

Electromagnetic coupled with Elasticity 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 DOCUMENTATION 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	Introduction	3
2	Deforming Conductor	4
2.1	Analysis of Magnetic Forces	4
2.2	Internal Elasticity Solver Usage	12
2.3	Transfer Magnetic nodal-Forces to Nastran Solver	15
2.4	Transfer Magnetic total-Forces to Nastran Solver	20
3	Dynamics of Three Conductors	22
3.1	File Structure Setup	22
3.2	Fem File Setup	23
3.3	EM Loads Setup - DC, AC, Short Circuit	24
3.4	Elasticity Constraints Setup	26
3.5	Solution: EM Static, Elasticity Static	26
3.6	Solution: EM Frequency, Elasticity Static	27
3.7	Solution: EM Frequency, Elasticity Frequency	28
3.8	Solution: EM Transient Short Circuit, Elasticity Transient	29
3.9	Solution: EM Transient Short Circuit, Elasticity Transient, Bidirectional	30
4	Elasticity Contact	33
4.1	Contact Concepts	33
4.2	The Start Model	34
4.3	Create Surface-to-Surface Contacts	35
4.4	Contact Parameters Overview	37
4.5	Solving and Monitoring	39
4.6	Post Processing Contact Results	40
4.7	Create Node-to-Node Contacts	40
5	Touching and Releasing of Conductors	42
5.1	The Start Model	42
5.2	Creating a Thin Gap with EM-Elasticity Coupling	46
5.3	Solving and Post Processing	47
6	Electromechanical Switch	49
6.1	The Demo Model	49
6.2	Elasticity Joint Connection	50
6.3	Handling of Elasticity Bolt Pre-Loads	50
6.4	Contacts with EM Elasticity Coupling	52
6.5	Additional Contacts to Avoid Penetration	53
6.6	Dynamic Solution for EM-Elasticity	53
6.7	Solve and Post Processing	54

1 Introduction

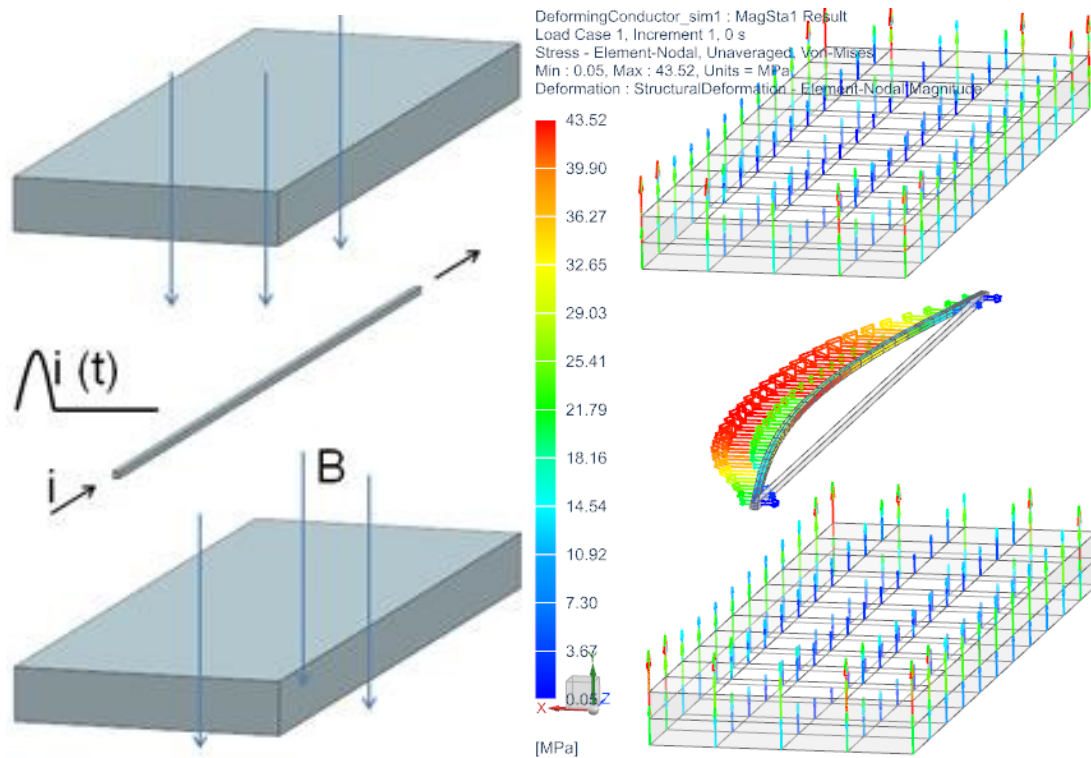
This guide shows different problems with electromagnetic elasticity coupling and their corresponding simulations. The electromagnetic part is solved using the Magnetics solver. The elasticity part is either solved by Magnetics and its integrated elasticity solver or by Simcenter Nastran.

2 Deforming Conductor

In this example we want to analyse for the elastic deformations that result from magnetic forces. A conductor is positioned in a magnetic field resulting from two permanent magnets. For checking purposes we first create a static solution and compare the Lorentz forces against theory.

In a time domain analysis we assign a half sinus current running through the conductor and solve for the Lorentz forces that result. The Lorentz forces on the conductor are computed for every time step and can be post processed as graph or plot.

There follows a transient dynamic elasticity analysis to find the conductors deformations, stresses and reaction forces. First we use the internal elasticity solver that is delivered with Simcenter/NX Magnetics. Because the solver is internally this process is very easy. Alternatively to the internal elasticity solver we also want to solve by Simcenter/NX Nastran. We apply fixed boundary conditions and import a file that contains the Lorentz forces. As result we get deformations and stresses.




Estimated time: 1.5 h.

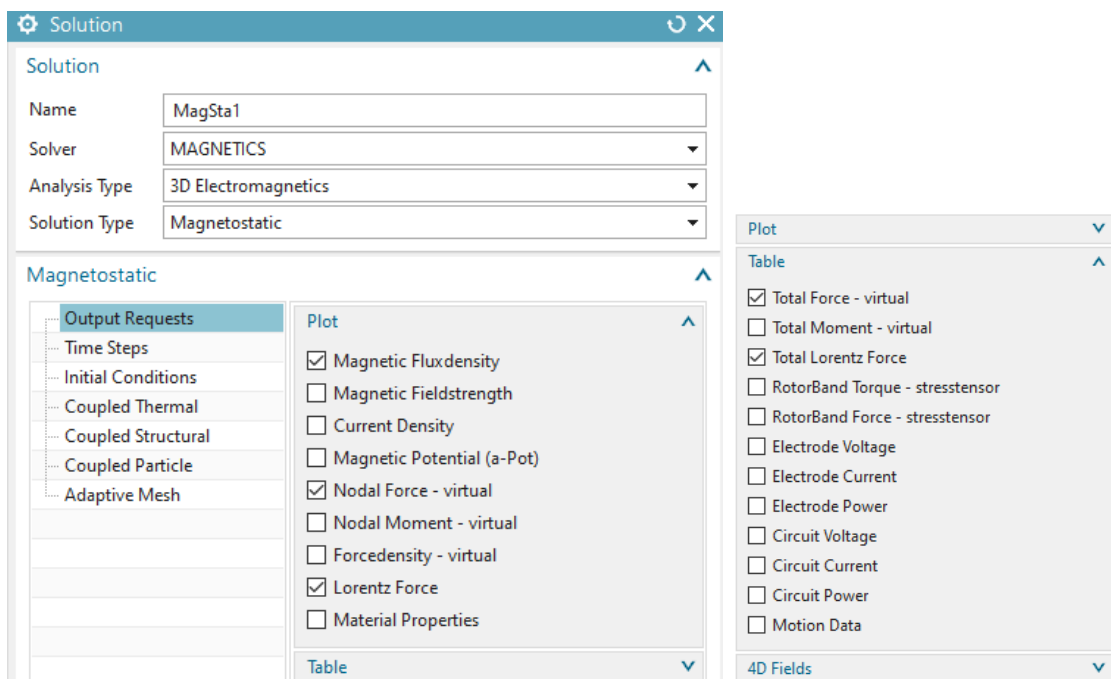
Follow the steps:

2.1 Analysis of Magnetic Forces

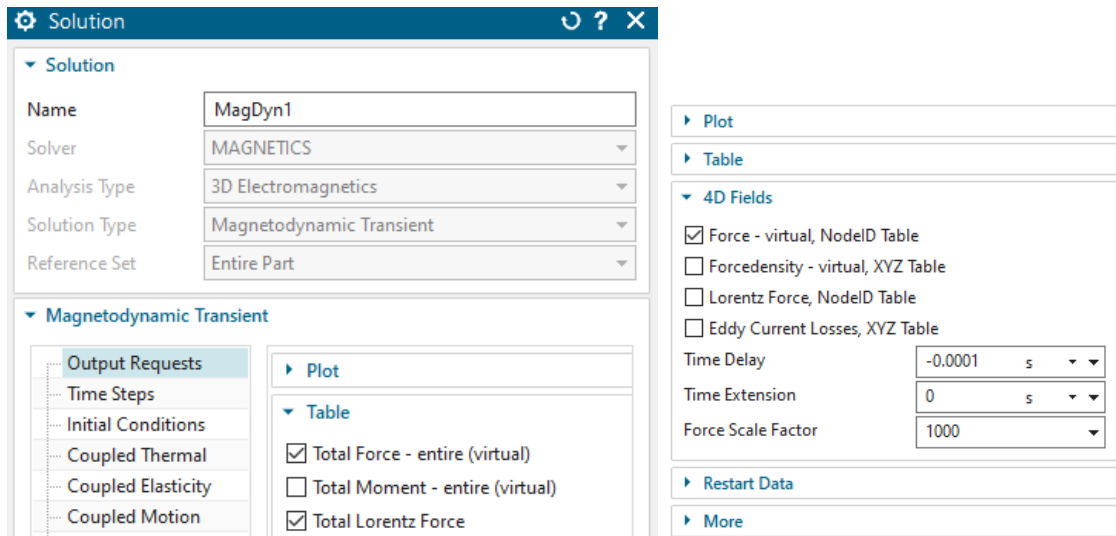
1. download and unzip the model files for this tutorial from the following link:
<https://www.magnetics.de/downloads/Tutorials/8.CouplStructural/8.1DeformingConductor.zip>

2. Start Simcenter, click Open  and navigate to folder 'start'. Select the file 'Deforming-Conductor.prt' and click OK.
3. Start application Pre/Post, Create a 'New Fem and Simulation', use Solver MAGNETICS and Analysis Type '3D Electromagnetics'. Switch off the 'Create Idealized Part'.
Hint: Because of the simple geometry, we don't use the 'Non-Manifold' method, so that we can take advantage of the hex elements, which enable precise results.

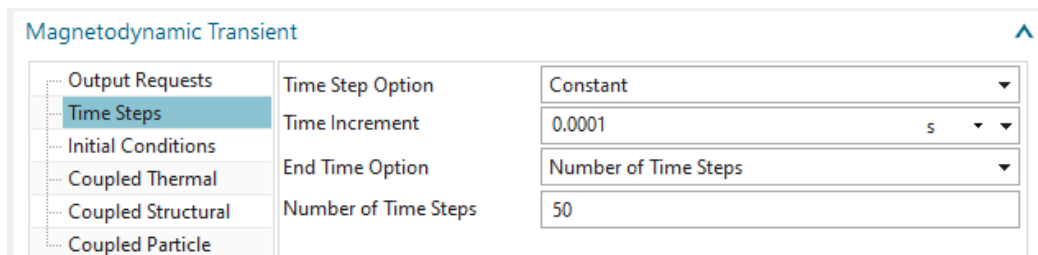
- Create a first Solution of Type 'Magnetostatics' and name it 'MagSta1'. (For checking purposes we first create a static solution.)
- In the Output Requests under box 'Plot' activate 'Nodal Force - virtual' and 'Lorentz Force' and under box 'Table' activate 'Total Force - virtual' and 'Total Lorentz Force'. You can also activate other types you are interested in. Click Apply.



- Create a second Solution of Type 'Magnetodynamic Transient' and name it 'Mag-Dyn1'.
- Activate again the 'Plot' output requests 'Nodal Force - virtual' and 'Lorentz Force'. This is for viewing the forces in the plot.
- Additionally activate again in the 'Table' box 'Total Force - virtual' and 'Total Lorentz Force'
- and in box '4D Fields' also activate 'Force-virtual, NodeID Table'. Here, at 'Time Delay' insert -0.0001 s. This setting will shift the magnetic forces by one time step. This way Nastran will activate the forces at the same time as Magnetics does.
- Additionally, key in the value 1000 into the field 'Force Scale Factor' because the forces must be converted from standard SI units to mN for further use in Simcenter as input.



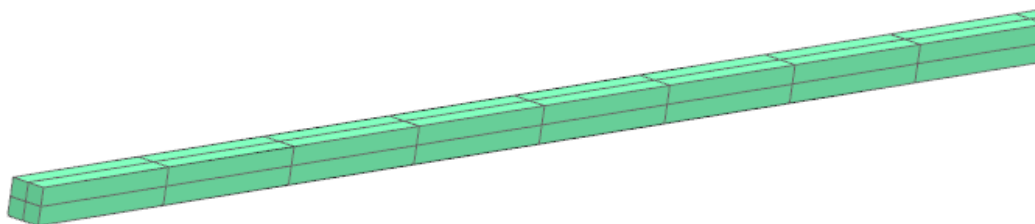
- Hint: The virtual energy method is a more complete analysis of forces since it also captures reluctance forces and those forces that result from permanent magnets. Such are not included in Lorentz forces, because Lorentz forces compute the vector product of electric current and magnetic flux density.
- In the Time Steps you set the 'Number of Time Steps' to 50 and the 'Time Increment' to 0.0001 s.



