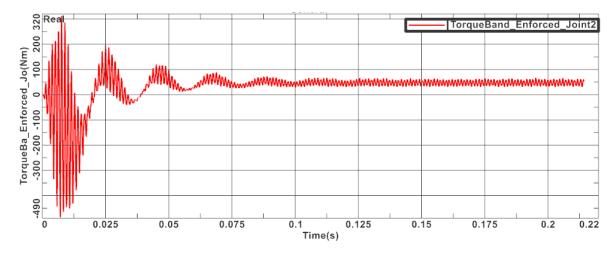
- 11. Create a 'Simulation Object' of type 'Enforced Motion 2D'. Accept the default type 'Revolute by Moving Band'.
 - Select the Rotor and Stator Edges,
 - At 'Airgap, auto meshed', choose 'Air',
 - For 'Rotor Geometry', select the physicals Rotor, BARs, AirRotor.
 - Set the 'Driver' to 'Angular Velocity',
 - Key in 'Velocity', 'Number of Poles total' (see picture below).
 - Click OK to finish the dialogue.

Enforced Motion 2D	_ย x		
🚯 Revolute by Moving Ban	d 👻		
Name	~	Rotor Geometry	^
Destination Folder	v	Moving Parts (3)	III
Airgap Rotor Edge	^	Definition	^
Group Reference	_	Rotation Axis	Ζ -
🗸 Select Object (1)	•	Stator Shift X0	0 mm - -
Airgap Stator Edge		Driver	Angular Velocity 🔻
Group Reference		Velocity Tolerance Factor for Links	1400 rev/min • • 1e-05 •
🗸 Select Object (1)	+	Periodicity	
Airgap, auto meshed	^	Number Poles in Model	1
Number of Air Gaps	1	Number Poles Total	4
Material	Air 👻 🕼	Card Name MBEnfRevolute	_

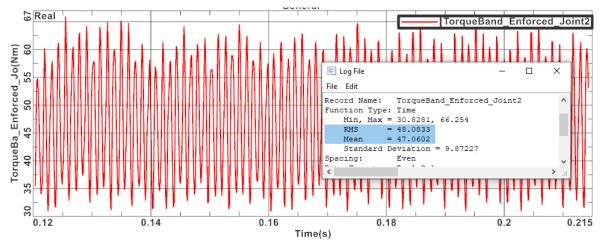
- 12. Solve the solution. Because of the number of steps the solve process will take about 10 15 minutes.
- 13. After the solution has finished check the tabular results.
 - Hint: By default the AFU graphs show time on the abscissa. Instead you can set the rotor angle (or any other result) to the abscissa using the following method: Select in the XY Navigator two curves, Torque and TotalDispl. Then choose RMB Two Function and choose the second from the two possibilities (see picture).



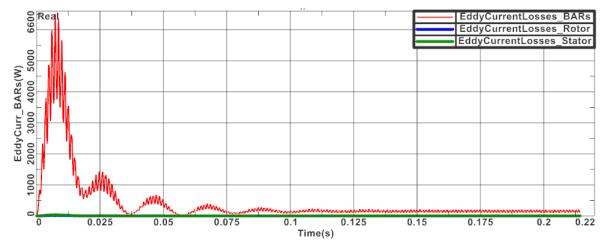
• The Torque result shows transient behaviour at the beginning and becomes more and more stationary after time.



• Let's compare this to the torque we found with the prior frequency domain analysis: There we had 47.7 Nm at 1400 rev/min. Here, if we cut the transient as in picture below, there appears an oscillation between about 30 and 60 Nm with RMS 48 Nm. So, both time and frequency results are very near. The time domain analysis is more precise and additionally it captures torque ripple information.



• The Eddy Current losses on bars, rotor and stator also shows the transient, mainly on the bars. Other results, like power on the thress phases are also available.



• The Hysteresis Losses on Rotor and Stator are computed after all time steps have finished. This additional post processing step can consume much time. For the com-

putation Fourier transformations are performed at each element to find dominant frequencies and amplitudes. By default the system computes 15 frequencies. Let's compare again these values to those from the frequency domain analysis: in time domain, we find 77 watt on the stator and 5.6 watt on the rotor. The frequency analysis showed much smaller values and is less precise, because it takes into account only the first, dominant frequency.

 InductionMotor_3kW_sim1-MagDynTrans1_Speed1400.HystLossKh.txt I

 HysteresisLossKh__Unit_W:
 Rotor:
 5.633220708178191
 Stator:
 77.3782574594274
 BARs:
 0.2230170578517194

6.1.7 Start of Motor

It is also of interest to analyse the transient start behaviour of an induction motor. To do so we have already applied nearly all information. Two things are missing: The mass inertia property of the rotor and a dynamic joint instead of the enforced one.

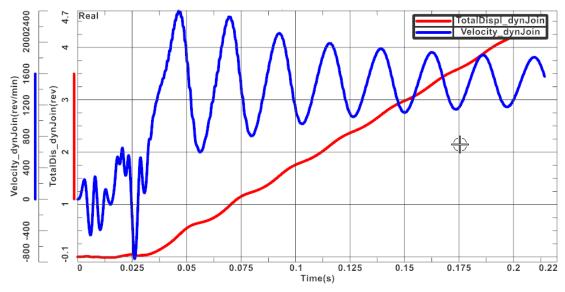
1. Change to the Fem part and edit the physical properties of the rotor. Key in the value 1500 Kg mm2 for Inertia RZ. This value can be found by running the CAD function Anal-

ysis, Measure Body using the detailed CAD geometry and mass density information applied to the rotor.

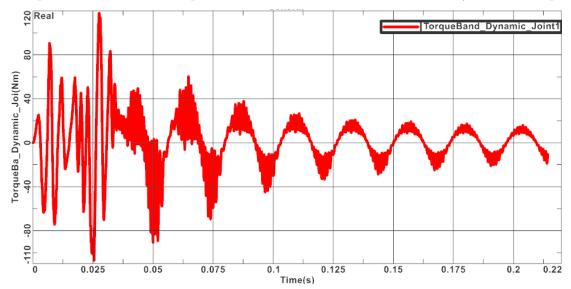
			Dynamic Motion 2D		ა x
			🔹 Revolute by Moving Ban	d	•
			Name		v
			Destination Folder		v
			Airgap Rotor Edge		v
			Airgap Stator Edge		v
PlanePhysical	ს ?	×	Airgap, auto meshed		v
Physical Property Table		^	Rotor Geometry		^
Name	Rotor		Moving Parts (3)		
Label	7		Definition		^
Properties		^	Rotation Axis	Z	•
Material	ElectroSheet_Sample1 -	¢	Stator Shift X0	0	mm • •
Conductor Model	L	^	Tolerance Factor for Links	1e-05	•
Model	Laminated	•	Periodicity		^
Fillfactor	1	-	Number Poles in Model	1	
Thickness of one Sheet	0.3 mm •	•	Number Poles Total	4	
Rigid Body Motion		^			
Mass	kg 👻	-	Initial Conditions		v
Inertia RZ	1500 kg·mm ² •	-	Spring, Damper		V
CSYS		v	More		v
013		1	Card Name MBDynRevolute		

2. Change to the Sim file again and

- 3. Clone the solution 'MagDynTime1' and rename the new one to 'MagDynTime2_MotorStart'.
- 4. Remove the Simulation Object 'Enforced Joint' from the new solution.
- 5. Create a new Simulation object of Type 'Dynamic Motion 2D' (see picture above right). Select Rotor and Stator Edge, key in 'Number of Poles' and 'Rotor Geometry' same way as before.
- 6. Modify the voltage load to 50 percent.
- 7. Solve the solution. Again it takes 10 15 minutes.
- 8. After solving you can display the following graphs:
 - Total angular displacement (red curve) and Velocity (blue curve) over time (set the units to rev and rev/min). It can be seen that after the transient there appears still oscillation but the velocity reaches more and more the synchronous value of 1500 rev/min.

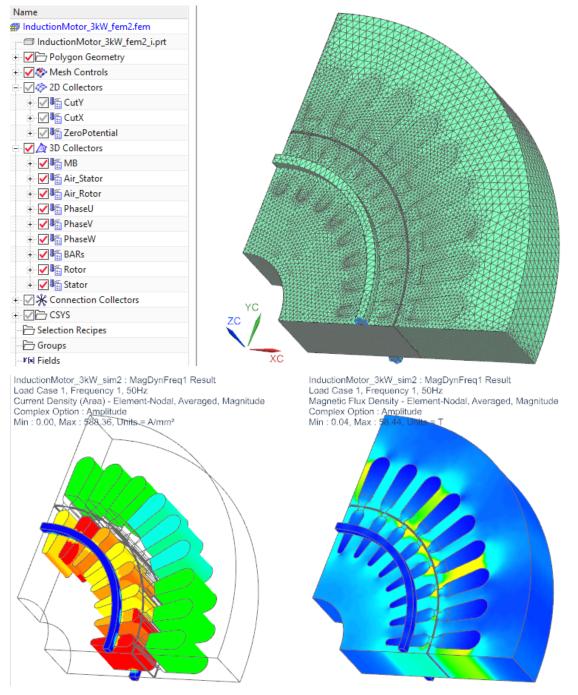


• Torque over time. The torque becomes smaller the more we reach synchronous speed.



6.1.8 3D Model of Induction Motor

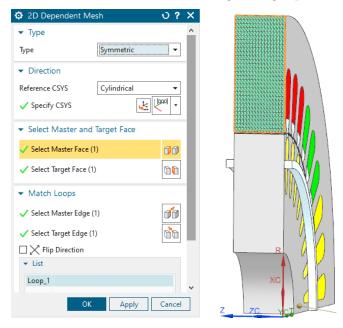
In the complete folder of the tutorial there is also a 3D model of the motor that is set up pretty much the same as the 2D model from the beginning. This 3D model would additionally allow to study end effects and other 3D specific effects. The thickness is reduced to 12.7 mm. So, if results shall be compared to the 2D model, this must be set to the same thickness. All used features can be found in the model files.



- 1. Open the file 'InductionMotor_3kW_sim2.sim' from folder 'complete'.
- 2. First check the solution 'MagDynFreq1'. It is from type 'Magnetdynamic Frequency' with

50Hz. For 'Output Requests' we are interested in the RotorBand-Torque, the 'Current Density' and the 'Magnetic Fluxdensity'.

- 3. Switch to the fem file.
- 4. Notice that there are three 2D mesh collectors. The 'CutY' and the 'CutX' collector, that contain the meshes of the cutted faces and the 'ZeroPotential' collector.
- 5. Open the 'CutY' collector to see the meshes. Every 'CutY' mesh has a corresponding mesh in the 'CutX' collector. Those are created by the '2D Dependend' feature.
- 6. Click 'Edit' on 'CutX4' to see the settings of this mesh. For 'Master Face' the upper face is selected, the program automatically selects the corresonding 'Target Face' on the other side of the motor. It also selects the 'Master' and Target Edges'. It is important, that the arrows of the 'Master' and 'Target Edge' point in the same direction.



- 7. Check the 3D collectors. We use the same physical properties as in the 2D simulation.
- 8. Now switch to the sim file.
- 9. Check the 'Simulation Object' in the solution.
- 10. It is from type 'Enforced Motion 3D Frequency', for 'Moving Parts' the 'Rotor Air', the 'Bars' and the 'Rotor' are selected.
- 11. For 'Airgap 1 Ring' the 'Moving Band' is selected, and the 'Angular Velocity' is set to 800 rev/min.

Enforced Motion 3D F	requency	୦ ?	×				
Name			^				
Destination Folder							
 Rotor Geometry 							
Moving Parts (3)				Physical	Property Tab	le Manager	?
 Airgap 				▼ Create			
2.				Туре	SolidPhysic	al	•
Number of Air Gaps	1	-		Name	SolidPhysic	al1	
Туре	FluidPhysical	•		Label	18		
🗸 Airgap 1 Ring	🔒 MB::Induct 👻 🔩	≸ - -					Create
Airgap 1 outer Radius	45.8447481773	mm • •					
Airgap 1 inner Radius	45.6881141449	mm • •		 Filter Selection 			
				Name		Label	Туре
Axis	Z				ctionMotor		FluidPhysi 🔨
Driver	Angular Velocity				r::Induction		FluidPhysi
Angular Velocity	800 rev/	min 🝷 👻		<	::Induction	9	FluidPhysi ¥
 Periodicity 					S		× () 🖻
Number Poles in Model	1			✓ List			
Number Poles Total	4					_3kW_fem2::[9] ①
ard Name MBEnfRevolute	Frequency		~		ionMotor_3kV tionMotor_3k\		\times
	ОК	Cancel					Close

12. Check the Constaint, 'Link(1)'. For 'Independent' the 'CutY' physical is selected, for 'Dependent' the 'CutX'. The 'Coupling' is simular to the 2D link.

🕽 Link		0?×
Name		
Destination Folder		
 Linked Regions 		
Selection Type	Physical Property Table	•
Туре	BoundaryConditionSet	•
🗸 Independent	CutY::InductionMotor_3kW_fem2::[2]	- 😻 - 🗕
Туре	BoundaryConditionSet	•
🗸 Dependent	CutX::InductionMotor_3kW_fem2::[3]	- 😻
 Coupling 		
Correspondence	Rotate	•
Angle X	0	° • •
Angle Y	0	° • •
Angle Z	-90	° • •
Symmetry	Antiperiodic (Odd)	•
Start Time	0	s • •
Tolerance Factor for Links	0.0001	•
ard Name LINK		
		OK Cancel

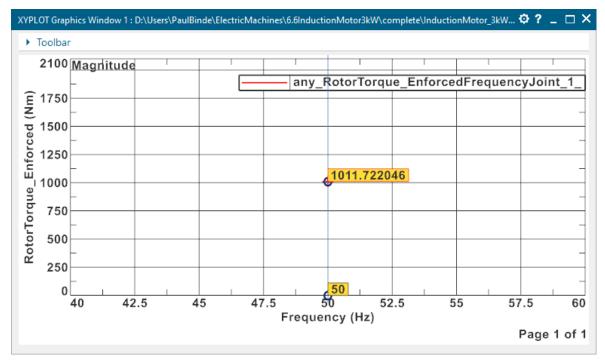
- 13. Open the 'Loads'.
- 14. Now have a look at the loads. Load 'U', 'V' and 'W' are shifted by -120° . They have a

amplitude of 77.7817 V.

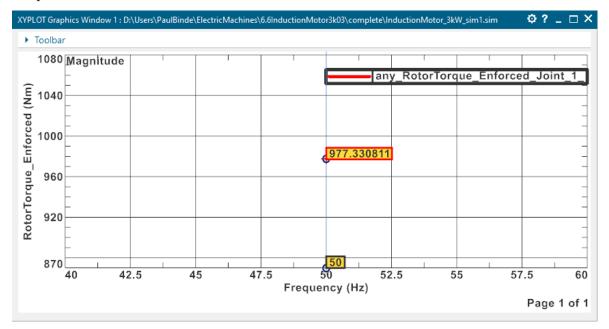
15. Now solve the solution.

6.1.9 Post Processing

1. Open the AFU-Graph of the 'RotorTorque_Enforced'.



2. To compare this picture shows the torque of the 2D model with the same thickness. The torque has almost the same value.

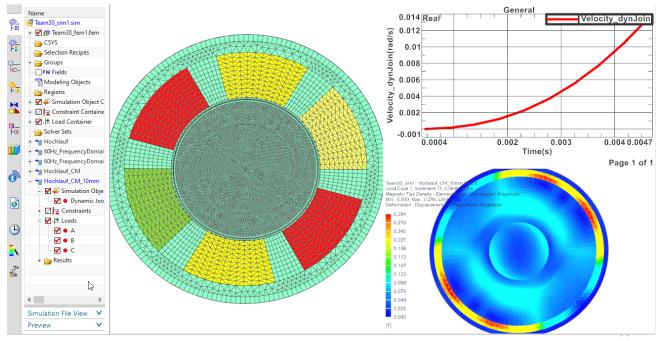


The tutorial is complete.

7 Tutorial 6

7.1 Motor (Team30) with Moving Band/Constant Rotor Shape (Conductor Motion)

In some cases there is motion, but the geometry remains constant. This is the case for instance if the rotor is a simple tube as in the following example. In such situations the simulation is much simpler, because all movements can be handled solver internally, thus we can use the feature 'Conductor Motion const Shape'. For comparison we first solve this model with a moving band and then again with conductor motion.



The following tutorial shows such an application. Example is the induction motor 'TEAM30' that is one of the Compumag reference models.

- 1. Download the model files for this tutorial from the following link: https://www.magnetics.de/downloads/Tutorials/6.CouplMotion/6.1InductionMotorTeam30.zip
- 2. Open the file Team30.prt.
- 3. Create a new Sim and Fem file(Non-Manifold) from 'Magnetics' Toolbar.
- 4. Use solver MAGNETICS, Analysis type '2D or axisym Electromagnetics' and Solution Type 'Magnetodynamic Transient'.
- 5. Name the solution 'Hochlauf' (e.g. start-up).
- 6. Activate the desired data in Output Requests (here we activate 'Rotorband Torque')

Output Requests Time Steps Initial Conditions Coupled Thermal Coupled Motion 2D	 Plot Magnetic Fluxdensity Magnetic Fieldstrength Electric Fluxdensity Electric Fieldstrength Current Density Eddy Current Losses Density Magnetic Potential (a-Pot) Nodal Force - entire (virtual) Nodal Moment - entire (virtual) Lorentz Force (j x b) 	 Table Total Force - entire (virtual) Total Moment - entire (virtual) Total Lorentz Force RotorBand Torque - stresstensor RotorBand Force - stresstensor Fluxlinkage - Vectorpotential on Conductors Electrode Voltage Electrode Current Electrode Power

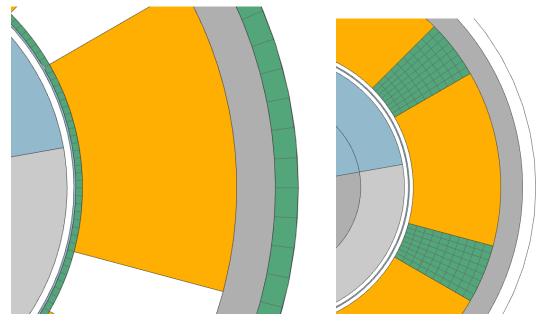
- 7. In time steps, set the 'Time Increment' to 1/2160 and the 'Number of Time Steps' to 10.
- 8. Set the Electromagnetic fields as shown in the 'Initial Conditions' options.

Output Requests	Electromagnetic fields	Zero	Vector Poten	tial 👻	-
Time Steps	Initial Time	0		s + +	Ī
 Initial Conditions 				-	
 Coupled Thermal 	Solve Electromagnetic Solution				
- Coupled Motion	✓ Output Step 0				
2D	Full Wave (HighFreq) Extension				
	Zero Current on unloaded Conductor	s			
	Magnetic Permeabililty (mur) in Nonlinear Domain from File				
		ОК	Apply	Cancel	

- 9. Click OK.
- 10. Switch to the FEM file.
- 11. Mesh the inner part of the air gap (quads, 0.75mm) and use FluidPhysical and 'Air'.

 Properties 		
 Physical Propert 	у	
Туре	FluidPhysical	•
🗸 PlanePhysical	Air_In	- B -
Name	Air_In	

12. The outer air consists of 3 different quad meshes. Mesh the inner circle with 1mm and the outer circle with 4mm. Mesh the gaps between the coils with a 2D Mapped Mesh and the Element Size of 2mm. Assign the fluid Physical and select the material 'Air'.



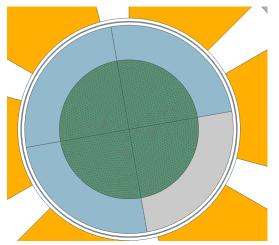
13. Mesh the Stator with tri meshes and an element size of 2mm.

Mesh Collector	ა? ×
Properties	
✓ Physical Property	
Type PlanePhysical	
✓ PlanePhysical Stator	 ✓ ✓
Name Stator	
Oł	Cancel

14. Edit the physicals and create a new Material with the name 'Team_30_Steel_Non_Con'. In 'View' select All Properties and add the electromagnetic properties as seen in the picture below.

Mechanical	 MAGNETICS Solver EM Properties 	
Strength	lefe shout the used Medel	The University of Mardel (shows) is also as with a set of the
- Durability	Info about the used Model The 'Low Frequency' Model (above) is also possib	
- Formability	Electric Conductivity Type	Constant/Table
- Thermal	Electric Conductivity (sigma) [S/m]	0
Electromagnetic	Electric relative Permittivity (epsr)	1
- Creep	Manager and State Trans	
 Viscoelasticity 	Magnetic Permeability Type	Linear, mur
Viscoplasticity	Magnetic relative Permeability (mur)	30
- Damage	Magnetic Remanent Fieldstrength X (hr)	0 A/m 🕂
 Other Physical Properties 	Magnetic Demagnetization Fieldstrength	A/m 🗟
Pedestrian Protection	, , ,	

15. Mesh the Rotor_Steel with tri meshes and an element size of 0.75mm.



- 16. Edit the physicals and create a new Material with the name 'Team_30_RotorSteel'
- 17. In 'View' select 'All Properties' and add the electromagnetic properties as seen in the picture below.

Mechanical	 MAGNETICS Solver EM Properties 	
Strength	Info about the used Model	The 'Low Frequency' Model (above) is also possible and preferred
Durability Formability	Electric Conductivity Type	Constant/Table
- Thermal	Electric Conductivity (sigma) [S/m]	1.6e6 🕞
Electromagnetic	Electric relative Permittivity (epsr)	1
Creep Viscoelasticity	Magnetic Permeability Type	Linear, mur
Viscoplasticity	Magnetic relative Permeability (mur)	30
- Damage	Magnetic Remanent Fieldstrength X (hr)	0 A/m 🕀
 Other Physical Properties 	Magnetic Demagnetization Fieldstrength	A/m 🕞
Pedestrian Protection		

18. Mesh the Rotor_Alu (tri, 0.75mm) and choose material Aluminum: 3.8e7 Siemens/meter. Set the Inertia RZ to $4688 kgmm^2$

	PlanePhysical	ა? X
	 Physical Property Table 	^
	Name	Rotor_Steel
	Label	6
	 Properties 	
	Material	Rotor_Steel 👻 🔦
	Material CSYS	Absolute 🔻
	▼ Electromagnetic Solutions	
20-20-	Active	
	Conductor Model	Massive 👻
	EM Shape Functions	Second Order 👻
	Thermal Solutions	
	Elasticity Solutions	
	▼ Motion Solutions	
	Active	
	Mass	kg 🔹 👻
	Inertia RZ	4688 kg·mm² • •
		OK Cancel

19. Mesh all the coils (tri, 0.75mm) and for each coil set the physical as shown in the picture below.

PlanePhysical		ა? X
 Physical Property Table 	E Constanting of the second	
Name	A_Pos	
Label	10	
✓ Properties		
Material	Copper simple	▼
Material CSYS	Absolute	•
▼ Electromagnetic Solution	15	
Active		
Conductor Model	Stranded	•
Number of Turns	1	•
Fill Factor	1	-
Skin/Proximity-Effect in	Wire	
EM Shape Functions	Second Order	-

20. Mesh the Airgap (tri, 1.69mm) and choose fluid and Air.



- 21. We are done with the FEM file.
- 22. Change the window to the Sim file.

7.1.1 Simulation with Moving Band

- 1. In this section we will use a Moving band to allow the rotor to move.
- 2. Create a simulation object 'Enforced Motion 2D' and select type 'Revolute by Moving Band'. Select the 'Airgap Rotor Edge' and the 'Airgap Stator Edge'. Use the same setting as seen in the picture below.

Enforced Motion 2D		ა? x	3				
Revolute by Moving Ban	d	•					
▶ Name							
Destination Folder							
✓ Airgap Rotor Edge							
Group Reference							
Body Focus							
✓ Select Object (1)		ф····					
✓ Airgap Stator Edge							
Group Reference							
Body Focus			Physical Property Tab	ole Ma	anager	?	×
✓ Select Object (1)		\oplus …	Create				^
 Airgap, auto meshed 			▶ Filter				
Number of Air Gaps	1	•	✓ Selection				
Material	Air	- (Name	La	Туре		
			🔒 Stator::Team30_fem	3	PlanePhysical	^	
 Rotor Geometry 			Rotor_Steel::Team30	4	PlanePhysical		
Moving Parts (3)			Rotor_Alu::Team30_f	5	PlanePhysical		
		1	A A PostTeam30 fem	6	PlanePhysical	×	
 Definition 			59	ß		D 🖻	
Rotation Axis	Z	•	▼ List				
Stator Shift X0	0	mm 🝷 🔻	Rotor_Steel::Team30_fem1:	:[4]		\oplus	
Driver	Angular Step	•	Rotor_Alu::Team30_fem1::				
Angular Step	1	• • •	Air_In::Team30_fem1::[12]				~
Tolerance Factor for Links	1e-05	•				Close	

3. Create another Simulation object, 'Deactivation Set' and select the airgap in 'Deactivated Physicals':

	4	Physical Property Tak	ole Manager		?	×
		Create				^
Deactivation Set) ? X	 Filter 				
▼ Name		 Selection 				
Deactivation Set(1)		Name	Label	Туре		
Description		Air_Out::Team30_fe	16	FluidPhysi 🖌	~	
		Air_Gap_OnOff::Tea	17	FluidPhysi		
 Destination Folder 				•	1	
Simulation Object Container Root 👻	ľ	< 7		> × ()) ,	
 Deactivated Physicals 		11.1				
Select Physicals (1)		 List Air_Gap_OnOff::Team30_fe 	em1::[17]		€	
Card Name DeactivationSet				5	/	۷
ОК Са	ancel			Clos	se	

- 4. Create a Constraint of type 'Flux Tanget' and select the outside edge of 'Air_Out'.
- 5. Lastly we create the Loads.
- 6. Create a Current and select the settings as below.