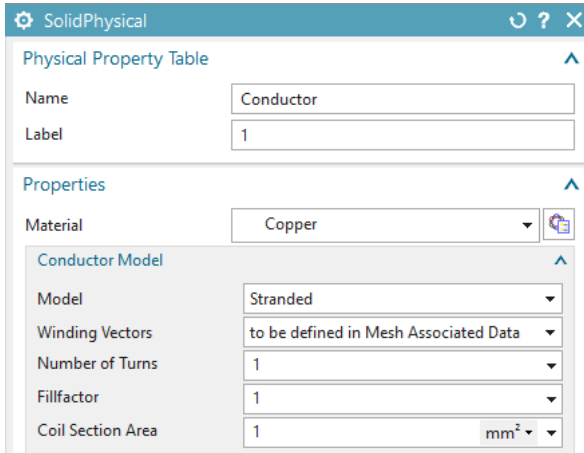
- Hint: The conductor has a natural frequency at 235 Hz. You can check this by a NX Nastran Solution 103. Because $1/235\text{Hz} = 0.004\text{s}$ our current will force the conductor to vibrate in natural frequency.
- Click OK

4. Switch to the Fem file

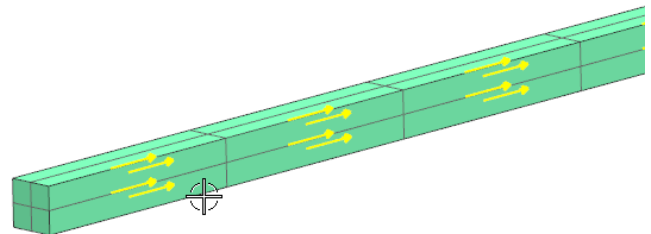
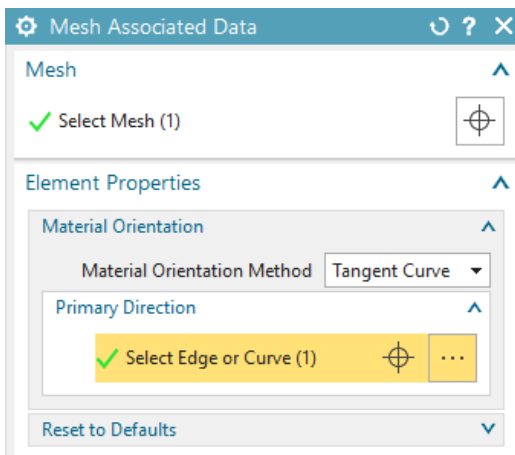
- Create automatic mesh mating conditions. There should be 16 conditions created.
- Mesh the conductor using hex elements. (Tet elements also work). Choose the suggested element size divided by 2, and 30 layers. Prior to meshing, set mesh controls on the four edges and force the mesh to include four elements in the section of the conductor (see picture below).



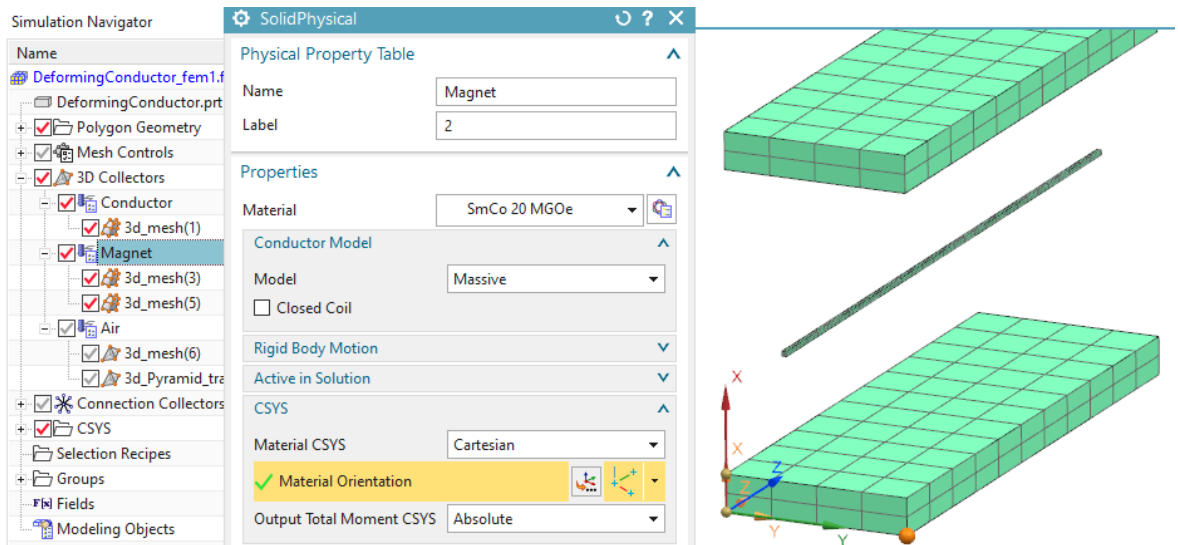
- Name the mesh collector 'Conductor' and assign material 'Copper' from the library to the conductor. Set the 'Conductor Model' to 'Stranded' and use the settings as shown in the picture.



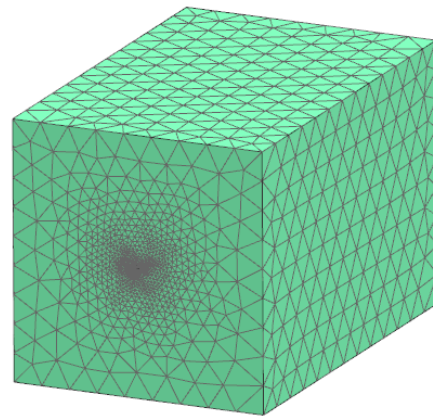
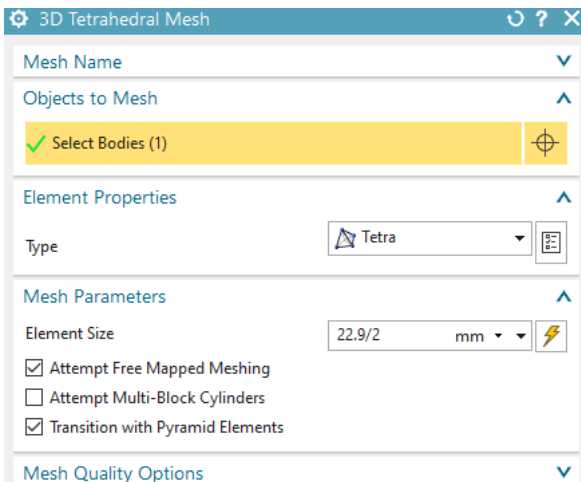
- Edit the 'Mesh Associated Data' of the conductor mesh and apply the direction of the windings. Select one of the edges of the conductor for this. Optionally press the 'Preview' button to see arrows pointing in positive current direction.




- Mesh the two magnets using hex elements (Others work too).
- Apply the material 'SmCo 20 MGOe'. This can be found in the material list in the NX installation folder.
- Set the Magnet CSYS to Cartesian and let the X direction of it point as shown in the below picture.



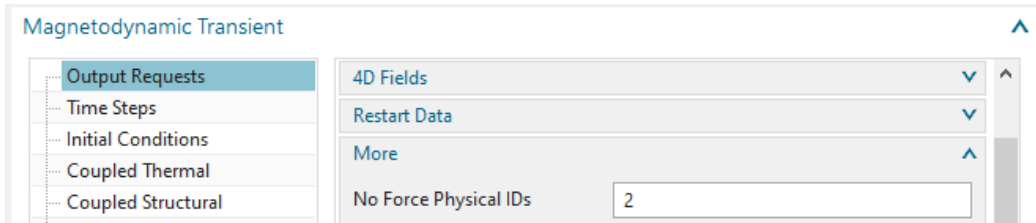
- Mesh the air volume using tets and the half of the suggested element size. Apply the button 'Transition with Pyramid Elements' if you used hex before.



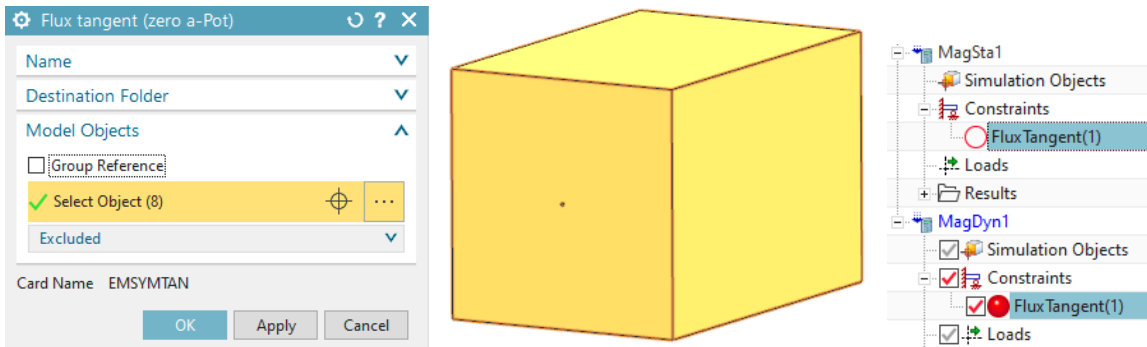
- Apply a 'FluidPhysical' and 'Air' material to this mesh collector.
- Because we want to exclude from the force computation all bodies but the conductor we must check the id labels of the Magnets physicals. We find they have id 2. We will use this in a following step in the Sim file.
- Select the button 'Rename Meshes and Physicals by Collectors' from toolbar 'Magnetics' . All meshes and Physicals will be renamed and post processing will be simplified.

5. Switch to the Sim file.

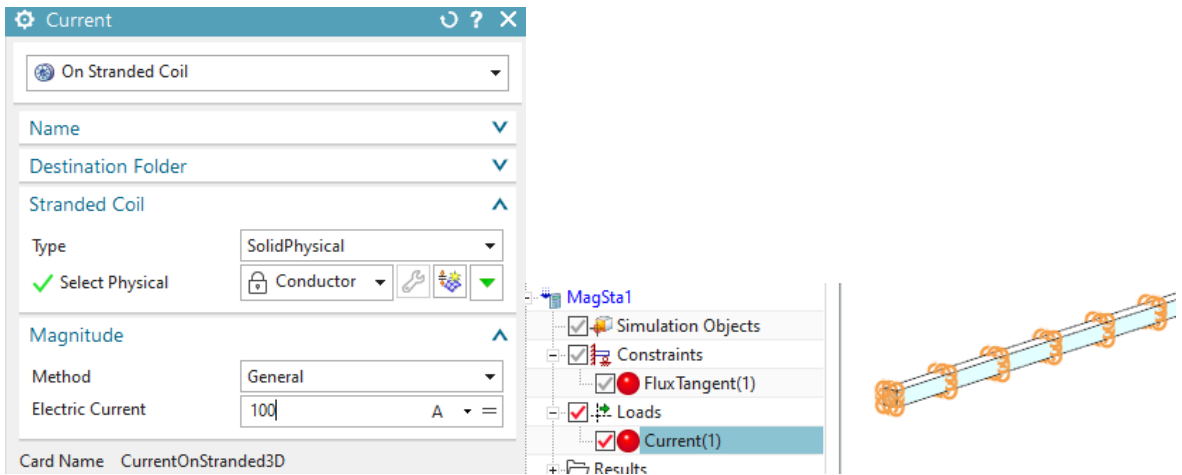
- Edit solution 'MagDyn1', in register 'Output Requests' box 'More' at 'No Force Physical IDs' assign 2 because we do not want force results on the magnets.



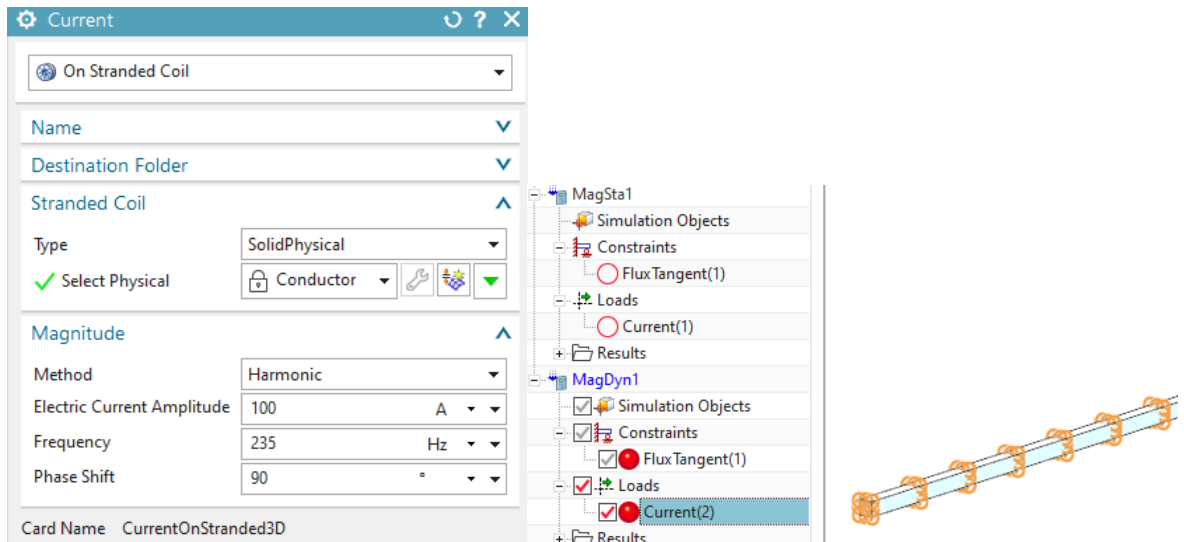
- Create a constraint 'Flux tangent (zero a-Pot)' on all 6 faces of the air volume. Also apply this to the two small electrode faces of the conductor. Apply this constraint to both solutions.