Current 2D	ა? X					
▶ Type						
Name						
Destination Folder						
✓ Primary Direction +Z						
Туре	PlanePhysical 👻					
 Select Physical 	🕞 A_Pos::Team30_fi 🔻 😻 🔻					
▼ Back Direction -Z (opt	tional)					
Туре	PlanePhysical 👻					
 Select Physical 	🕞 A_Neg::Team30_f 👻 🐳 🔻					
✓ Magnitude						
Method	Harmonic (cos) 🔹					
Electric Current Amplitude	2892.3142 A 🔹 👻					
Frequency	60 Hz 🔹					
Phase Shift	0 ° • •					
Card Name CurrentOnPhysi	cal2D					
	OK Cancel					

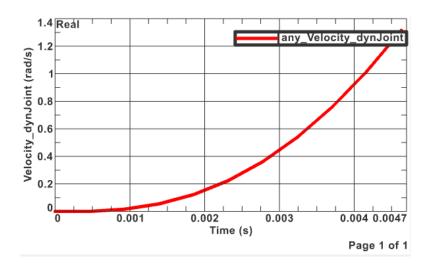
7. Create two more currents 'B' and 'C'. Select the same settings and add a phase shift of 120° for 'B' and 240° for 'C'.

Current 2D	ა? X	Ourrent 2D	ა? X
▶ Type		▶ Type	
Name		Name	
Destination Folder		Destination Folder	
✓ Primary Direction +Z		 Primary Direction +Z 	
Туре	PlanePhysical 👻	Туре	PlanePhysical 🔻
 Select Physical 	B_Pos::Team30_fi ▼ S ▼	 Select Physical 	🕞 C_Pos::Team30_fi ▾ 😻 ▾ ▼
▼ Back Direction -Z (opt	ional)	▼ Back Direction -Z (op	tional)
Туре	PlanePhysical 👻	Туре	PlanePhysical 🔹
 Select Physical 	B_Neg::Team30_f ★	 Select Physical 	🕞 C_Neg::Team30_f ▾ 😻 ▾ ▼
✓ Magnitude		✓ Magnitude	
Method	Harmonic (cos) 🔹	Method	Harmonic (cos) 🔹
Electric Current Amplitude	2892.3142 A 🔹 🗸	Electric Current Amplitude	2892.3142 A 🔹 🗸
Frequency	60 Hz 🔹 🗸	Frequency	60 Hz • •
Phase Shift	120 ° 🗸 🗸	Phase Shift	240 ° • •
Card Name CurrentOnPhysi	cal2D	Card Name CurrentOnPhysi	ical2D
	OK Cancel		OK Cancel

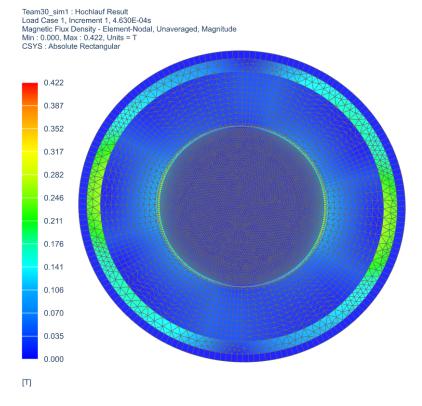
8. Solve the solution.

7.1.2 Post Processing

1. Display the graph results for the velocity. This should look like in the following picture:



2. This picture shows the 'Magnetic Flux Density' of the motor:



7.1.3 Solution with Conductor Motion

- 1. Clone the solution
- 2. Instead of using the Moving band, we are now going to use the type 'Conductor Motion'.
- 3. Delete the two existing simulation objects. Create then a new one of type 'Dynamic Motion 2D' with the following settings:

Dynamic Motion 2D		ა ?	' X				
▼ Туре							
Conductor Motion, const Sha	pe						
Name							
 Destination Folder 							
Simulation Object Container	Root	• E	ŝ				
 Motion Definition 				Physical Property Ta	ble Mar	ager	?
Moving Parts (2)		E	=	Create			
	Rotation		-	Filter			
Type Rotation Axis	Z		-	 Selection 			
Airgap for Torque Output				Name	Label	Туре	
				🔒 Stator::Team30_fem	3	PlanePhysical	^
Periodicity				Rotor_Steel::Team30	4	PlanePhysical	
Initial Conditions				Rotor_Alu::Team30_f	5	PlanePhysical	
			-	A A PostTeam30 fem	6	PlanePhysical	Ý
Spring, Damper			_	6		# =# × O	Ľ,
Limits				▼ List			
More				Rotor_Steel::Team30_fem1	::[4]		Ð
Card Name CMDyn				Rotor_Alu::Team30_fem1:			\mathbb{R}
	ОК	Cance	el				Close

4. Solve the solution.

7.1.4 Post Processing

The result graphs should match with the graphs of the 'Hochlauf' solution.

The tutorial is completed.

7.2 Motor (Team30_3D) with Conductor Motion

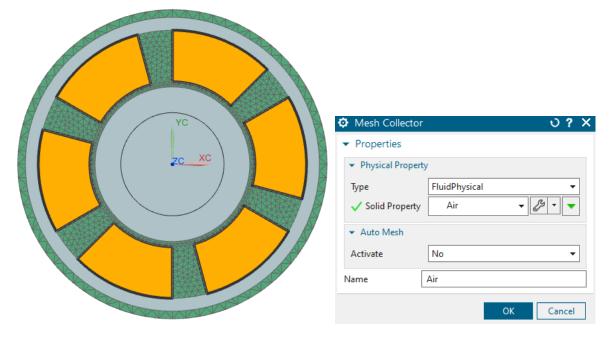
In this simulation, we are going to look at the same motor but in a 3 dimensional space.

- 1. Open the file Team30_3D.prt.
- 2. Create a new Sim and Fem file(Non-Manifold) from 'Magnetics' Toolbar.
- 3. Use solver MAGNETICS, Analysis type '3D Electromagnetics' and Solution Type 'Magnetodynamic Transient'.
- 4. Name the solution 'Hochlauf' (e.g. start-up).
- 5. Activate the desired data in Output Requests (here we activate 'Rotorband Torque')
- 6. In time steps, set the 'Time Increment' to 1/2160 and the 'Number of Time Steps' to 10.

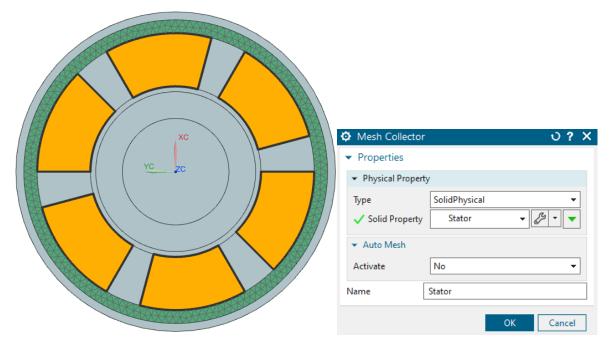
7. Set the 'Electromagnetic fields' as shown in the 'Initial Conditions' options.

Output Requests	Electromagnetic fields	Zero Vector	Potential	•
Time Steps	Initial Time	0	< • •	Ţ
Initial Conditions	Solve Electromagnetic Solution		-	

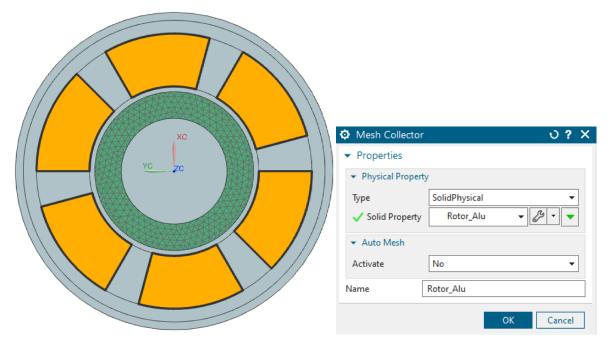
- 8. Click OK.
- 9. Switch to the FEM file.
- 10. Mesh the 'Air' using tets (3.2/2 mm)



- 11. Mesh the 'Stator' with tets and an element size of 5.25/2 mm.
- 12. Add the same material to the 'Stator', which was used in the 2D simulation.



- 13. Mesh the 'Rotor_Alu' with tets and use the element size of 5.1/2mm.
- 14. Add the same material to the 'Rotor_Alu', which was used in the 2D simulation.



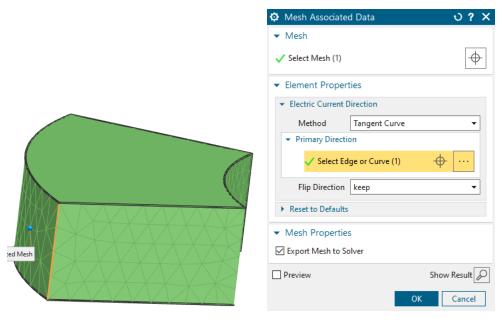
15. Mesh the 'Rotor_Steel' with tets and use the element size of 7.67/2 mm and use the same material, which was used in the 2D simulation.

Image: Operation of the second sec
▼ Physical Property Type SolidPhysical ✓ Solid Property Rotor_Steel ✓
Auto Mesh Activate No
Name Rotor_Steel OK Cancel

16. Mesh every coil using tets and an element size of 6.04/2 mm. Set the physicals for each coil as shown in the picture below.

	SolidPhysical		ა? X
	Physical Property Table		^
	 Properties 		
VC VC	Material	Copper	- <
	Material CSYS	Absolute	
	▼ Electromagnetic Solutions		
	Active		
	Conductor Model	Stranded	•
	Winding Vectors	to be defined in Mesh As	SO(🔻
	Coil Section Area	659.596557617 mm	² • •
	Number of Turns	1	-
	Fill Factor	1	-
	Skin/Proximity-Effect in Wire	e	
	Shape Functions	First Order	-
		ОК	Cancel

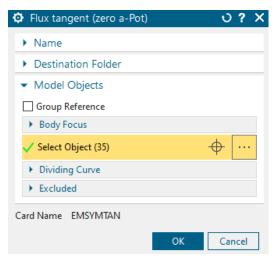
17. Edit the 'Mesh associated data' and select one of the vertical, outer lines (highlighted edge in the picture below).



- 18. We are done with the FEM file.
- 19. Change the window to the Sim file.
- 20. Create a simulation object 'Dynamic Motion' of type 'Conductor Motion'.Select the moving parts as seen in the picture below.

Dynamic Motion		ა ?	×				
💌 Туре			^				
Conductor Motion, const Sha	ipe			Physical Property Ta	ble Manager	1	' ×
▼ Name				Create			^
Dynamic Joint CM				▶ Filter			
Description				▼ Selection			
 Destination Folder 				Name	Label	Туре	
				☐ C_Pos::Team30_3D_f	6	SolidPhysi 🔺	
Simulation Object Container	Root	• E		🔒 C_Neg::Team30_3D	7	SolidPhysi	
✓ Motion Definition				Rotor Alu::Team30	8	SolidPhysi ¥	
Moving Parts (2)				Î	- [] == =	⊧×0 🖻	
Туре	Rotation	•		✓ List			
Rotation Axis	Z	-		Rotor_Alu::Team30_3D_fe	1[0]		
Airgap for Torque Output			~	Rotor_Steel::Team30_3D_f		\oplus	~
	ОК	Cancel				Close	2

21. The only constraint we use here is a 'Flux Tangent'. Select all outside faces (35 in total).



- 22. Lastly, we create the loads. Every pole has a positive and a negative current, so we have to add a load of 2892.3142 amps to the positive and a load of 2892.3142 amps to the negative coil.
- 23. A: Phase shift of 0°

Current	ა? X	Ourrent	ა? X
▶ Type		 Type 	
Name		Name	
Destination Folder		Destination Folder	
 Stranded Coil 		 Stranded Coil 	
Туре	SolidPhysical 👻	Туре	SolidPhysical 👻
 Select Physical 		 Select Physical 	A_Neg::Team30_: V
✓ Magnitude		✓ Magnitude	
Method	Harmonic (cos) 🔹	Method	Harmonic (cos) 🔹
Electric Current Amplitude	2892.3142 A - =	Electric Current Amplitude	-2892.3142 A -=
Frequency	60 Hz • •	Frequency	60 Hz
Phase Shift	0 ° • •	Phase Shift	0 ° • •
Card Name CurrentOnStran	ded3D	Card Name CurrentOnStran	ded3D
	OK Cancel		OK Cancel

24. B: phase shift of 120°

Ourrent	ა? ×	¢	Current		ა	? ×
▶ Type			Туре			
Name			Name			
Destination Folder			• Destination Folder			
 Stranded Coil 			 Stranded Coil 			
Туре	SolidPhysical 👻	1	Гуре	SolidPhysical		-
 Select Physical 	B_Pos::Team30_3 ▼	•	 Select Physical 	B_Neg::Team30_3 ▼	₽.	· 🔻
✓ Magnitude			 Magnitude 			
Method	Harmonic (cos) 🔹	1	Method	Harmonic (cos)		•
Electric Current Amplitude	2892.3142 A - =	E	Electric Current Amplitude	-2892.3142	Α	- =
Frequency	60 Hz 🔹 🕶	F	Frequency	60	Hz	• •
Phase Shift	120 ° 🗸 🗸	F	Phase Shift	120	۰	• •
Card Name CurrentOnStran	ded3D	Ca	ard Name CurrentOnStran	ded3D		
	OK Cancel			ОК	Ca	ncel

25. C: phase shift of 240°

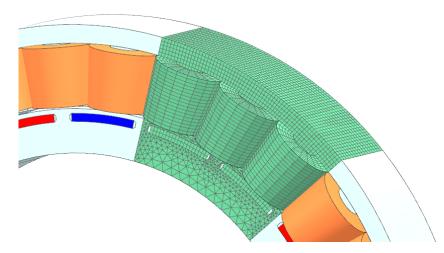
Current	ა? ×	🗘 Current	ა? X
▶ Type		▶ Type	
Name		Name	
Destination Folder		Destination Folder	
 Stranded Coil 		 Stranded Coil 	
Type ✓ Select Physical	SolidPhysical ⊕ C_Pos::Team30_3 ▼ ⊕ ▼ ♥ ♥ ♥ ♥ ♥	Type ✓ Select Physical	SolidPhysical ⊕ C_Neg::Team30_: ▼ ⊕ ▼ ♥ ▼ ♥ ▼ ●
✓ Magnitude		✓ Magnitude	
Method	Harmonic (cos) 🔹	Method	Harmonic (cos) 👻
Electric Current Amplitude	2892.3142 A - =	Electric Current Amplitude	-2892.3142 A •=
Frequency	60 Hz • •	Frequency	60 Hz 👻
Phase Shift	240 ° 🗸 🗸	Phase Shift	240 ° 🗸 🗸
Card Name CurrentOnStran	ded3D	Card Name CurrentOnStran	ded3D
	OK Cancel		OK Cancel

- 26. Solve the solution.
- 27. The results should be similar to the ones from the 2D simulation.

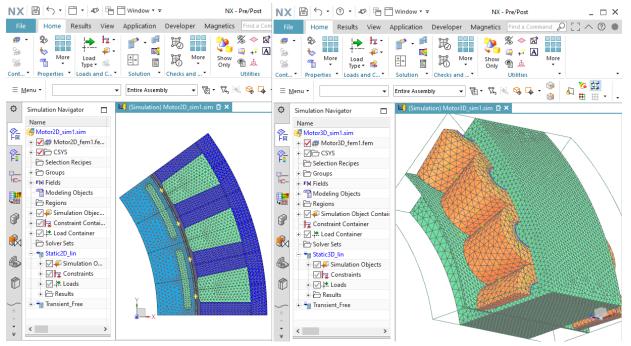
8 Tutorial 7

8.1 Motor Analysis in 3D by Sliding Motion

In this tutorial we analyze a permanent magnet electric motor in 3D. For motion we use the Sliding Motion technique. A Fem and Sim file are already created and in this exercise we walk through the existing model to check and explain the used features.



In this tutorial the motor is set up in 3D but also in 2D. Both models base on a 2D skeleton geometry. The skeleton contains only basic dimensions and allows geometry updates into both simulation models. For working on such motors in practice we recommend the process shown here: Having both 2D and 3D simulation models in parallel. Because in 2D solving runs much faster, it is a good idea to first test many things using the 2D model and when this runs fine, set up the 3D model in the same way and investigate the 3D behaviour.



Previous tutorials already have shown features for 2D models, so we now will concentrate on the 3D model. The following files contain all features for the set up of the model. There is also a youtube video (click Link) available for this example.

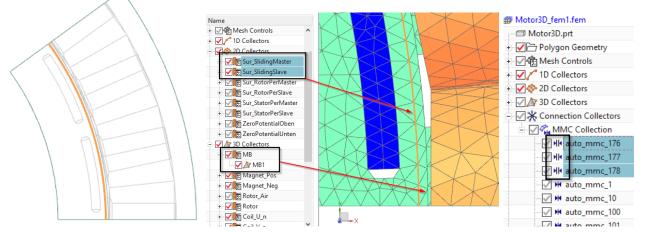
- Download the model files for this tutorial from the following link: https://www.magnetics.de/downloads/Tutorials/6.CouplMotion/6.7Motor3D.zip
- Open the file 'Motor3D_sim1.sim'.
- Change the displayed part to the Fem file to check the following steps.

8.1.1 Properties for Sliding Surface

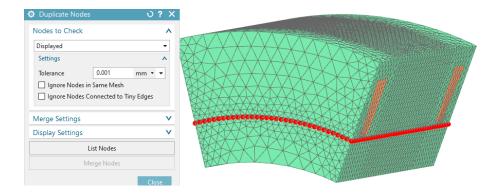
In models that use the Sliding Surface technique, there must be such two contacting surfaces that later will slide. Normally this sliding is designed into the air gap of the motor. If a motor has two such air gaps it is also possible to define two such sliding surface pairs. It is important that the sliding surfaces have the following properties:

- One surface must be connected to the rotor and the other surface must be connected to the stator. At the contact, the two surfaces do NOT share the same nodes. Mesh Mating Conditions using the option 'Free Coincident' allow for such a condition. Following we call such two surfaces 'Sur_SlidingMaster' (the one at the stator) and 'Sur_SlidingSlave' (the one at the rotor). The software will create link constraints between the edges of these two.
- The surface must be meshed in structured way. So, when rotation appears, the link condition can snap from one element edge to the following. The optimal step size is exactly the element size, but rotor positions in between such snap points are also possible, because the neighbouring 3D elements are allowed to be slightly deformed to meet such positions.

The following picture shows the position of the sliding surfaces in the tutorial motor. Also there is shown the neighbouring 3D element layer, called 'MB'. In this layer we allow the deformation of elements. Also this layer is used to compute the torque results using the maxwell stress tensor technique. The right picture below shows the special Mesh Mating Conditions on those faces that belong to the sliding surface. The symbols show their type 'Free Coincident'.



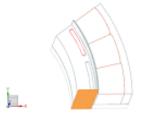
Also, for checking the sliding surface conditions for correctness, it is a good idea to ask the model for duplicate nodes. This check must show all nodes at the sliding surfaces highlighting. See picture below.



8.1.2 Properties for Periodicity Segments

Of course, to keep simulation time as short as possible, one should exploit the periodicity that most motors have. Therefore, we will model a link constraint between the right and left section faces of the motor segment. The following recommendations are given:

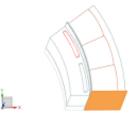
- These section faces are meshed at the very beginning of the meshing process using the '2D Dependent' meshing feature.
- The resulting 2D meshes at these periodicity faces will later be referenced in a link condition. Therefore we will call such 2D meshes as follows:
- 'Sur_RotorPerMaster': The faces on the right (master) side, belonging to the rotor.



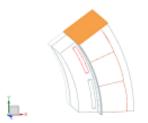
• 'Sur_RotorPerSlave': The faces on the left (slave) side, belonging to the rotor.



• 'Sur_StatorPerMaster': The faces on the right side, belonging to the stator.



• 'Sur_StatorPerSlave': The faces on the left side, belonging to the stator.



8.1.3 The Rotor Joint Definition

Most specific definitions for the sliding surface technique are in a simulation object. So we take a closer look on that side.

- 1. Change the displayed part to the Sim file for the following steps.
- 2. Edit and check the simulation object 'Enforced Joint(1)'. Notice its Card Name, it is a 'SMEnfRevolute', meaning an 'Enforced Revolute by Sliding Motion'.
- 3. Only the 'Driver' and the step size (of Velocity) are defined in this dialogue. All remaining definitions correspond to the geometrical description of the rotor and are stored in a modeling object 'Rotor Geometry Definition1'.
- 4. Edit that modeling object to see its properties. There are several boxes to define the geometry. We walk through them following.

ndeling Objects	Enforced Motion		0? X	Rotor Geometry D	efinition	ა? X
🗁 Regions	Turno		v	Modeling Object		×
🗹 🐗 Simulation Object Conta	Туре		· · ·	Wodeling Object		•
🔁 Constraint Container	Name		V	Properties		^
🖉 🖈 Load Container	Destination Folder		v	Description		L\$
🗁 Solver Sets	B. 6. 34					
📲 Static3D_lin	Definition		^	Definition		*
🖃 🗹 🐗 Simulation Objects	Driver	Angular Step	-	Periodicity		~
Enforced Joint(1)	Enforced Angular Stepsize	0.5000 °	• =	Rotor		v
Constraints		Rotor Geometry Definition1 👻 🖉		Airgap		~
🖃 🗹 🗈 Loads	Rotor Geometry Definition	Kotor Geometry Definition 1 👻				v
				Link Periodic Faces		v
Phase_U	Card Name SMEnfRevolute			Link Airgap Sliding Fac	es	~
Phase_W		ОК	Cancel	Tree Lines		v
+ 🗁 Results						

5. The 'Definition' box gives some information about the orientation of the model. A button 'Show Orientation Image' can be set. The model must be oriented in this way: Rotation is about global X and the start position must be aligned with X.

Definition		^		
Model Orientation	Rotate about Z, align with X			
Show Orientation Image				
ř.			Definition Periodicity	v
* × ×			Periodicity	Periodic (Even) 👻
Tolerance Factor for Links	0.0001 -	•	Number of Element Divisions	33
Periodicity		×	Model Segment Angle	36 • •

• The 'Tolerance Factor for Links' must sometimes be modified.

- It must be increased (use factor 10) if a message like the following appears in the solution monitor:
 Warning : Could not find edge corresponding to reference edge ...
 Error : Constraint Link: bad correspondance of number of edges ...
- It should be decreased, if this message appears: Warning : Edge ... (...) already exists with tolerance ...
- 6. In box 'Periodicity' there must be chosen between periodic (even) and anti periodic (odd) conditions. The 'Number of Element Divisions' on the sliding surface as well as the 'Model Segment Angle' are defined here.
- 7. box 'Rotor' must be used to define the 'Moving Parts and Moving Air'.

		Rotor		v
		Airgap		^
		Number of Air Gaps	1	-
		Туре	FluidPhysical	-
Periodicity	Y	🗸 Airgap 1 Ring	⊖ MB	- & 😻 -
Rotor	^	Airgap 1 outer Radius	120	mm 👻 💌
Moving Parts and Moving Air (4)		Airgap 1 inner Radius	119	mm 🔹 💌

- 8. In box 'Airgap' the 'Number of Air Gaps' is defined as well as the Physicals of the 'Airgap 1 Ring'. Also the two radii are there defined.
- 9. Box 'Link Periodic Faces' must be used to define the four previously already meshed 2D dependent meshes on the rotor and the stator, master and slave sides. The naming here corresponds to the names we used in the previous text. Such 2D meshes must be assigned a physical of type 'BoundaryConditionSet'.