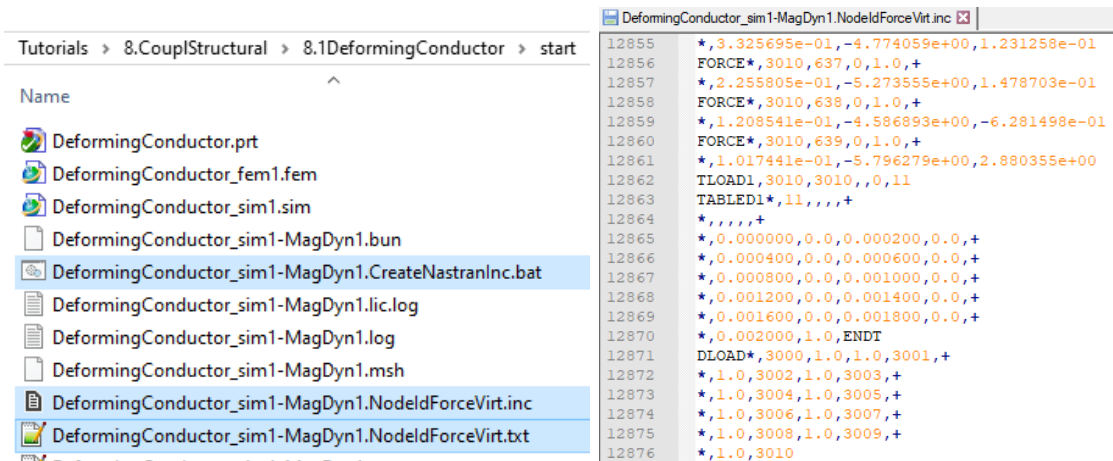
- Activate the static solution and Create a load of type 'Current', use the default type 'On Stranded Coil'. Select the Conductor physical. Insert an 'Electric Current' of 100 A.



- Activate the dynamic solution and Create a load of type 'Current', use the default type 'On Stranded Coil'. Select the Conductor physical and set the 'Method' to 'Harmonic'. Insert an 'Electric Current' of 100 A, a 'Frequency' of 235 Hz (remember, this is near the natural frequency) and a Phase Shift of 90 deg. (The Phase Shift changes the default cosines type load to sinus).

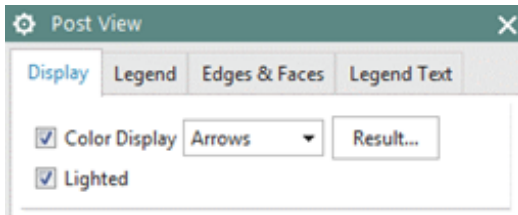


6. Solve both solutions
7. Notice that after the dynamic solution has completed, there is a new text file 'Deforming-Conductor1_sim1-MagDyn1.NodeIdForceVirt.txt' in the working folder. This file contains the force that will later be transferred to NX Nastran elasticity analysis. Notice also there is a batch file in the same folder, named 'DeformingConductor_sim1-MagDyn1.CreateNastranInc.bat'. This batch must be executed to convert the text file into the format that can be included into Nastran.
8. In the working folder, double click that batch file ('*.CreateNastranInc.bat'). There will be a file '.inc' created. This inc file contains in Nastran syntax FORCE entries assigned to nodes and to time steps. The DLOAD ID is set to 3000. We will later include this file in the Nastran input file.



9. Postprocess the static force results.
 - Open the results for the static solution and display the x component of the result 'LorentzForce'. (The same can be done for the virtual forces, witch are named 'Force'. Both results are very near.)
 - Blank the Air mesh and all 2D meshes.

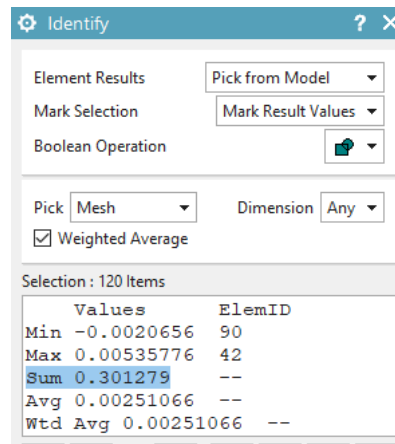
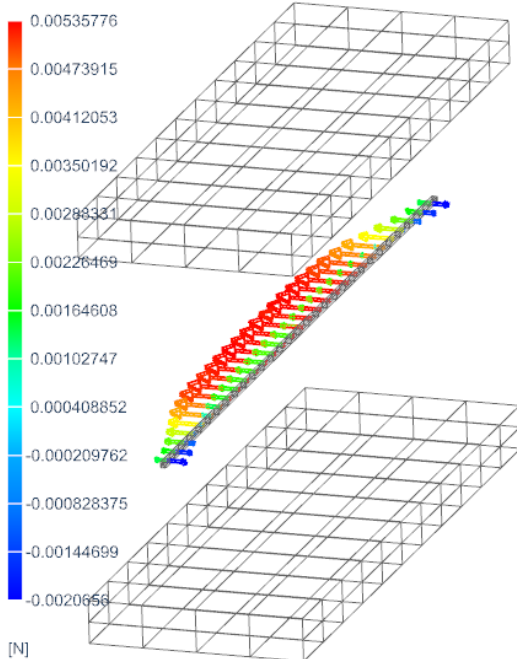
- Switch from 'Contour' to 'Arrows'.



- you should see a picture like below.

Hint: Depending on your current direction the direction of the force may be flipped.

DeformingConductor_sim1 : MagSta1 Result
 Load Case 1, Increment 1, 0s
 LorentzForce - Elemental, X
 Min : -0.0020656, Max : 0.00535776, Units = N
 Deformation : StructuralDeformation - Element-Nodal Magnitude



- To calculate the sum of all forces acting on the conductor you can now compute the sum of the element forces using the 'Identify Results' function ?
 - Use the 'Identify Results' function and set the 'Pick' to 'Mesh'. Select the conductor mesh and find the sum is about 0.3 N.
10. The analytical solution for the Lorentz force on the conductor can be found as follows:

$$\vec{F}_L = q(\vec{v} \times \vec{B}) = I(\vec{\ell} \times \vec{B})$$
 with

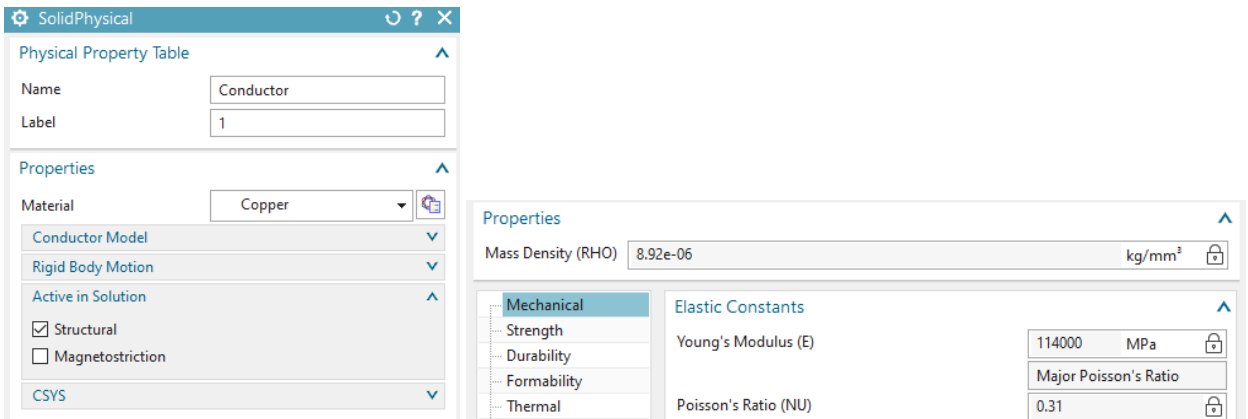
$$I = 100A, B = 0.03T, \ell = 0.1m$$
 the Lorentz force results to

$$F_L = 0.3N$$
 Though, the result from our simulation is close to this value.
 11. Post process the dynamic solution as you like.
 12. Save your parts. Don't close them.

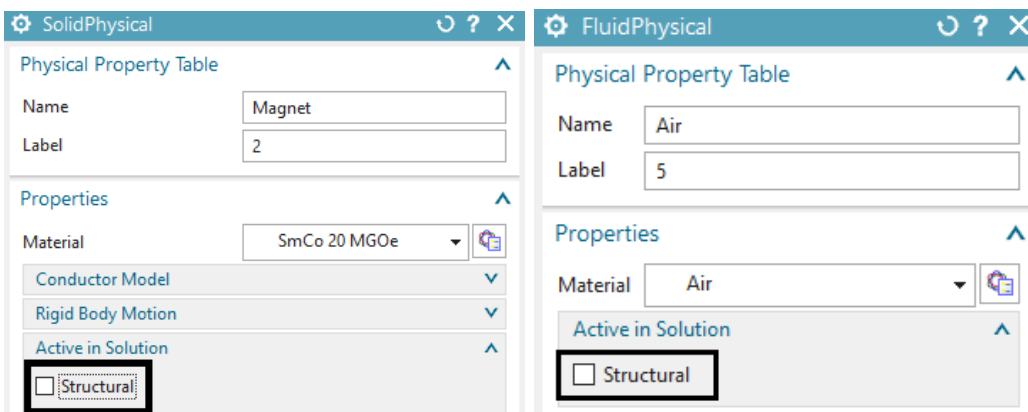
2.2 Internal Elasticity Solver Usage

The internal elasticity solver can handle 3D and 1D elements. Tetra and hexa elements are by default solved by second order nodal shape functions. Instead of mid nodes the elasticity solver uses the element edges and faces, so result quality is high and similar to NX Nastran second order hexa and tetra elements. Also pyramids and wedges are possible, but in first order only, so be aware that these elements are not very accurate. Also there are 1D rod elements available. The solver can be used for static and transient dynamic solutions. There is no nonlinear capability available. Following we set up the deforming conductor model to use this internal elasticity solver.

1. Set the Fem file to the displayed part.
2. Edit the conductor physical. In box 'Active in Solution' activate 'Elasticity' (this is already activated by default).
3. Also verify that the used material of the conductor has 'Mass Density', 'Young's Modulus' and 'Poisson Ratio' assigned.

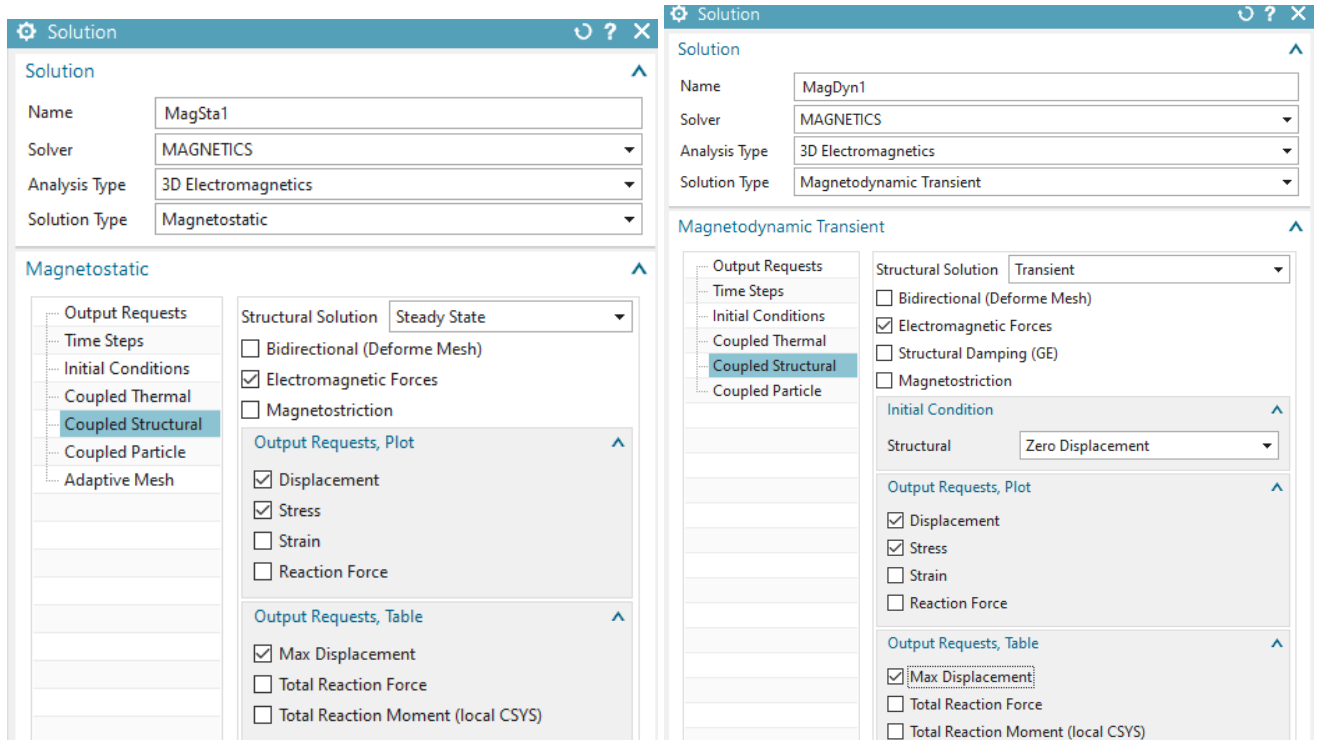


4. Check all other bodies for their setting 'Active in Solution'. This button must be deactivated. Otherwise we would have to assign materials and constraints to those bodies also and of course, the simulation would require much more time, memory and disk space.