Airgap	×		
Link Periodic Faces	^		
Show Images		Туре	BoundaryConditionSet -
Туре	BoundaryConditionSet 👻	✓ Surface for Stator per Master	🔒 Sur_StatorPerMaster 🔻 🔗 😻 🔻
✓ Surface for Rotor per Master	🕞 Sur_RotorPerMaster 👻 🌽 🔻		
	A	j.,	
Ľ.		Туре	BoundaryConditionSet -
Туре	BoundaryConditionSet 👻	✓ Surface for Stator per Slave	🕞 Sur_StatorPerSlave 🔻 🔗 😻 🔻
✓ Surface for Rotor per Slave	🔒 Sur_RotorPerSlave 🔻 🖉 😻 🔻		
		Ŀ.	
<u> </u>		Link Airgap Sliding Faces	×

10. Box 'Link Airgap Sliding Faces' defines the two previously meshed sliding surfaces on the rotor and the stator. Also, the two lines 'Lin_SlidingSubmaster' and 'Lin_SlidingSubslave' are defined here.

Link Periodic Faces	×		
Link Airgap Sliding Faces	^		
Show Images			
Туре	BoundaryConditionSet -	Туре	BoundaryConditionSet 👻
✓ Surface for Sliding Master	🔒 Sur_SlidingMaster 👻 🤣 💌	\checkmark Line for Sliding Submaster	🔒 Lin_SlidingSubmaster 👻 🎸 🔻
	in.		b.
Туре	BoundaryConditionSet 👻	Туре	BoundaryConditionSet 👻
Surface for Sliding Slave	🔒 Sur_SlidingSlave 👻 🌽 😻 🔻	Line for Sliding Subslave	🔒 Lin_SlidingSubslave 👻 🌽 🐳 💌
			L.

11. Finally, the box 'Tree Lines' can be used to define the previously meshed main edges of the rotor and the stator. This selection is only necessary if the gauge type 'Tree, Cotree' is used (in case of sliding surface motion and linear material properties).

Tree Lines	٨
Show Images	
Туре	BoundaryConditionSet 👻
Tree Lines for Gauge	🔒 Lin_TreeLines 🛛 🗸 😻 🔻
Ŀ	

8.1.4 Recommendations for Meshing

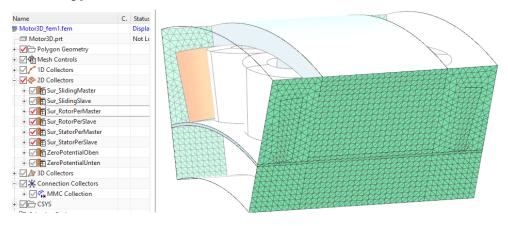
We recommend the following process for the meshing of 3D motors with the sliding surface technique.

- 1. Create the CAD model in the correct orientation.
- 2. Include the air gap in the CAD model. For better accuracy of torque results, the air gap should be divided into three layers.



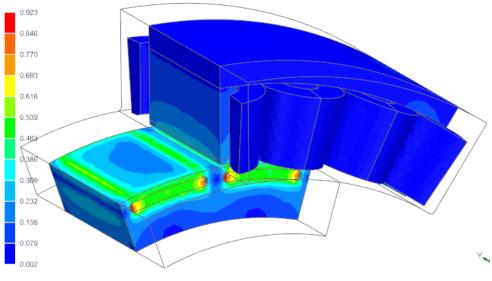
3. In 'Edit FEM', set the 'Default Cyclic Symmetry Cylindrical CSYS' to absolute (see picture above, right). This allows the 2D dependent meshes to become more robust.

- 4. Start with the automatic creation of the Mesh Mating Conditions. First create all with option 'Glue Coincident'. Then locate those conditions at the sliding surface and set these to option 'Free Coincident'.
- 5. Then create the 2D dependent meshes at the two segment sections. Try as much as possible using the structured mesh option there. This is a bit sensitive, because the link constraints will not work, if these 2D dependent meshes are not accurately positioned. Put the meshes into mesh collectors. Assign names, as they are later referenced (Sur_RotorPerMaster, ...).
- 6. Then create the sliding surface mesh at the stator side (Sur_SlidingMaster). Use the '2D Mapped' mesh command for that. When defining the number of elements along the circular edge, keep this number for later use in the modeling object for the rotor definition.
- 7. Now create the 3D meshes at the air gap.
- 8. Next create all other 3D meshes. Easiest way is to use only tetrahedral elements, but also hex and pyramid transitions can be used.



8.1.5 Solving and Post Processing

The remaining features in this model are not very special: Meshes, mesh mating conditions, materials, electric current loads, solution properties. Of course, the solve time for this 3D model is much higher than for the 2D case. Particular if there is nonlinear material included this is the case. We want to mention that also dynamic solutions are possible with this Sliding Motion feature. Results show either tabular graphs with torque, losses and others or plot results showing field results with time steps and motion. Following we present some result illustrations.

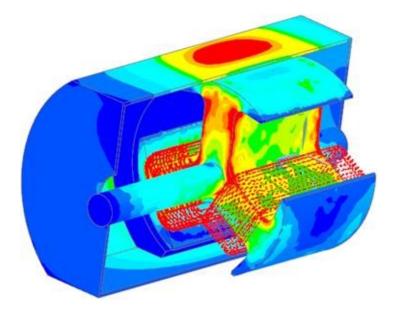




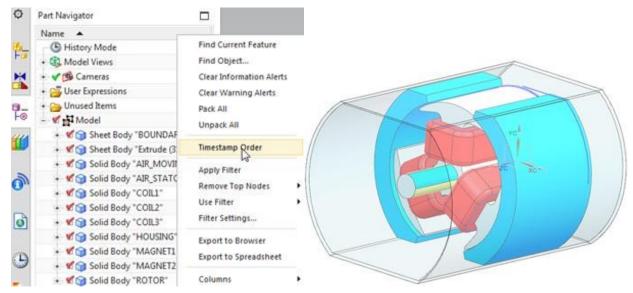
9 Tutorial 8

9.1 Motor Analysis in 3D by General Motion

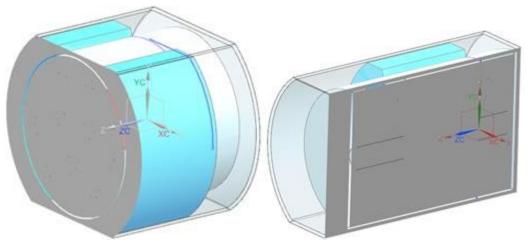
In this tutorial we analyse a permanent magnet electric motor in 3D. For motion we use the General Motion technique. A Fem and Sim file are already created and in this exercise we walk through the existing model to check and explain the used features.



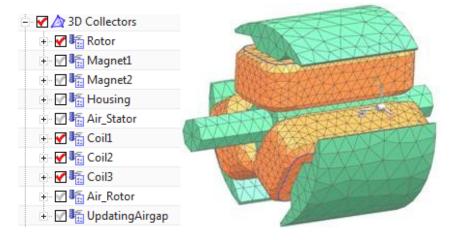
- Download the model files for this tutorial from the following link: https://www.magnetics.de/downloads/Tutorials/6.CouplMotion/6.9Motor3DGenMot. zip
- 2. Open in Simcenter the file 'Motor.prt'.
- 3. For checking this CAD file, use the modeling application. In the Part-Navigator deactivate the 'Time Stamp'. This setting allows to see the volumes and sheet bodies regardless of their creation history.



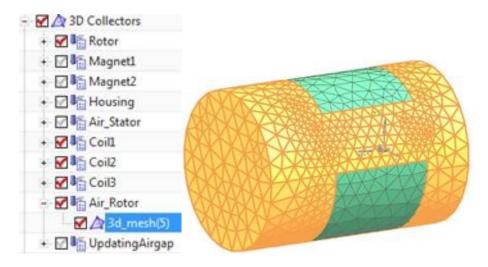
4. For the General Motion feature there must be a separation between moving and stationary parts. All nodes that belong to the moving parts will be rotated during the analysis process. The area between moving and stationary parts will be meshed newly at every time step. To keep this process efficient one should try to only have a small area for that mesh update. In many cases – and also in this example – a small air gap around the moving parts can be used for the mesh update. When sectioning the view the air gap that surrounds the rotor can be seen.



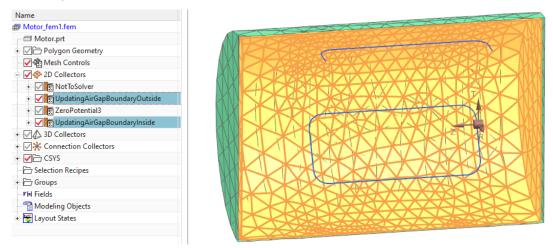
- 5. Open the Fem file Motor_fem1.fem. All meshes are very coarse. Let's check the meshes that reside in the file.
- 6. First check the Rotor mesh and the three coil meshes. There is nothing very special here.



7. Next check the 'Air_Rotor' mesh. This mesh represents the rotating part of air. All rotating meshes are referenced in the Sim file by the Simulation Object 'Enforced Revolute by General Motion'.



8. Around these rotating parts there is the updating gap (picture below). That gap is bounded by 2D meshes. When solving, the system will automatically create and update that gap mesh using a 'Solid from Shell Mesh' type mesh. This mesh will update with each time step. The boundary meshes reside in two mesh collectors: 'UpdatingAirGapBoundaryOutside' and 'UpdatingAirGapBoundaryInside'. The number and names of these boundary meshes can vary as needed.



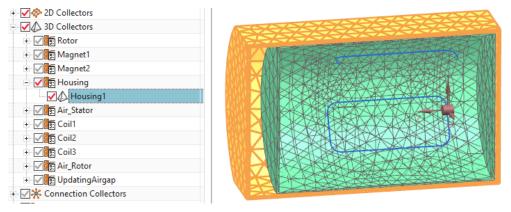
9. Notice the 3D mesh collector 'UpdatingAirgap'. Edit and open the box 'Auto Mesh' and see that this collector has activated 'Auto Mesh at Solve'. In 'Shell Boundary for 3D Mesh' there are the two Physicals of the boundary meshes selected. That way, the air gap mesh will be recreated at each solve step.

 Auto Mesh Magnet1 Magnet2 Magnet2	🗹 🛱 Mesh Controls	Mesh Collector	ა?	x			
 Notiosolver Uses Notiosolver Uses Notiosolver Uses Notiosolver Uses Notiosolver Uses Notiosolver Uses Notiosolver Solid Property UpdatingAirgap UpdatingAirgap Create Selection Selec	🖃 🗹 🗇 2D Collectors						
Image: Second Secon	🕂 🔽 📴 NotToSolver	 Properties 					
Image: Spectral system Image: Spectral system <td>🕀 🔽 🌆 UpdatingAirGapBoundaryOutside</td> <td> Physical Property </td> <td></td> <td></td> <td></td> <td></td> <td></td>	🕀 🔽 🌆 UpdatingAirGapBoundaryOutside	 Physical Property 					
• ✓ 🛱 UpdatingAirGapBoundaryInside ✓ Solid Property UpdatingAirgap ✓ Physical Property Table Manager ? × ✓ 🛱 Rotor ✓ Magnet1 ✓ Auto Mesh ✓ Create · Create · Create · Filter · Shell Boundary for 3D Mesh (2) Element Type Element Type List UpdatingAirGapBoundaryInside ✓ ⑦ Coil2 Name UpdatingAirgap ✓ Create List UpdatingAirGapBoundaryInside OK Cancel OK Cancel UpdatingAirGapBoundaryInside ✓	🛨 🔽 📷 ZeroPotential3	Ture)hurical -				
Auto Mesh Auto Mesh at Solve Auto Mesh at Solve Auto Mesh at Solve Shell Boundary for 3D Mesh (2) Bernent Type Tetra4 Ver Coil2 Name UpdatingAirgap OK Cancel OK	🗄 🔽 🌆 UpdatingAirGapBoundaryInside		· · · · · · · · · · · · · · · · · · ·				
 Auto Mesh Magnet1 Magnet2 Magnet2	🗄 🗹 🥼 3D Collectors	V Solid Property U	pdatingAirgap 🔻 🌽 🐔 🔻				
 Magnet1 Magnet2 Activate Auto Mesh at Solve Shell Boundary for 3D Mesh (2) Name UpdatingAirgapBoundaryOutside UpdatingAirGapBoundaryInside OK 	E- Rotor	A + 14 1		-	Physical Property Table Manager	?	×
Image: Shell Boundary for 3D Mesh (2) Image: Shell Boundary for 3D Mesh (2) <td>🕀 🔽 🛐 Magnet1</td> <td> Auto iviesn </td> <td></td> <td>_</td> <td>Create</td> <td></td> <td>^</td>	🕀 🔽 🛐 Magnet1	 Auto iviesn 		_	Create		^
Image: Stator Ima	🕀 🔽 🛐 Magnet2	Activate Auto	Mesh at Solve 🔹		Filter		
• View Air_Stator • View Coil1 • View Coil2 • View Coil3 • View Coil3 • View Coil3 • View Coil3 • View Coil3 • View Coil3 • View Coil3 • View Coil3 • View Coil3 • View Coil3 • View Coil3 • View Coila • View Coila • View Coila <td< td=""><td>🕀 🔽 🚋 Housing</td><td>Shell Boundary for 3D Mes</td><td>h (2)</td><td></td><td>Colortion</td><td></td><td></td></td<>	🕀 🔽 🚋 Housing	Shell Boundary for 3D Mes	h (2)		Colortion		
	🕀 🔽 📷 Air_Stator	Flowerst Trees Total					
Vince OpdatingAirgap	🕀 🔽 📷 Coil1	Element lype	• •		▼ List		
Concel	🕀 🔽 📑 Coil2	Name Updati	ngAirgap			\oplus	
OK Cancel	🕀 🔽 📑 Coil3				UpdatingAirGapBoundaryInside	\times	
UpdatingAirgap Close	+		OK Cancel			E	۷
	👘 UpdatingAirgap		Cancel			Close	

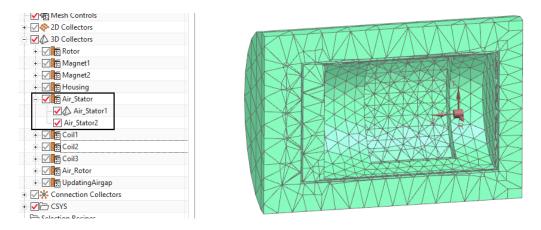
10. Next look at the magnets. They use cylindrical coordinate systems and point in opposing directions.

SolidPhysical	৩ ? ×	
 Physical Property Table 		SolidPhysical
Name	Magnet1	Physical Property Table
Label	2	Name Magnet2
Properties		Label 3
Aaterial	N30EH at 100C 👻 🕼	Properties
Naterial CSYS	Cylindrical 👻	Material N30EH a
Material Orientation		Material CSYS Cylindrical
		Material Orientation
 Electromagnetic Solution 	S	Electromagnetic Solutions
Active		✓ Active
Conductor Model	Massive 🔻	Conductor Model Massive
 Thermal Solutions 		Thermal Solutions
 Elasticity Solutions 		Elasticity Solutions
 Motion Solutions 		Motion Solutions
Post Processing		Post Processing

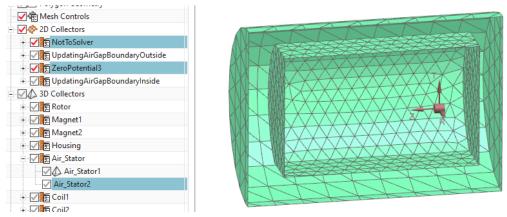
11. And check the housing, a thin walled metal geometry.



12. The Air_Stator mesh collector represents those air parts that do not move. This is made by two 3D-meshes.



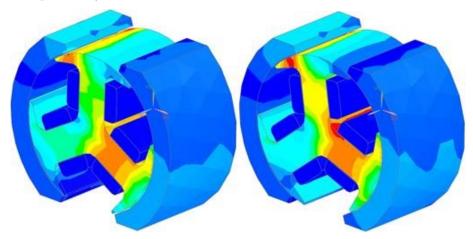
13. One of the above two Air_Stator meshes is created by the function '3D Mesh from Shells' and the used shells are shown here: One shell mesh uses a simple 'Not to Solver' Physical and the other a 'Zero Potential'.

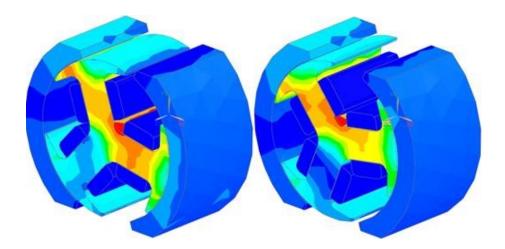


- 14. Change to the Sim file now.
- 15. We use a Magnetostatic solution type. Magnetodynamic transient is possible also.
- 16. See the Enforced Revolute joint feature: We rotate about Z and Angular Velocity set to 1 rev per min.

Enforced Motion	ა? X		
▶ Type			
Name			
Destination Folder		Ourrent	ა? X
✓ Rotor Geometry		 Туре 	
Moving Parts (5)		Name	
Airgap for Torque Out		Destination Folder	
suppress Remesh		✓ Stranded Coil	
Definition		Type ✓ Select Physical	SolidPhysical Coil1 Coil1 SolidPhysical SolidP
Axis Driver	Z 🗸 🗸	✓ Magnitude	
Angular Velocity	1.0000 rev/min - =	Method	Harmonic 🔹
1.11.1.0		Electric Current Amplitude	-5.0000 A -=
 Initial Conditions 		Frequency	0.0167 Hz =
Angle	0 ° • •	Phase Shift	0.0000 ° · =
Card Name GMEnfRevolu	te	Card Name CurrentOnStran	ded3D

- 17. All currents are set to harmonic as shown above for coil 1. You can change these conditions to your needs.
- 18. Solve the solution for as many steps as you desire.
- 19. Postprocess your results.





10 Tutorial 9

10.1 Electric Motor Analysis for NVH

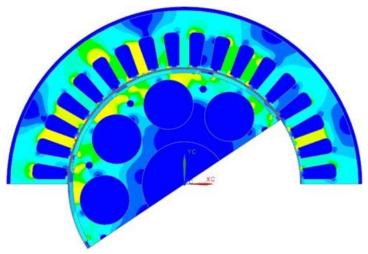
Vibration and noise from in runner electric motors comes mainly from the stator deformation, induced by the constantly varying magnetic field activating individual phases. The deformation on the outer surface of the motor causes the air around it to move, generating pressure differences, which are perceived as noise for the human ear. There are three disciplines to be considered in this analysis—magnetic, vibrational (structural), and acoustic [SantosAnthonis-NaclerioGyselinck].

In this tutorial we analyze a servo motor for the magnetic and vibrational (structural) disciplines using MAGNETICS for NX and NX NASTRAN. For the third part, acoustics, NX NASTRAN can also be used, but we will not show it here because we focus on the magnetic part.



Picture: Siemens 1FT6 Servomotor

The results base on magnetic forces on the teeth. For this we will first perform a transient analysis in 2D and compute forces on the teeth. We will use a postprocessing solver feature that converts the time dependent forces into frequency dependent ones using Fourier transformation. Then those frequency forces will be imported into NX10 using the feature load recipe that is new in NX10. Finally in NX a frequency response analysis in NX NASTRAN Solution 111 with those forces as input will be performed.



Picture: Complete Simulation Model of 1FT6 Motor. Fluxdensity distribution in 1FT6 Servomotor. Two out of four poles are modeled using periodicity conditions.

Starting point is an already complete simulation model of a servo motor. We first check it and then add the necessary steps for NVH export and solve in NX NASTRAN.

- 1. Download the model files for this tutorial from the following link: https://www.magnetics.de/downloads/Tutorials/6.CouplMotion/6.8MotorNVH.zip
- 2. Open the part Motor1FT6_sim1.sim.
- 3. Change to the FEM file and notice the following features that are specific for NVH export:
 - This motor has a total of four poles and is modeled using two poles. There are 18 teeth in the model on which we want to analyze for forces.
 - Notice there are faces under all teeth. The next picture shows the face under the second tooth. These faces are defined in CAD by subdivisions of the neighboring faces of the air gap.

Physical Prop	A CALLER OF LEVEL	> O X	11	h	~	11
Name	tooth2			I		11
Label	102			1 1112		
roperties		v		B		-+
	ОК	Cancel	1			
/	1	1				
	1					
	1					
×X			-			

- Each of these tooth faces has its own physical. The numbers of them start at 101 and then count up to 118.
- The thickness of these gaps is 0.43 mm as you see in the next picture.



- 4. Change to the SIM file
- 5. Notice there already exists a magnetostatic solution with all loads and BCs already given. If you want solve this solution for checking, but there is nothing very special to learn from. The torque should vary between about 18-20 Nm.
- 6. Create a new solution of type 'Magnetodynamic Transient'.
 - At Output Requests under 'NVH Coupling' activate the switch 'NVH Motor Export' (also activate Motion Data under Tables). Key in the first and last physical ID of your tooth faces at 'NVH Start PID' and 'NVH End PID'. Also key in the Gap Thickness to 0.43mm.

Solution		<u>ა</u>		
Name	MagDyn			
Solver	MAGNETICS			
Analysis Type	2D or axisym Electromagnetics			
Solution Type	Magnetodynamic Transient			
Magnetodyna	mic Transient		A	
- Output Req		^	Plot	
Time Steps			Magnetic Vectorpotential	
- Initial Cond			Magnetic Fluxdensity	
- Coupled Th	NVH Coupling	^	Magnetic Fieldstrength	
2D	NVH Motor Export		Current Density	
	First Tooth ID	101	Eddy Current Losses Density	
	Last Tooth ID	118	Nodal Force - virtual	
	Gap Thickness	0.43 mm • •	Nodal Moment - virtual	
	Force Scale Factor	1	Lorentz Force	
	Add to existing unv	·	Displacement	
			Table	
	4D Fields	^	🔽 Total Force - virtual	
	Force - virtual, NodelD Tabl		Total Moment - virtual	
	Forcedensity - virtual, XYZ		Total Lorentz Force	
	Lorentz Force, NodelD Tabl		RotorBand Torque - stresstensor	
	Eddy Current Losses, XYZ T		RotorBand Force - stresstensor	
	Time Delay	0 sec • •	Fluxlinkage - Vectorpotential on Conductors	
	Time Extension	0 sec • •	Electrode Voltage	
	More	^	Electrode Current	
	Torque X Position	0 mm • •	Electrode Power	
	Torque Y Position	0 mm • •	Circuit Voltage	
	Torque Z Position	0 mm	Circuit Current	
	No Force Physical IDs		Circuit Power	
	FFT Max Number Frequencies	89	Eddy Current Losses	
	FFT Start Time	0 sec • •	Steinmetz Hysteresis Loss	
	FFT End Time	180*5.55e-005 sec • •	Steinmetz Eddy Current Loss	
		Set 1 v	Steinmetz Excess Loss	
			Motion Data	

٨

Set all other settings as follows:

• Time Steps: 180 steps will result in one electric period.

Magnetodynamic Transient

Output Requests	Time Increment	5.55e-005	sec 🔻 🔻
Time Steps	Number of Time Steps	180	
Initial Conditions	Number of Time Steps	100	
Coupled Thermal			
- 2D			

• 2D:

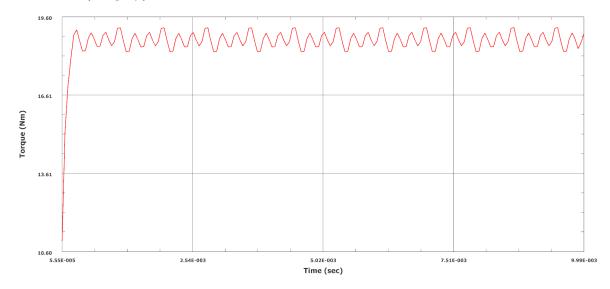
Plane	Absolute X,Y	
Axisym	metric	
THICKNESS	123	mm •
	·····	Axisymmetric

• In Solver Parameters under 'Numeric' set 'Epsilon' for better precision as shown:

Parameters

- General	Nonlinear Magnetic Material	
Numeric	Mathad	A damaged Coloma
Parameter Sweep	Method	Advanced Scheme
User Defined	Max. Number Iterations	30
	Absolute Tolerance	5e-005
	Relative Tolerance	0.0005
	Convergence Check	a-Field in whole Domain
	Error Norm	Mean L2 Norm
	BH-Curve Interpolation	Akima
	Allow BH-Curve in Freque	ncy Domain
	Relaxation Type	Adaptive 2 (min 0.1 max 1 steps 10, Keep b 🔻

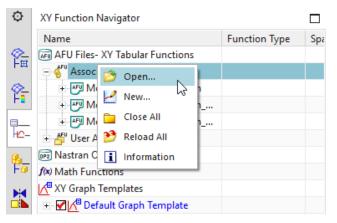
- 7. Put all existing loads, constraints and simulation objects into the new solution.
- 8. Solve the dynamic solution. This will take about 2-5 min.
- 9. Notice that after the solve process a couple of text files have been created.
 - As a first check, switch to the 'XY-Function Navigator' and plot the Torque results of the joint (in case that the .afu files are not already loaded, reload them); said results should lie (roughly) around 18 Nm.