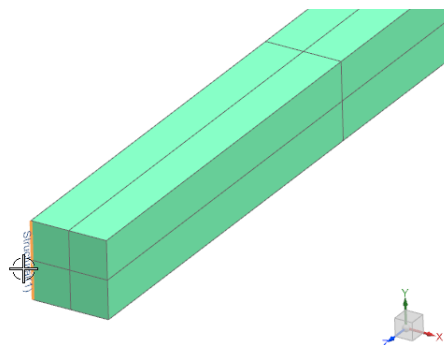
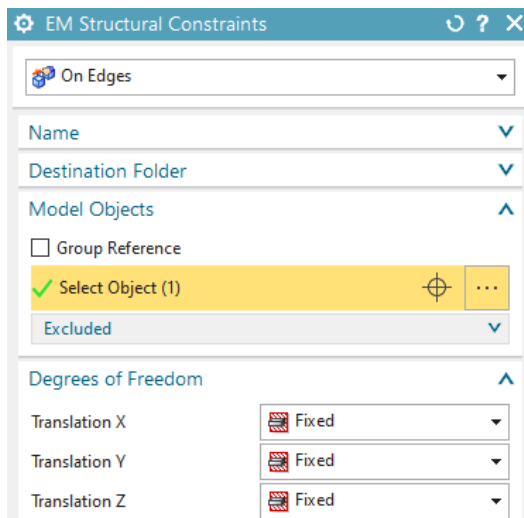


5. Make the Sim file to the displayed part.
6. Edit the static solution, in register 'Coupled Elasticity' set the 'Elasticity Solution' to 'Steady State'. Activate the settings as in the below picture.

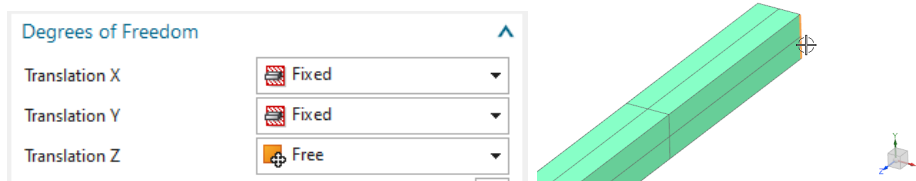
7. Edit the transient solution, in register 'Coupled Elasticity' set the 'Elasticity Solution' to 'Transient'. Activate the settings as in the below picture.



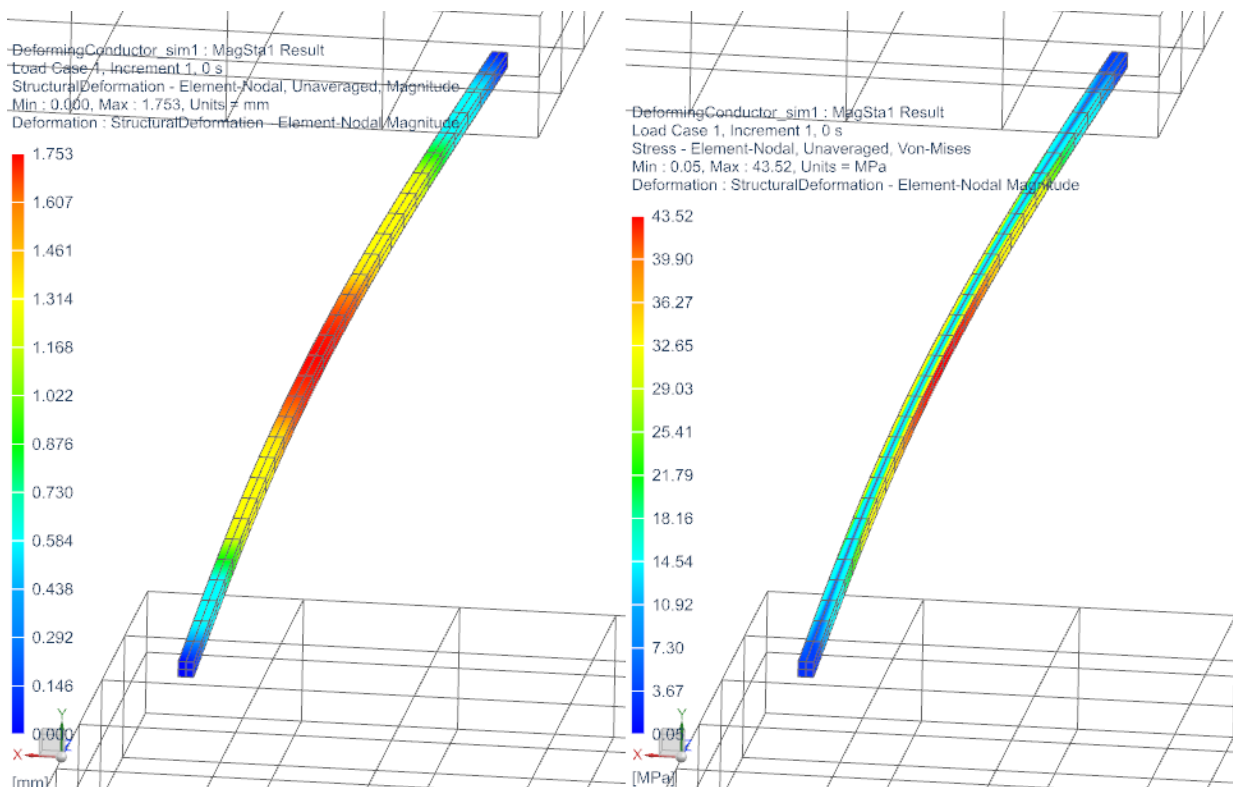
8. Create a constraint of type 'EM Elasticity Constraint', set the type to 'On Edges' and select the two edges of the conductor as shown below. These edges will allow the conductor to deform easily because rotation is allowed. Set all degrees of freedom to 'Fixed' and press OK.



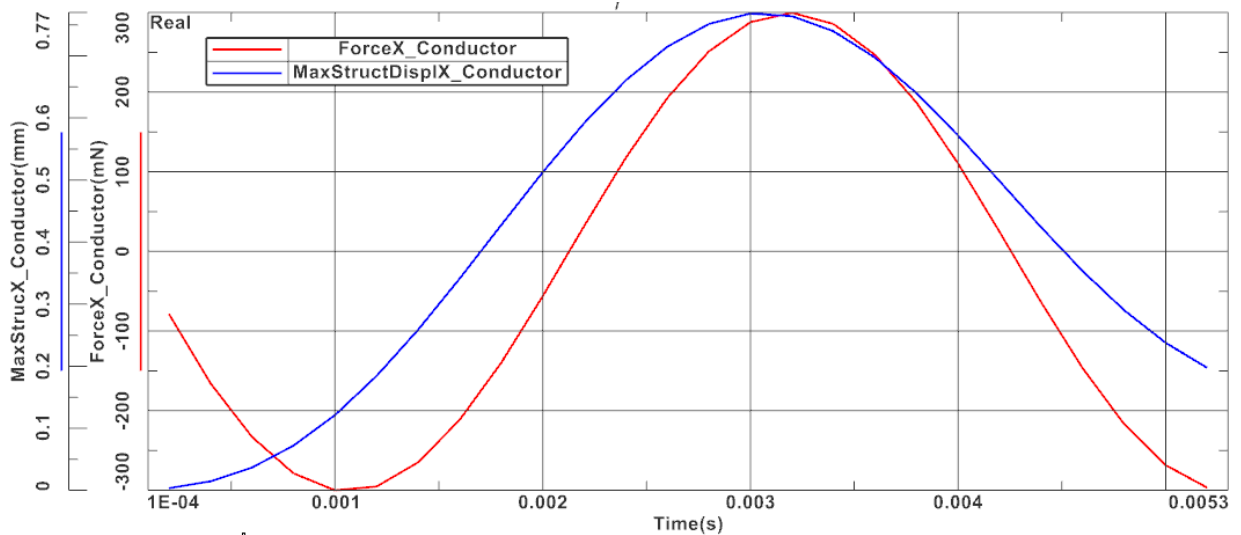
9. Create a second 'EM Elasticity Constraint' on on edge of the other side of the conductor. Fix only the x and y directions here.



10. Assign these two constraints to both solutions. Later we will also assign these constraints to the Nastran solution.
11. Solve both solutions
12. Post processing: The deformation (left) and von Mises stress (right) result of the static solution are shown in the below picture.



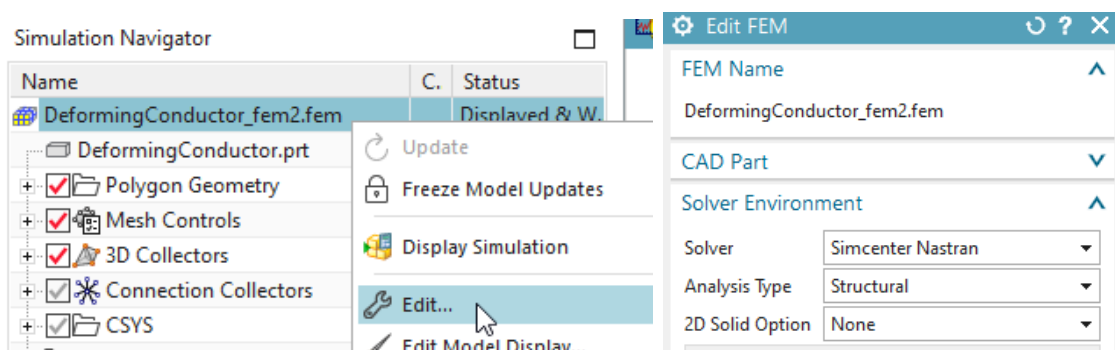
13. The maximum deformation over time (blue) and the force (red) from the transient solution are shown in the picture below. Both curves are in the AFU file, because they have been requested in the tabular output requests of the dynamic solution.



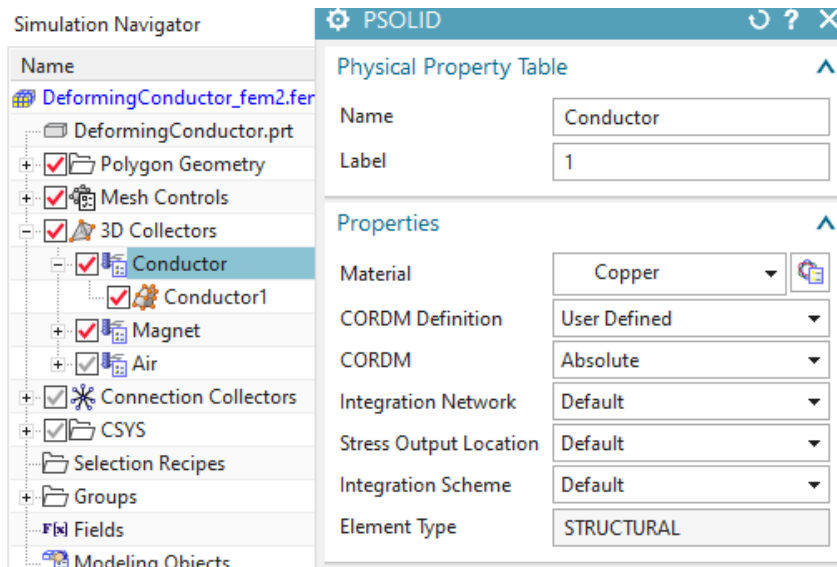
2.3 Transfer Magnetic nodal-Forces to Nastran Solver

We will now create a copy of the Fem and Sim files for the following Nastran elasticity analysis. Notice that in this analysis it is necessary to use the same node ID numbers as in the electromagnetic analysis. It is possible to add meshes. It is also possible to add mid nodes to the existing mesh for better stress results (we will do this here). Only it is not allowed to remove nodes or elements. In this case we will use a Nastran solution 109 to include transient dynamical effects. Other Nastran solution types would also work corresponding to their capabilities. The results will match with those from the previous section where the internal elasticity solver of Magnetics was used. The advantage of using Nastran is in many additional functions (contact, nonlinear materials, ...) that Nastran can handle.

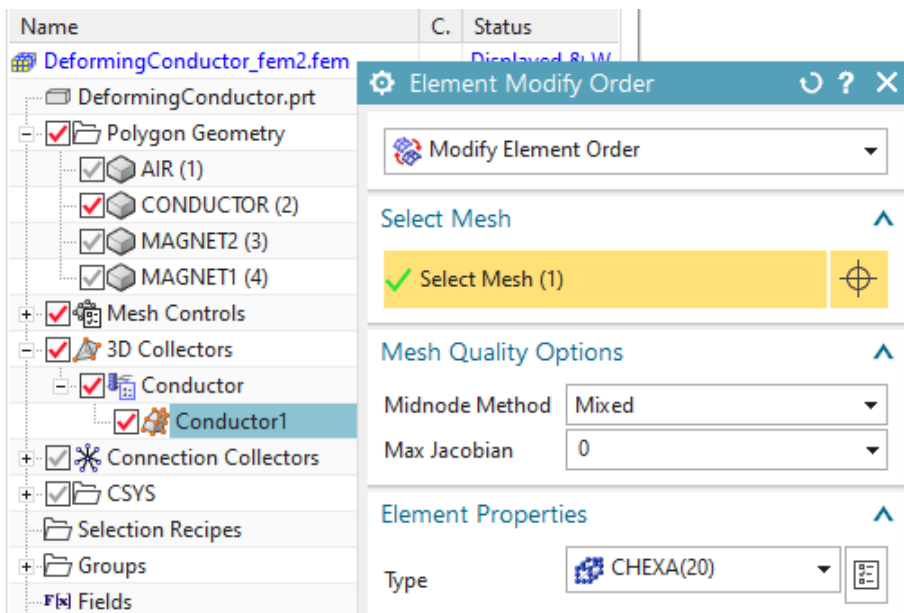
1. Make a copy of the Fem and Sim files. Therefore, first save the files,
2. then set the Fem file to the displayed part,
3. Do a 'Save As' and assign the name 'DeformingConductor_fem2.fem'
4. The system also asks for a new Sim file. Key in 'DeformingConductor_sim2.sim'.
5. A 'Save As' window appears. Press 'Yes'.
6. In the Fem file:
 - Edit the FEM file and set the Solver to NX/Simcenter Nastran, OK.



- Edit the physical property of the conductor. Notice that the type of property automatically has changed to PSOLID, because you switched to Nastran. The material 'Copper' is still there and since this material has young's modulus, poisson ratio and density available, it is valid also for this solver.

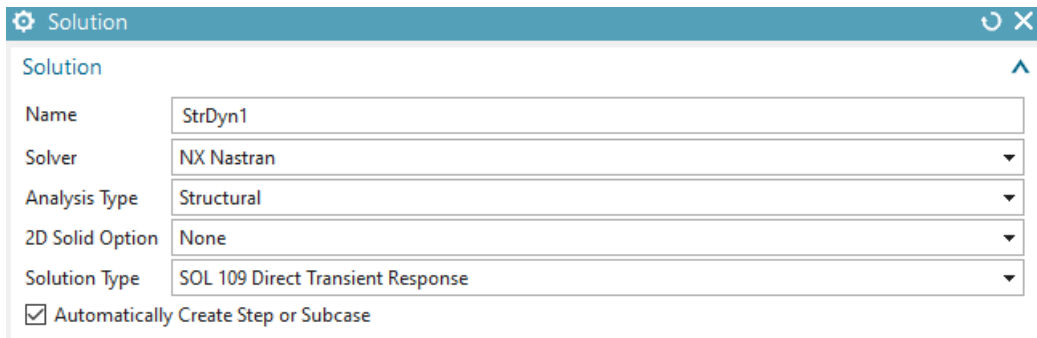




- Delete all other 3D meshes for this elasticity analysis.
Hint: This deleting is not absolutely necessary since the solver would accept the meshes.
- We want to use the similar shape functions in Nastran and Magnetics. Therefore, change the mesh to Hex20: Use the function 'Order' and use the type 'Element Modify Order' from toolbar 'Nodes and Elements' and set the mesh to Hex20.
Hint: Remember, it is possible to add nodes to the model. But it is not allowed to change the numbers of the existing nodes on which we have computed forces and which are written into the include file.

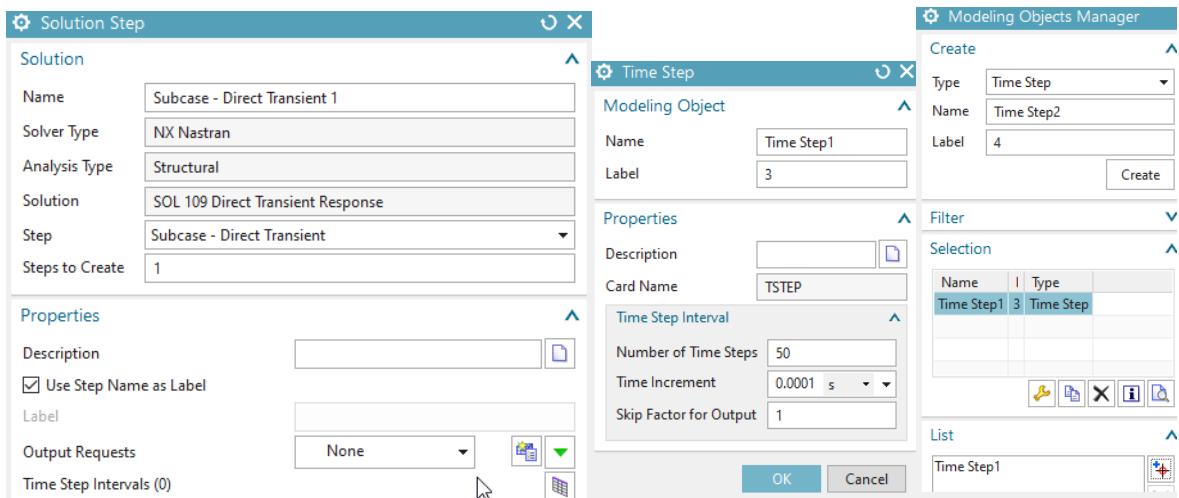


7. Switch to the Sim file.

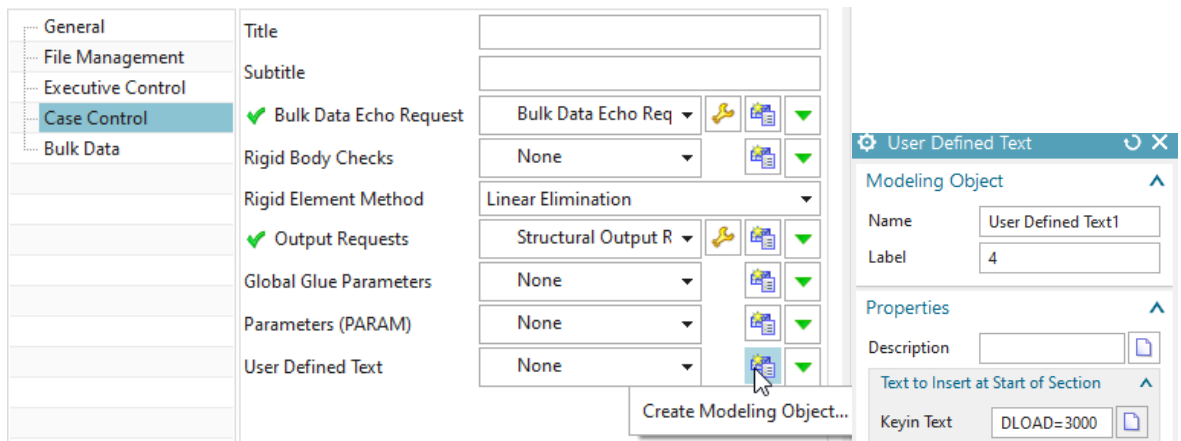
8. Optionally delete the two existing solutions 'MagSta1' and 'MagDyn1' and also the old loads. Keep the constraints because we can use them here.
9. Create a new solution of type 109 with solver Simcenter Nastran and name it 'StrDyn1', (other transient NX Nastran solution types are also possible) Click 'Create' then OK. Click 'Create' again.



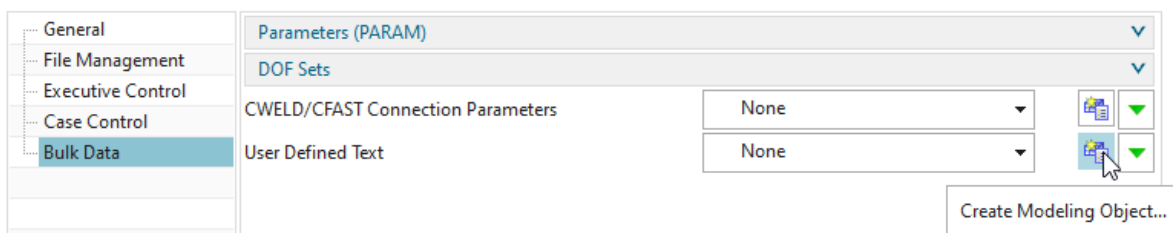
10. In the following dialogue 'Solution Step' click 'Create'  for 'Time Step Interval'. In the 'Modeling Solution Manager' click 'create' again and set the 'Number of Time Steps' to 50 and the 'Time Increment' to 0.0001 s. Click OK. Add the newly created Time Step object to the list . Click Close and OK.



11. To add the magnetic forces, witch reside in the text file with extension 'inc', proceed as follows:
 - Edit the solution. In the 'Case Control' section, create a modeling object for 'User defined Text'. At 'Keyin Text' insert 'DLOAD=3000'. This is the group of electro-magnetic forces that will be included.

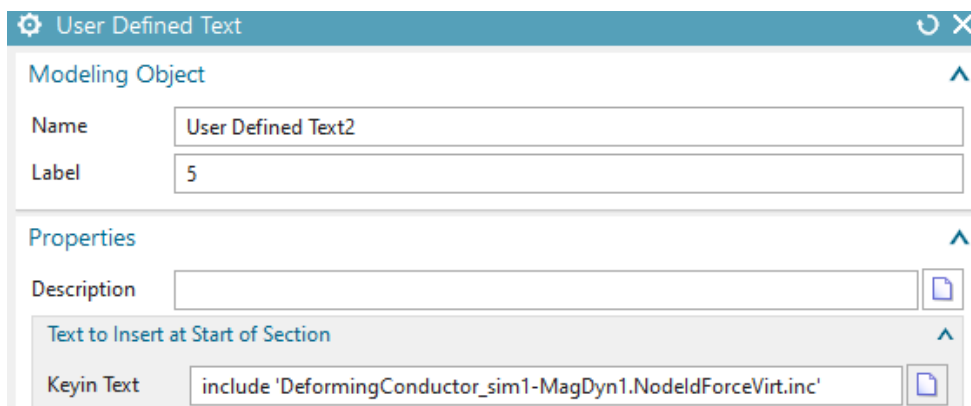


- In register 'Bulk Data': Again create a modeling object for 'User Defined Text'.



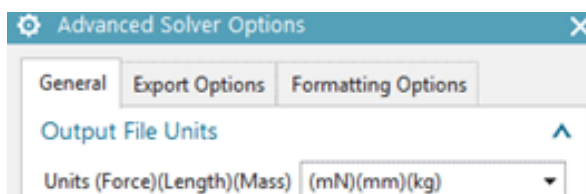
Key in the text line that points to the include file for forces:

include 'DeformingConductor_sim1-MagDyn1.NodeIdForceVirt.inc'

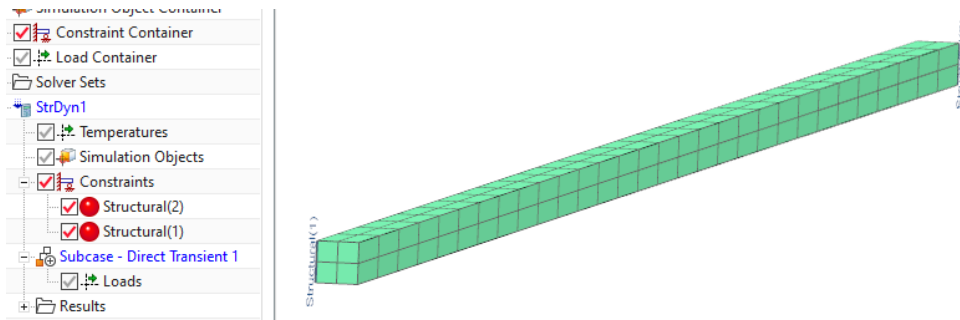


Click Ok, Ok.

12. Because the force units in the include file are written in mN we have to check that these units are used by the Nastran solution also. Edit the Advanced Solver Options (RMB on solution) of this solution and switch to register General. Ensure that the Output File Units are set to (mN)(mm)(Kg).