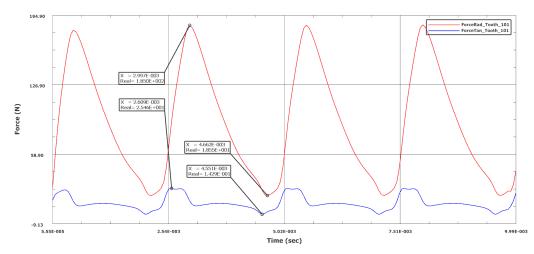
- In addition the files '..._NVH_ForceRad.txt' and '..._NVH_ForceTan.txt' where created; these files contain the tangential and normal forces on all 18 teeth for each time step (time domain).
- To create the associated .afu files, use the function 'Table-Result to AFU Graph' and select the above mentioned .txt files.

File Home Results View	Application M	lagnetics
? 🥻 🔓	≥ ∕	
	name Meshes Table-Re Collectors to AFU G	
Magnetics		N34
🕆 Menu 🕶 🔍 Wit	hin Work Part O 🔻	Table-Result to AFU Graph
Simulation Navigator		Convert MAGNETICS Tabular Results to AFU Graph Files

• In the XY-Function Navigator, press 'Open', to load the newly created .afu files.



• Then, plot the radial and tangential Forces of tooth 101



These forces are computed by the following equations based on the Maxwell Stress tensor that is evaluated on the previously defined teeth faces:

$$f_{t} = \nu (B_{n}B_{t})dl$$
$$f_{n} = \frac{1}{2}\nu (B_{n}^{2} - B_{t}^{2})dl$$

with: fn, ft: Radial and tangential force $v: 1/\mu_0.$ B_n, B_t : Radial and tangential magnetic fluxdensity

• '..._NVH.txt': This file is generated at the end of the run. It contains the Fourier transformed forces on the 18 faces. For each frequency (column 1) the amplitude (Column 2) and phase (column 3) are written.

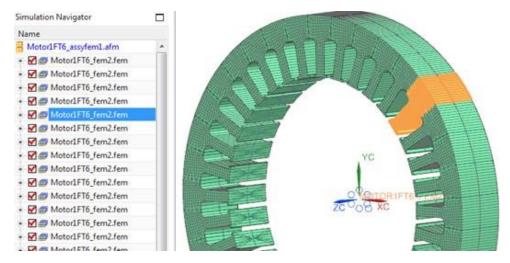
Some background on the method: You need to postprocess a time evolution with the help of a Fourier Transformation, in order to obtain the Frequency spectrum (that is e.g. required by NASTRAN 111 as input). Simplified (i.e. without oversampling) Nyquist's theorem states that If there are N time steps over this period (so N+1 points) then a maximum of N/2-1 frequencies will be obtained, where N is the Nyquist frequency. In our example, there are 180 time steps and we obtain 89 frequencies.

🔚 Motor1	IFT6_sim1-MagDyn_NVH.txt 🗵	
1	ForceTan on 101	
2	0 10.8575443990852	9 0
3	100.1001001001001	0.5109283383378073 0.04114257883124724
4	200.2002002002002	0.5046303137190914 0.1072966894853234
5	300.3003003003003	0.4919625875184704 0.1718146680351747
6	400.4004004004004	5.868126078333566 -1.11200430440846
7	500.5005005005005	0.4973035276757329 0.2211725640997294
8	600.6006006006005	0.4814235295587715 0.2714188739542077
9	700.7007007007007	0.4697348119215426 0.3102080459887557
10	800.8008008008007	6.260014500659615 -1.023974647448176
11	900.9009009009008	0.4413730559054543 0.3899204565605453
12	1001.001001001001	0.4306148613243748 0.417127511784962

• '..._NVH.unv': This file also contains the frequency domain forces, but it is formatted in unv format using dataset 58 what is capable for LMS Virtual Lab and NX10 Load Recipes. We are going to read it into NX to perform a NX NASTRAN dynamic response analysis.

🔚 Motor 1 F	T6_sim1-M	agDyn_NVH.un	v 🖂									
18		58										
19	101:	+Z										
20	Magn	etics :	for NX									
21	Empt	У										
22	Empt	У										
23	1											
24	1	2	1	1		0	ForceTan	101	2	NONE	0	0
25		6		90		1	0.00000e+0	0 1.00100e+0	002	0.00000e+00		
26		18	0	0	0			NONE		Hz		
27		13	0	0	0			NONE		N		
28		0	0	0	0			NONE		NONE		
29		19	0	0	0			NONE		RPM		
30	1.0	857544	39909e	+001	0.000	00000	0000e+000	5.1049597212	204e-	001 2.10149795	4887e-0	002
31	5.0	172830	11388e	-001	5.404	13301	1488e-002	4.8471899655	500e-	001 8.41111282	8555e-0	002
32	2.5	987898	38416e	+000+	-5.261	29214	6134e+000	4.8518965648	829e-	001 1.09095352	2418e-0	001
33	4.6	379932	09057e	-001	1.290	68992	1717e-001	4.4731440836	603e-	001 1.43389726	2857e-0	01
34	3.2	550510	22317e	+000+	-5.347	18845	6617e+000	4.0824328467	731e-	001 1.67772450	0586e-0	01
35	3.9	369241	82063e	-001	1.744	57555	4119e-001	3.7958825421	144e-	001 1.79326282	4941e-0	001
36	4.9	045255	36089e	-001	-3.185	96950	4131e+000	3.4563576769	949e-	001 1.87063222	2918e-0	001

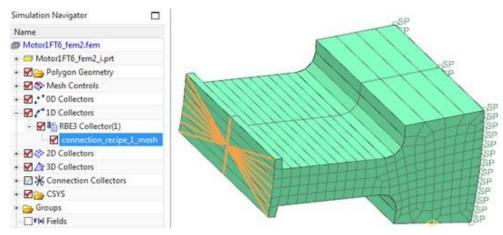
- Some information about the .unv file, see prior picture: The numbers 101 and 2 in line 24 define the source-function 101 and the direction 2. In line 25 you find 0.00000e+00 as the first frequency and 1.00100e+002 as the frequency step. Starting in line 30 there are couples of complex numbers, e.g. real part followed by their imaginary part.
- 10. Save your files and close them.
- 11. Open in NX10 or a later version the existing Sim file 'Motor1FT6_assyfem1_sim1.sim'. This model is already mostly build up for the NASTRAN NVH structural analysis.
- 12. Notice the following features in the model:
 - It is set up as an assembly fem. E.g. only one tooth is meshed in a FEM file called 'Motor1FT6_fem2.fem' and this mesh is placed in an assembly 36 times. The thickness (z-direction) is set to only 1/6 of the real 125 mm because we are not interested in z direction vibration effects.



• The nodes are merged in the assembly fem. Another possibility would be to use glue conditions in the SIM file.

Duplicate Nodes		υx					
Nodes to Check		~		0	-		
Displayed		•	-	0 0	2	2	
Settings		^	8		8	1	
Tolerance	0.001 m	nm • •	8	ğ ğ	ğ		
Ignore Nodes in		idges	8				2
lerge Settings		~ 2				J C	- 1000°
Merge Componer	nt FEM Nodes		The sum	2			
reference	None	•					
Display Settings		× 9	1				100
L	ist Nodes						11
Me	erge Nodes						12

• In the piece FEM file 'Motor1FT6_fem2.fem' there is a 1D Connection defined that connects the nodes that belong to the force face with a point in the middle. This node will be used in the SIM file to apply forces on.



• There is a Cartesian nodal coordinate system assigned to the force node. (This can be done with 'Edit, Node, Assign Nodal Coordinate System ...'). X points into the radial and y to the tangential direction. Later in the assembly mesh there will be such coordinate systems for each tooth. If we apply forces on the teeth they will use these coordinate systems.

Star Mash Controls	A stentily Nodal CIVS	οx
D Collectors D Collectors	Type	^
Bo 20 Collectors	Digliacement	
BA 10 Collectors B Connection Collectors	Object to Identify	~
- 83 (315	Select Object (3)	+
Groups	Coordinate System	
Modeling Objects (Filtered)	Displacement CS cays	
	Highlight Option	~
	Highlight Generality Orlighlight Name	
	Show All	

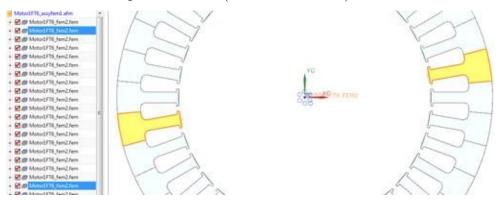
• There are bushing elements of grounded type (0-D Mesh: CBUSH Grounded) connected to the outside face nodes of the tooth. These bushings are quite weakly defined and serve as boundary conditions.

Name	Physical Property Tabl	0			A 🖯	-	
Motor1FT6_fem2_fem + CMotor1FT6_fem2_i.prt + CO Polygon Geometry	Name Label	PBUSHIL 3					1
Mesh Controls OD Collectors	Properties			3	^	77777	7/
- 🔛 🔤 Chush Collector (Gr	Nominal Values (PBUS)	0 Depend	dent Properties (PBUSH)	ŋ		11/1	- Ad
Od_mesh(1) ID Collectors	Stiffness			. ^	1	-Att	THE
■ 2D Collectors	X Translation	1	N/mm	• •		ATT	- The
• 🖬 🛕 3D Collectors	Y Translation	1	N/mm			FITH	and the state
Connection Collectors CSYS	Z Translation	1	N/mm	• •		41111	+++
Groups	X Rotation per Radian		N-rom				
TN Fields	V Rotation per Radian		N-mm				
Modeling Objects (Filtered)	Z Rotation per Radian		N-mm	tines			

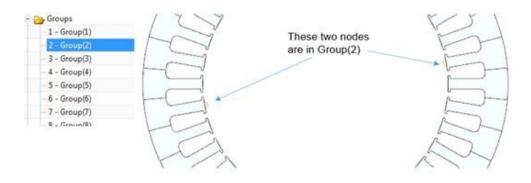
Correction: Set

the stiffness values to 10 N/mm.

• In the assembly fem file there are 18 groups defined on which we will later apply the forces. In the case of this motor we have a symmetry factor of 2 for the electromagnetic analysis, e.g. we had to analyze the half. So for the structural analysis all teeth forces must be applied to 2 teeth faces or their corresponding nodes. In case of other symmetry factors this must be considered similar. For easier selection those two nodes are put into a groups. So we have 18 groups each with 2 nodes. The next picture shows as an example two teeth (number 2 and 19) that must have the same forces:



• Consequently the two forces nodes of tooth 2 and 19 are put into one group. This can be seen in the next picture:



- 13. Change to the Sim file. Next you will create a load recipe that references the frequency dependent force information in the .unv file and that applies those forces respectively to the corresponding node groups.
 - Choose the function 'Load Recipes',

III More	Load Type •	Paging	t Type • n Object Type •	Solut		**
	tly Used	ary Materials				•
Prope	rties sical Prop	541994 - C	1000	lary Condition	n ID Manag	Jer
Mater	ials nage Mat	erials	📢 Manag	e Library Mat	terials	
	le Field ked Field		f(x) Formul			
		tion Tools equences	🔁 Load R	ecipes	6	•

• Set the 'Data Type' to 'Frequency Spectra' and click 'Create'.

Load Recipes	Manager	ა x
Create		^
Name	Load Recipe 1	
Description		
Data Type	Frequency Spectra	-
		Create

- In register 'Data Source' press 'Browse for a new data source' 11 and select the newly created unv file.
- Press 'Automatically populate the mapping table and ...,
- Set the register to 'Mapping'. You can see on the left that the system has found forces on nodes in the file.