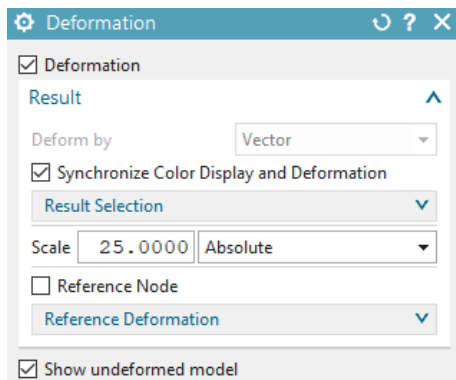


13. Reuse the two constraints that were used for the electromagnetic analysis with deformation.

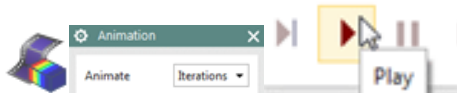


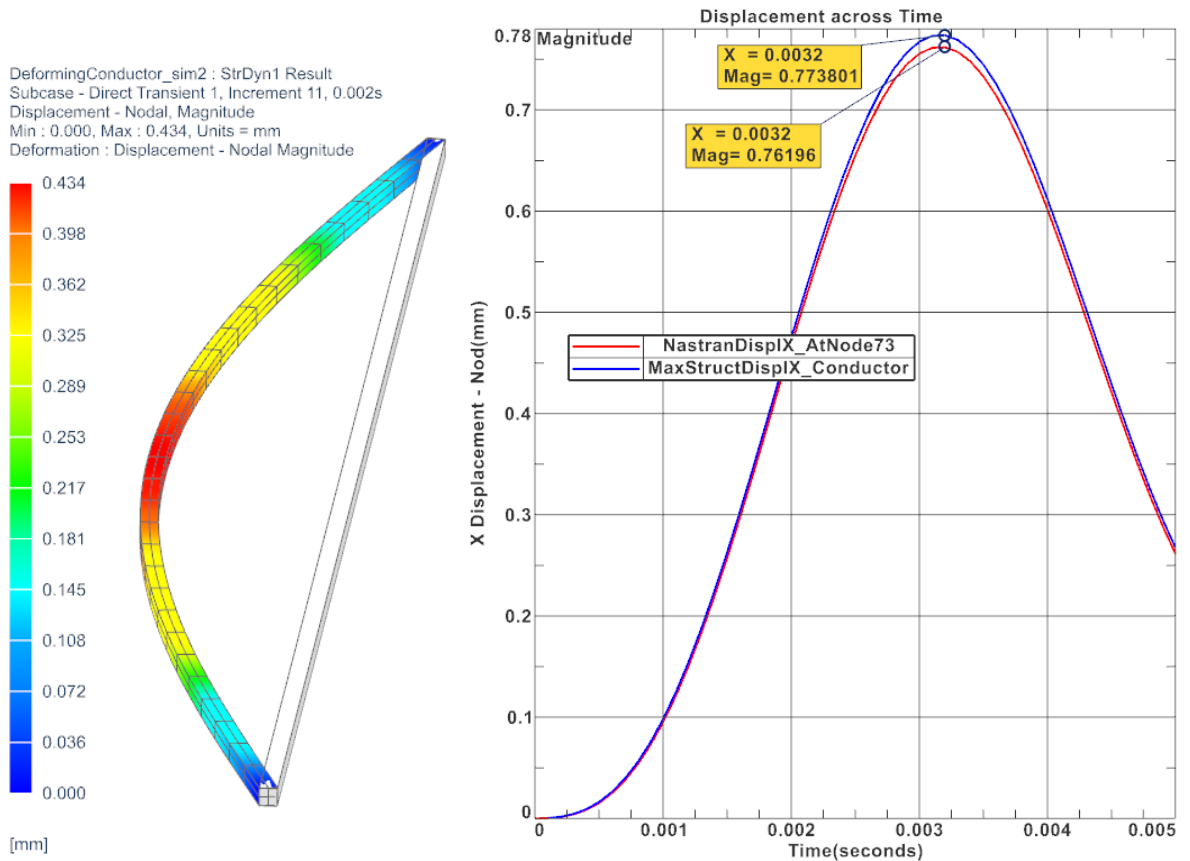
14. Solve the solution and postprocess the results.

- Display the displacement result of the first Increment.
- Set deformation to 'Absolute' and 'Scale' to 25. Activate 'Show undeformed model'



- Create an animation over the iterations. It should be seen a deformation that results from magnetic forces (picture below left).
- Picture below right shows a comparison of the maximum x displacement between the Nastran (red) and the Magnetics (blue) solver results. The deviations between the results of the two solvers are considerably small.





15. Save your parts and close them.

2.4 Transfer Magnetic total-Forces to Nastran Solver

Another possibility to transfer magnetic forces to Nastran uses the total forces that are computed on each physical body. This allows the force transfer between different models: Meshes can be different and even 2D and 3D types can be mixed. This method has been designed to work best with large assemblies and transient solutions in which the nastran model contains 2D meshes while in electromagnetic 3D are present. The process runs in the following steps:

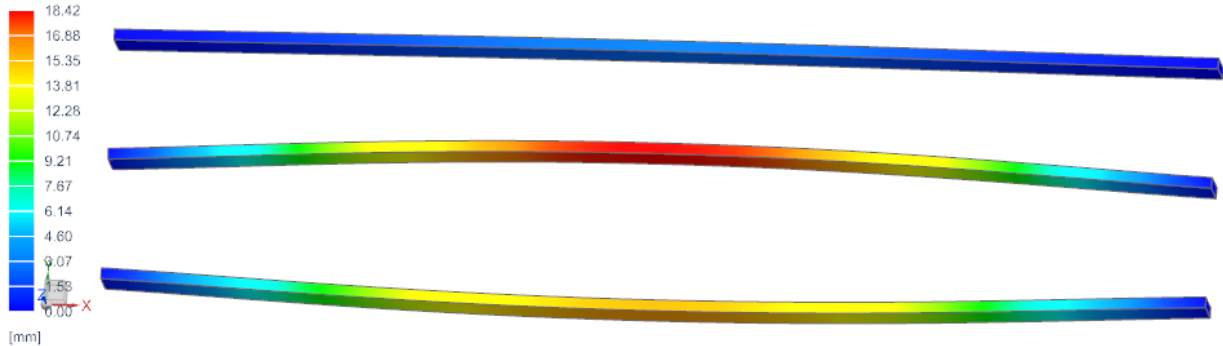
1. Initial situation: The electromagnetic solution is solved with one or more bodies. The tabular forces are requested, computed and written into the afu graph file. The graph names always show the force-direction and the physical body on which it is applied. Such names are typically as 'ForceX_Conductor1'.
2. The user now creates a Sim and Fem file for the nastran solution. All meshes, properties can be different. Only the names of the polygon bodies in that new model must match with those names from the electromagnetic solution. In the simplest case that new model is a copy of the electromagnetic model or we simply stay in that.
3. The user executes the feature 'Afu-Forces to Nastran' (found in Menu, Analyses) and will be asked for an Afu file. He assigns the one from the electromagnetic solution.
4. the program cycles through all graphs in that file

5. it checks graph names: If either 'Force' or 'LorentzForce' is found it extracts the force direction and the physical body name from it.
6. for each force, the program creates
 - a field from that graph.
 - a selection-recipe that searches for a corresponding polygon-body name. From that body all faces will be selected.
 - Three forces for the three directions that use the selection-recipe and the field.
7. all newly created forces are stored in a temporary nastran solution called 'tmp1'. From there they can be taken into the correct solution via 'drag and drop'.

The tutorial is finished.

3 Dynamics of Three Conductors

The 3 phase conductor system we analyse in this tutorial is loaded by either DC, AC or by a formula based short circuit condition. We want to analyse for the electromagnetic fields and mechanical deformations, stresses and reaction forces that result from said three load types.



Due to Amperes law, each conductor will create a rotating magnetic field around himself, that also influences each other conductor. The smaller the distance between them, the higher is this effect. In case the conductors deform large, the distance between them will either increase or decrease and thus, the produced fields will also change. This nonlinear effect can be captured by bidirectional simulation: The EM fields modify the geometry, the changed geometry results in a change of the EM field.

So, we will first create the Fem model with meshes and material properties, nothing very special. Then we will create solutions for electromagnetics with static (DC load), frequency (AC load) and transient (short circuit load). For each solution we will add coupled elasticity and finally we will run a solution with bidirectional coupling and compare the results.

Estimated time: 1.5 h.

Follow the steps:

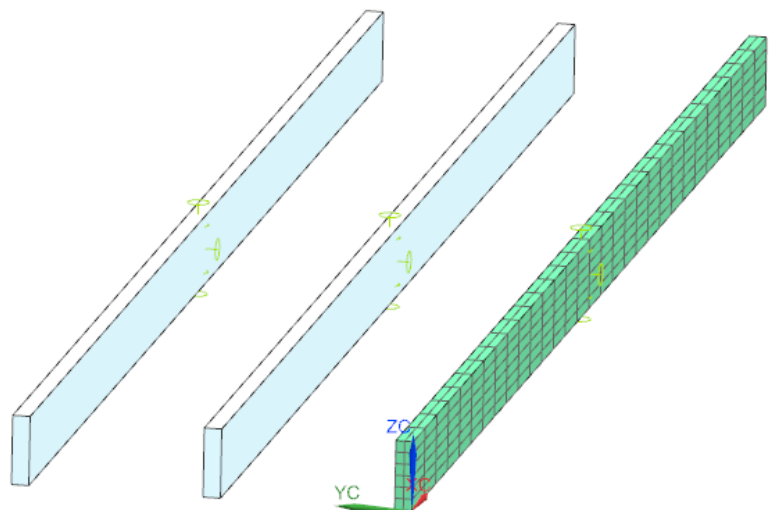
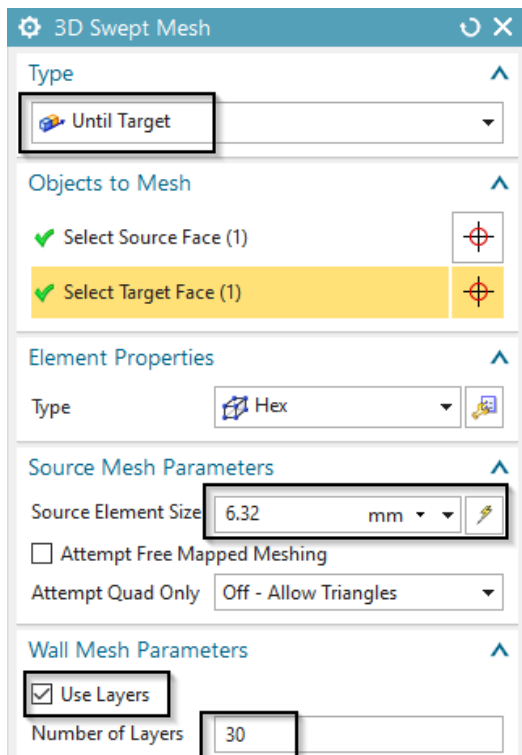
3.1 File Structure Setup

1. Download the model files for this tutorial from the following link:
<https://www.magnetics.de/downloads/Tutorials/8.CouplStructural/8.2ThreeConductors.zip>
2. Open the file 'ThreeConductors.prt'.
3. Start Simcenter Pre/Post, Create a 'New Fem and Simulation', use Solver MAGNETICS and Analysis Type '3D Electromagnetics'. Switch off the 'Create Idealized Part'. Create five solutions with the following types and names:
 - First solution:
Set type to 'Magnetostatic', name 'MagStaDC_ElasSta'.
The name will show the characteristics of the used physics: Electromagnetics will be set to static (MagSta) with direct current (DC), elasticity will also be set to static (ElasSta). We will fill up all solutions with details later.

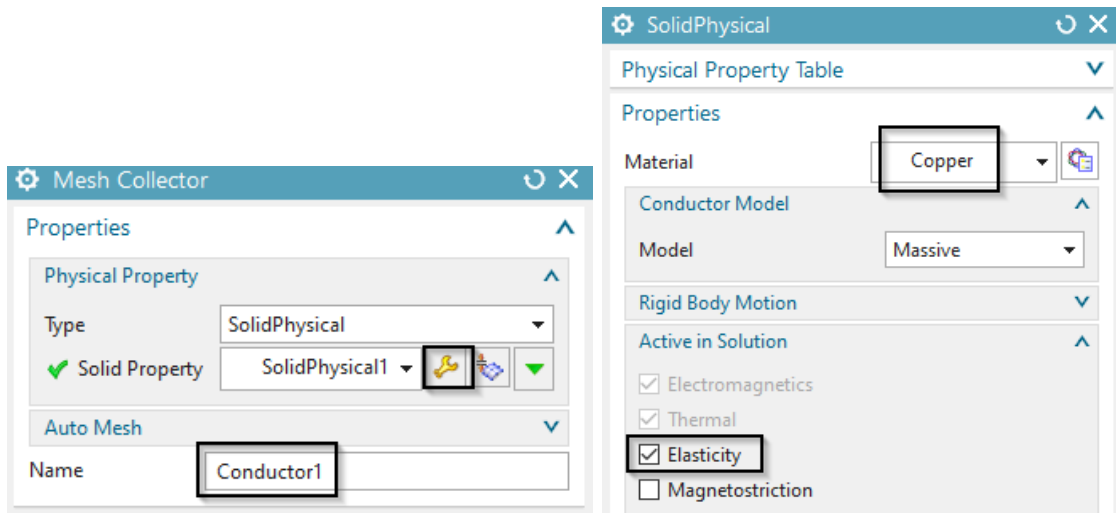
- Second solution:
Set type to 'Magnetodynamic Frequency', name 'MagDynFreqAC50Hz_ElasSta'.
Set the 'Forcing Frequency' to 50 Hz. So, we want to use a frequency domain solution for electromagnetics with AC current at 50 Hz and a steady state solution for the elasticity part.
- Third solution:
Set the type to 'Magnetodynamic Frequency', name 'MagDynFreqAC50Hz_ElasFreq'.
Set the 'Forcing Frequency' to 50 Hz. The extension 'ElasFreq' means that we want to use a frequency response elasticity solution.
- Forth solution:
Set the type to 'Magnetodynamic Transient', name 'MagDynTrans_ShortCircuit'. We will use a transient solution for both electromagnetics and elasticity. The current will be a short circuit type.
- Fifth solution:
Set the type to 'Magnetodynamic Transient', name 'MagDynTrans_ShortCircuit_BiDir'. Same as above, but we want to activate the bidirectional coupling.


3.2 Fem File Setup

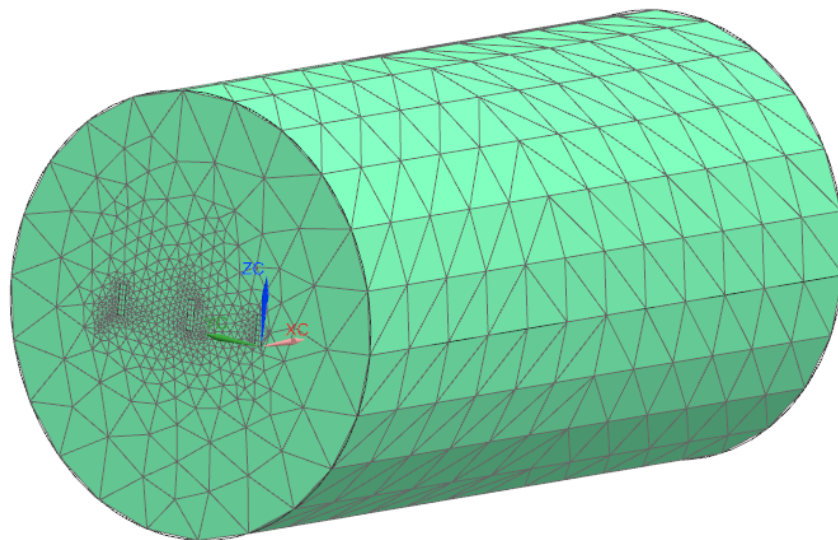
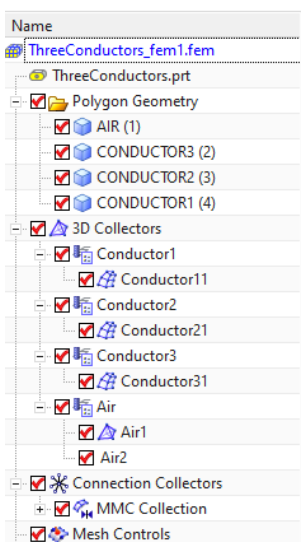
1. Change the displayed part to the Fem file.
2. Create mesh mating conditions. There should be 12 such conditions created. Blank the Air polygon body.
3. Mesh the 'Conductor1' with Hexaedral elements. Use the suggested element size (6.32 mm). Activate 'Use layers' and set the 'Number of Layers' to 30.



- Edit the newly created mesh collector and his Physical. Set the material to 'Copper' from the Magnetics material library. Check that in the setting 'Active in Solution' the option 'Elasticity' is on.



- In the same way create meshes and Physicals for the remaining two conductors.
- Unblank and mesh the Air body. Use tetrahedral elements and the suggested element size (129 mm). Because of the hex, tet-transition, activate the option 'Transition with Pyramid Elements'.
- For the Air mesh, create a 'FluidPhysical' and material 'Air' from the Magnetics material library.
- Select the button 'Rename Meshes and Physicals by Collectors' from toolbar 'Magnetics' . All meshes and Physicals will be renamed and post processing will be simplified.



3.3 EM Loads Setup - DC, AC, Short Circuit

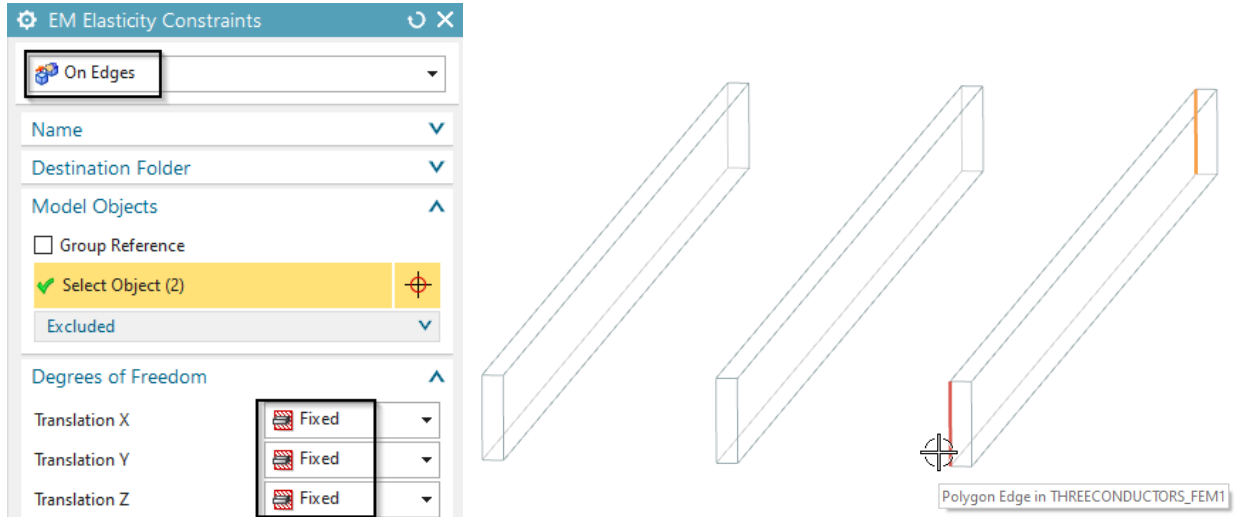
- Change to the Sim file.

2. Blank the 3D Collectors for easier selection. Activate (double click) the 'MagStaDC_ElasSta' solution.
3. Create a constraint of type 'Flux tangent (zero a-Pot)' on all 9 outside faces. Also the 6 electrode faces of the 3 conductors must belong to this group. This constraint must be used in all solutions, so drag-drop it into the others.
4. Create a Voltage load: Set the type to 'On Solid Face', key in 0 V and select the 3 electrode faces of the 3 conductors (no matter which side). This voltage load must be active in all solutions (all but the frequency solution), so drag-drop it into the such. For the frequency solution, create a separate one in the same way.
5. In the active solution 'MagStaDC_ElasSta', create three 'Current' loads as follows:
 - Set the type to 'On Solid Face' and the 'Method' to 'Harmonic'.
 - For 'Electric Current Amplitude' key in 50000 A, for 'Frequency' 50 Hz and for 'Phase Shift' 0 degrees.
 - Select the electrode face of conductor 1. Click OK.
 - Create a similar load on the second conductor with a 'Phase Shift' 120 degrees.
 - Create a similar load on the third conductor with a 'Phase Shift' 240 degrees.
6. Activate solution 'MagDynFreqAC50Hz_ElaSta' and create three 'Current' loads as follows:
 - Set the type to 'On Solid Face' and the 'Definition' to 'Amplitude/Phase'.
 - For 'Electric Current Amplitude' key in 50000 A and for 'Phase Shift' 0 degrees.
 - Select the electrode face of conductor 1. Click OK.
 - Create a similar load on conductor 2 with a 'Phase Shift' 120 degrees.
 - Create a similar load on conductor 3 with a 'Phase Shift' 240 degrees.
7. Activate solution 'MagDynTrans_ShortCircuit' and create three 'Current' loads with analytical formulas to describe the exponential behaviour of a short circuit as follows:
 - Set the type to 'On Solid Face' and the 'Method' to 'Analytic'.
 - Key in the following formula:

$$\text{Sqrt}[2]*2e4*(\text{Sin}[2*\text{Pi}*50*\text{\$Time}-1.57]+\text{Exp}[-\text{\$Time}/0.0455]*\text{Sin}[1.57])$$
 - notice the '1.57' that appears two times: This describes the phase shift for the 3 phase current system.
 - Create a second and a third Current load on the electrodes faces of the second and third conductors. On the second, use a phase shift of $1.57+2*\text{Pi}/3$ and on the third use $1.57+4*\text{Pi}/3$
 - Drag-drop these three analytical loads into solution 'MagDynTrans_ShortCircuit_BiDir'.

3.4 Elasticity Constraints Setup

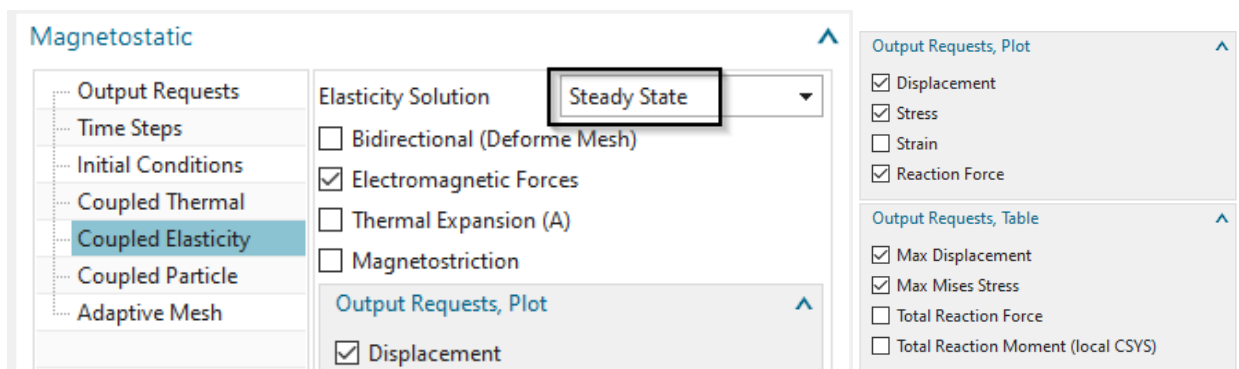
1. Activate (double click) the 'MagStaDC_ElasSta' solution.
2. Create a 'EM Elasticity Constraint' on the first conductor. Set the type to 'On Edges' and select the shown two edges. Set all 'Degrees of Freedom' to 'Fixed' and click OK.



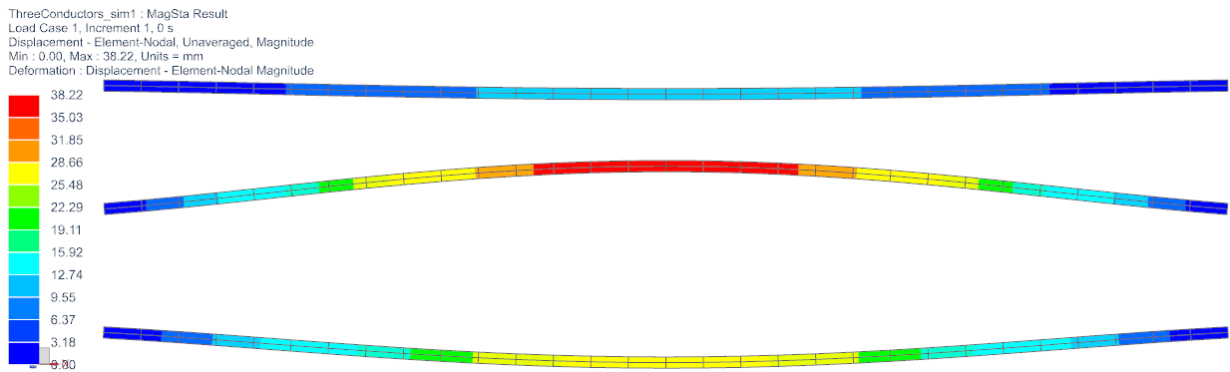
3. For the remaining two conductors, create constraints in the same way.
4. Drag-drop the three elasticity constraints into all other solutions.

3.5 Solution: EM Static, Elasticity Static

1. Activate solution 'MagStaDC_ElasSta'.
2. In register 'Output Requests' activate results as desired.
3. In register 'Coupled Elasticity', set the 'Elasticity Solution' to 'Steady State' and activate the 'Output Requests' as shown in the picture below.



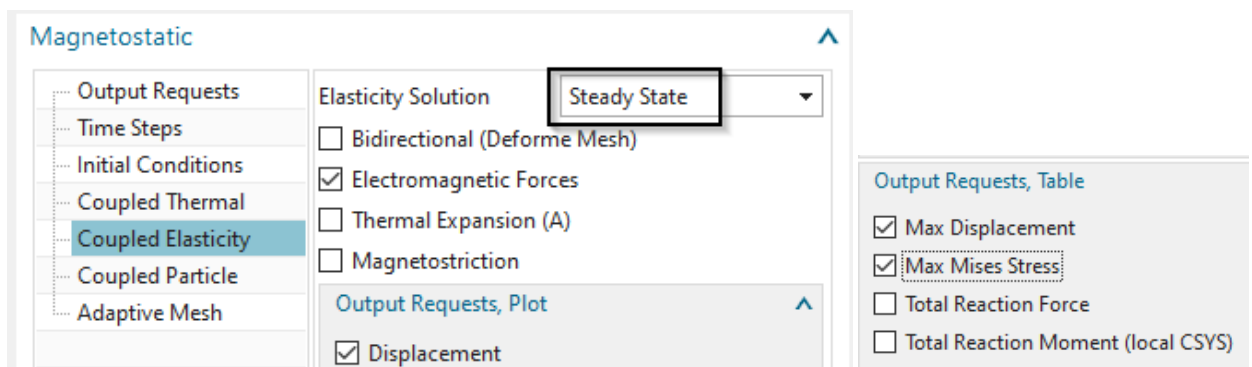
4. Solve the solution.
5. Display the Y displacement results. The maximum is 38 mm.