Data Sources Load Conditions Mapping						
Data Mapping						^
Force - Node	Orientation	Nodal	1	•		
O Pressure - Node	Type	Target	DOF	1 DOF2	DOF3	DOI
O Enforced Displacements	Node	101	MEI	01 ME:101		^
 O Enforced Velocities 	Node	102	ME:1	02 ME:102		
O Enforced Accelerations	Node	103	ME1	03 ME:103		
	Node	104	ME:1	04 ME:104		
	Node	105	ME:1	05 ME:105		
	Node	105	MEIT	06 ME:106		
	Node	107	MEIT	07 ME:107		
	Node	108	ME:1	08 ME:108		
	Node	109	ME:1	09 ME:109		
	Node	110	ME:1	10 ME:110		*
	<					>
	* ×					
	Row Editin	1000				^
	Type No	de	+ 7	arget	*	
	DOF1 No	t Assigned				
	DOF2 No	t Assigned	-			
	DOFS No	t Assigned				
	DOF4 No	t Assigned				
	DOF5 No	t Assigned				
	DOF6 No	t Assigned				
🛃 🔄 🖻 🧇						
kutofill Options						
Assembly Component Motor1FT6_as	syfem1.afm					• •

• On the right side you see the mappings of the found forces to FE entities. First thing you should do is set the 'Orientation' option to 'Nodal'. This will result in the use of our nodal coordinate systems.

ta Mapping						
Force - Node	Orientation	Orientation Nodal				
O Pressure - Node	Type	Target	DOM	DOF2	DOF3	DO
 O Enforced Displacements 	Node	101	AF-101	ME-101		

- In column 'Target' you see the source-functions that are found in the .unv file. These source-functions correspond to the computed forces on the 18 tooth faces computed in MAGNETICS. You will now assign those source-functions to the prepared node groups.
- Click on the first line (Target 101). In the lower area of the dialogue set 'Type' to 'Group' and press to change the target for this source-function.

Туре	Group	-	Target		*
DOF1	Manual Entity	•	Entity	101	W
DOF2	Manual Entity	•	Entity	101	

ا

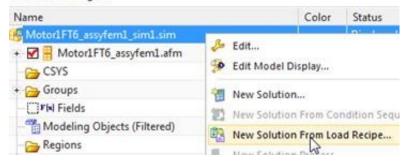
• In the next dialogue select the group Group(1) and Ok. The dialogue now shows that node group Group(1) will be applied by source-function 101 in the DOF1 (e.g. the radial) and DOF2 (the tangential direction).

Target Selection	on	υx						
Target Type		^						
Туре	Group	•	Туре	Target	DOF1	DOF2	D	D
Selection		^	Group	Group(1)::Motor1FT6_assyfem1	ME:101	ME:101		*
Group(1)::Mot	tor1FT6_assyfem1	• •	Node	102	ME:102	ME:102		=
· · ·				103	145 103	145 100		

• Repeat the last two steps for all 18 source-functions. At the end the dialogue should look like in the following picture. Maybe you want to run the validation check finally

									ں ا
Data Sources Load Conditions Mapping									
Data Mapping									
Force - Node	Orientation	Nodal 👻							
O Displacement Constraint	Type	Target	DOF1	DOF2	DOF3	DOF4	DOFS	DOF6	
 O Velocity Constraint 	Group	Group(1)::Motor1FT6_assyfem1		ME:101		1110-11			
 O Acceleration Constraint 	Group	Group(2)::Motor1FT6_assyfem1	ME:102	ME:102					
	Group	Group(3)::Motor1FT6_assyfem1	ME:103	ME:103					
	Group	Group(4)::Motor1FT6_assyfem1	ME:104	ME:104					
	Group	Group(5)::Motor1FT6_assyfem1	ME:105	ME:105					
	Group	Group(6)::Motor1FT6_assyfem1	ME:106	ME:106					
	Group	Group(7)::Motor1FT6_assyfem1	ME:107	ME:107					
	Group	Group(8)::Motor1FT6_assyfem1	ME:108	ME:108					
	Group	Group(9)::Motor1FT6_assyfem1	ME:109	ME:109					
	Group	Group(10)::Motor1FT6_assyfem1	ME:110	ME:110					
	Group	Group(11)::Motor1FT6_assyfem1	ME:111	ME:111					
	Group	Group(12)::Motor1FT6_assyfem1	ME:112	ME:112					
	Group	Group(13)::Motor1FT6_assyfem1	ME:113	ME:113					
	Group	Group(14)::Motor1FT6_assyfem1	ME:114	ME:114					
	Group	Group(15)::Motor1FT6_assyfem1	ME:115	ME:115					
	Group	Group(16)::Motor1FT6_assyfem1	ME:116	ME:116					
	Group	Group(17)::Motor1FT6_assyfem1	ME:117	ME:117					
	Group	Group(18)::Motor1FT6_assyfem1	ME:118	ME:118					

14. Now create a solution by RMB on the SIM file and 'New Solution From Load Recipe...'. Simulation Navigator



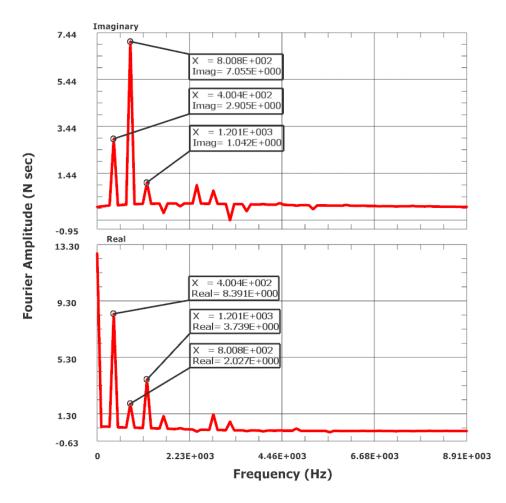
15. Choose the Solution 111. In this case we will accept all default settings, so press Ok and the solution is created.

Solution		
Solution		
Name	Solution 1 from Load Recipe 1	Solution 1 from Load Recipe 1
Solver	NX NASTRAN	- 🗹 🕂 Temperatures
Analysis Type	Structural	
2D Solid Option	None	+ P Data Source 1 - Empty
Solution Type	SOL 111 Modal Frequency Response	+ 🗁 Results

16. You can check every single force and the corresponding frequency domain data. In the subcase 'Data Source1' there are all forces shown. The next picture shows radial (DOF1) and tangential (DOF2) forces on node 10224. Use RMB and 'Edit' to find the corresponding table data for this force.

. Provide the second se			
🗹 🛹 Simulation Objects			
Constraints			
a ^p Data Source 1 - Empty	A Force -	Node on node 10224:DOF2(144)	
tt Loads	V Porce -	1400e 011100e 10224.00F2(144)	
- 🥁 Force - Node on node 10224:DOF2(144)	Name		Y
- 🙀 Force - Node on node 10224:DOF2(143)	Excitation		^
 Force - Node on node 10224:DOF2(72) 	Definition	Field	
- 😽 Force - Node on node 10224:DOF1(142)	V Data Se	ource1 - 118_+Z(72) 3 - 2	1 89
- 🙀 Force - Node on node 10224:DOF1(141)			
 Force - Node on node 10224:DOF1(71) 	Delay/Pha	ise-	V
+ 2 Force - Node on node 20448:DOF2(140)	Card Name	RLOAD1/RLOAD2	

17. Use to see the table, e.g. the amplitude and phase data as seen in next picture.



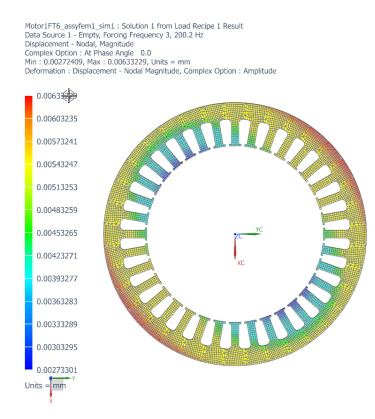
18. Activate the Source, by clicking RMB on 'Data Source..', 'Make Active'



- 19. We want to define the forcing frequencies, so choose RMB on 'Data Source' then 'Edit' and press 'Create Forcing Frequencies' . In the next window press 'Create'.
- 20. Choose the 'Frequency List Form' 'FREQ1' and key in as in the next picture. We use the same frequencies as they come out of the Fourier transformation of the electromagnetic tooth forces. But it is enough to check only for the lower frequencies, so we set the number to 15.

Forcing Free	0>			
Modeling Obj	ect		^	
Name	Forcing Frequen	Forcing Frequencies - Modal1		
Label	4	4		
Properties			~	
Description				
Frequency List			^	
Frequency List	Form	FREQ1	•	
First Frequency	y	0 н	z • •	
Frequency Increment		100.1 H	z • •	
inequency me				

- 21. Press Ok and in the next dialogue click 'Add' and then 'Close' and 'Ok'.
- 22. If you want check the information in 'Data Source': There are fields and loads on all 36 nodes in the two directions.
- 23. Solve the solution.
- 24. Postprocess the results.
- 25. Create animations: The computed results are in frequency domain. So all results are in real and imaginary part or in amplitude/phase. To see an animation of the shape how it would look like in the time domain we need to cycle about the phase. This can be done as follows:
 - Open the displacement result of one of the frequencies in the postprocessor.
 - Open the displacement result of frequency 2 which is at 200.2 Hz as we have applied it.
 - Use 'Set Result' to set the result. Set the 'Complex' option to 'At Phase Angle'. Ok.
 - Set the factor for 'Deformation' to a absolute value, for instance 100.
 - Use 'Animate' and set the 'Style' to 'Modal'. Ok. The animation for this frequency is shown. From here you can manually extract the maximal deformation (in this case 0.0063mm).



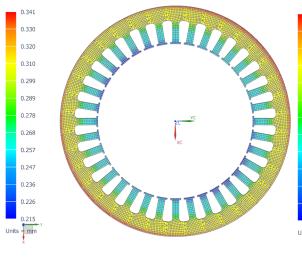
- Use the green buttons to cycle through the frequency results.
- 26. For validation of the computed results we show a table with frequencies and the corresponding maximum amplitudes. This information can be extracted from the NX NASTRAN 111 solution and should (for the NX 11 solution) be:

Frequency [Hz]	max. Amplitude [mm]
100.1	0.0048
200.2	0.0063
300.3	0.0143
400.4	0.3410
500.5	0.0043
600.6	0.0025
700.7	0.0019
800.8	0.0198
900.9	0.0024
1001	0.0060

Note: If NX 10 is used, slight variations might be present, due to the older NASTRAN functions.

It can be seen that at 400.4 and at 800.8 Hz there are maxima in the amplitude as was requested in the task. This results in the highest noise pressure at this frequency. Finally, we show the deformation shape and stress distribution for the resonant frequencies 400.4 Hz and 800.8 Hz.

Motor1FTG_assylem1_sim1 : Solution 1 from Load Recipe 1 Result Data Source 1 - Empty, Forcing Frequency 5, 400.4 Hz Displacement - Nodal, Magnitude Complex Option : At Phase Angle 0.0 Min : 0.215, Maix : 0.2411, Units = mm Deformation : Displacement - Nodal Magnitude, Complex Option : Amplitude



MotorIFT6_assyfem1_sim1 : Solution 1 from Load Recipe 1 Result Data Source 1 - Empty, Forcing Frequency 9, 800.801 Hz Displacement - Nodal, Magnitude Complex Option : At Phase Angle 0.0 Min : 0.00833867, Max : 0.019872, Units = mm Deformation : Displacement - Nodal Magnitude, Complex Option : Amplitude 0.019872 x. x. 0.0189109 0.0179497 0.0169886 0.0160275 0.0150664 0.0141053 0.0131442 0.012183 0.0112219 0.0102608 0.00929969 0.00833857 Units mm

11 References

• 'SantosAnthonisNaclerioGyselinck':

'Multiphysics NVH Modeling: Simulation of a Switched Reluctance Motor for an Electric Vehicle', Fábio L. M. dos Santos, Jan Anthonis, Francesco Naclerio, Johan J. C. Gyselinck, Member, IEEE, Herman Van der Auweraer, Member, IEEE, and Luiz C. S. Góes. IEEE TRANSACTIONS ON INDUSTRIAL ELECTRONICS, VOL. 61, NO. 1, JANUARY 2014.

• 'Eckhardt':

'Grundzüge der elektrischen Maschinen', Hanskarl Eckhardt, Teubner Studienbücher Elektrotechnik/Maschinenbau1982

• 'Fischer':

'Elektrische Maschinen', Rolf Fischer, Hanser Verlag 14. Auflage

• 'VAC':

'Magnetic Materials, Fundamentals, Products, Properties, Applications', Rainer Hilzinger, Werner Rodewald, VAC Vacuumschmelze

• 'Kost':

'Numerische Methoden in der Berechnung elektromagnetischer Felder', Arnulf Kost, Springer-Lehrbuch

• 'Steinmetz':

C. Steinmetz, On the law of hysteresis (originally published in 1892), Proceedings of the IEEE, vol 72, no. 2, pp. 197...221, 1984