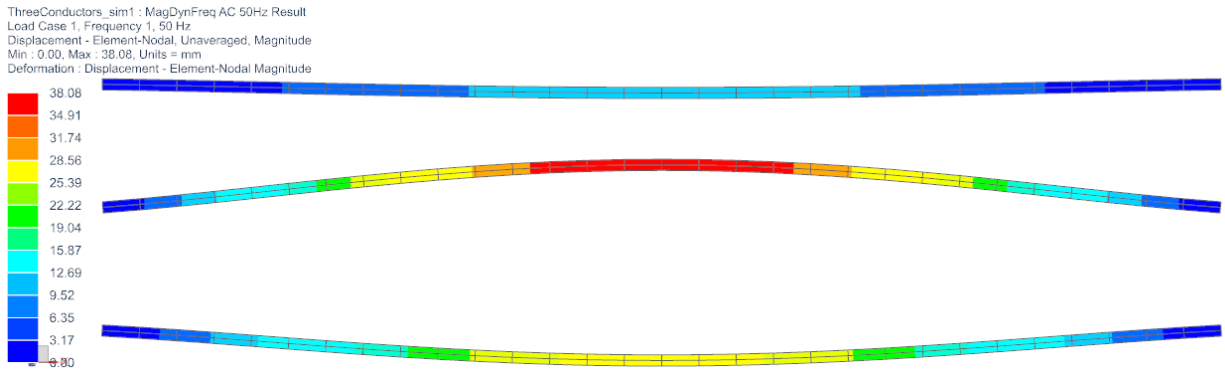
3.6 Solution: EM Frequency, Elasticity Static

In frequency domain solutions all loads and results are assumed to be harmonic. So, all results oscillate between phase angle 0 (real part), angle 90 (imaginary part), 180 and 270. Therefore, also the resulting mechanical forces come out as oscillating. To use the forces in the following elasticity solution, they are evaluated at phase angle 0.

1. Activate solution 'MagDynFreqAC50Hz_ElaSta'.
2. In register 'Output Requests' activate results as desired.
3. In register 'Coupled Elasticity', set the 'Elasticity Solution' to 'Steady State' and activate the 'Output Requests' as shown in the picture below.

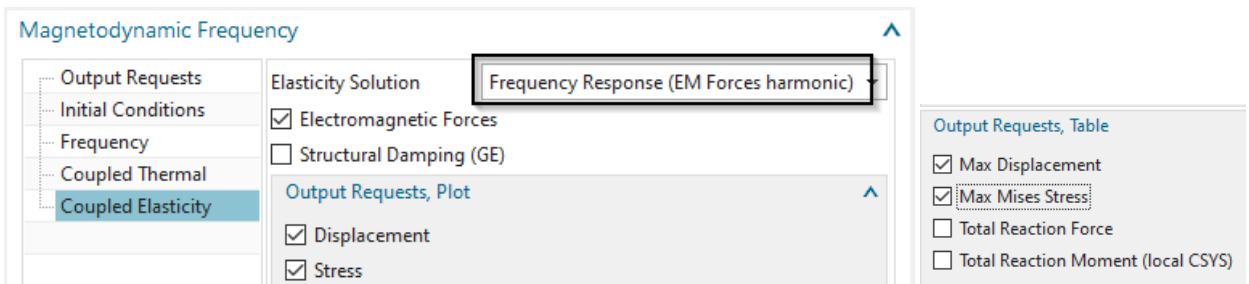


4. Solve the solution.
5. Check the resulting Y displacements. They should be very near to those from the static solution. The reason is that the frequency is quite low, so results are nearly static. With higher frequencies, skin effects would increase and results would change.

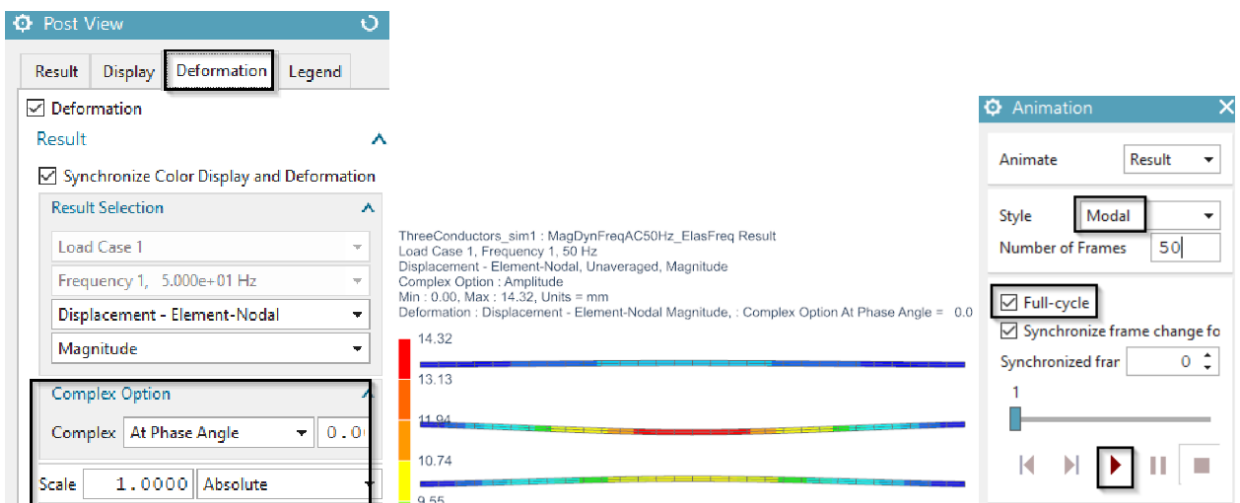


3.7 Solution: EM Frequency, Elasticity Frequency

1. Activate solution 'MagDynFreqAC50Hz_ElaFreq'.
2. In register 'Output Requests' activate results as desired.
3. In register 'Coupled Elasticity', set the 'Elasticity Solution' to 'Frequency Response (EM Forces harmonic)' and activate the 'Output Requests' as shown in the picture below.



4. Solve the solution.
5. To check the resulting displacements, run a animation to see the oscillation. Because displacements here are complex results (frequency domain) the same rules apply as for EM results in frequency domain. To animate, set in 'Post View', register 'Deformation' the 'Complex Option' to 'At Phase Angle' and the 'Scale' to 1, 'Absolute'. Then use the 'Animation', set 'Style' to 'Modal' and 'Full cycle', then 'Play'.

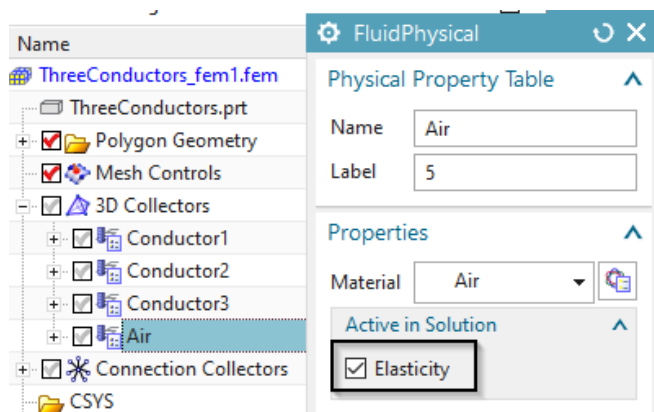


- The maximum displacements in this simulation are about 14 mm and thus much smaller than in the previous simulation, where we had 38 mm. A reason for this can be that the excitation frequency 50 Hz is away from the mechanical natural frequency of this conductor system. Such natural frequency can be found by a solve with NX Nastran solution 103. A nice additional training would be to find that natural frequency, apply the AC load at that frequency and find the displacements then again.

3.8 Solution: EM Transient Short Circuit, Elasticity Transient

For the following two transient solutions we want to use a trick: We include the air body into the elasticity solution. By that way the air mesh will deform smoothly, forced by the conductors deformations. This trick leads to a nice air mesh at each time step even if large deformations appear (only the bidirectional solution is affected by this and thus, we could activate that only here). Of course, the air material needs as material properties a small value for Youngs modulus, Poisson ratio and Mass density. The very small Youngs modulus leads to deformations, but nearly no stresses in the air, so the conductor deformations are nearly not affected by this. As a disadvantage, solve time and memory consumption will increase, because of many more elements in the elasticity system.

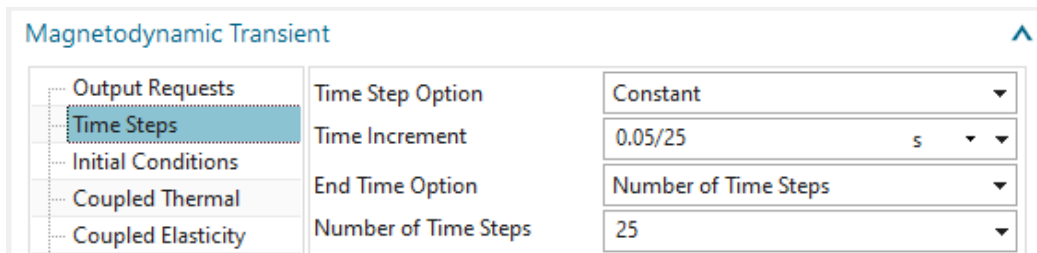
- Make the Fem part active.
- Edit the Physical of the air and at 'Active in Solution', activate 'Elasticity'.



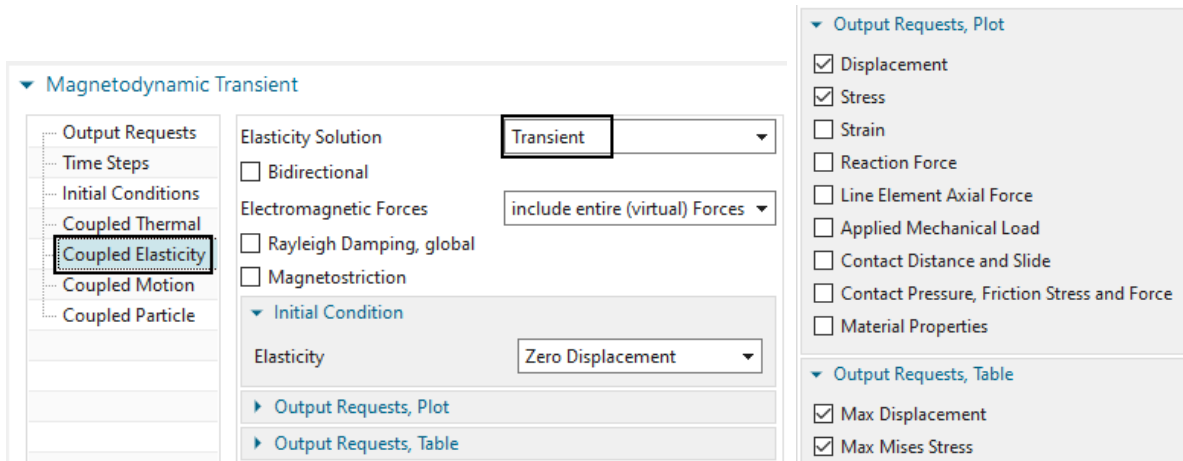
- Optionally check that the air material has said mechanical properties.
- Make the Sim part active again.

Following we activate the elasticity solution:

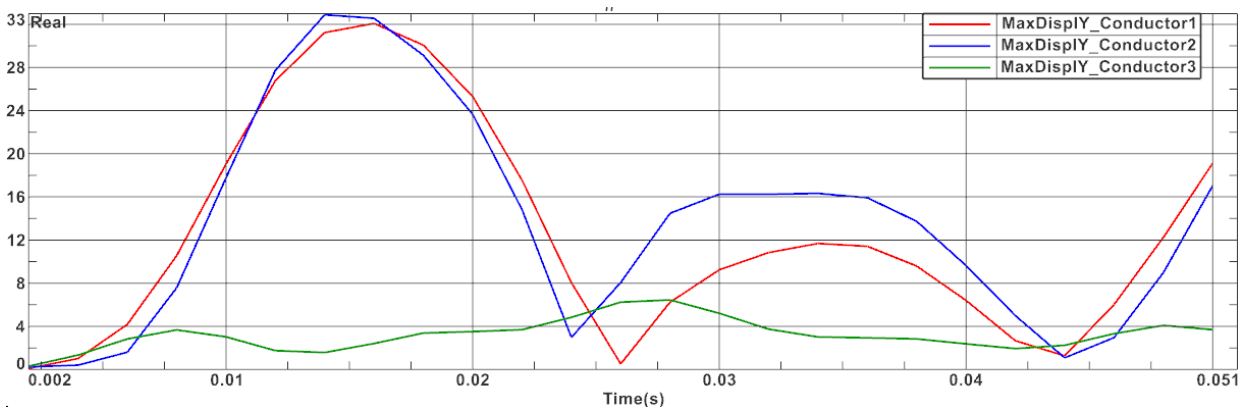
- Activate solution 'MagDynTrans_ShortCircuit'.
- In register 'Output Requests' activate results as desired.
- In register 'Time Steps', set the 'Time Increment' to 0.05/25 and the 'Number of Time Steps' to 25.



- In register 'Coupled Elasticity', set the 'Elasticity Solution' to 'Transient' and activate the 'Output Requests' as shown in the picture below.



- Solve the solution.
- Display the three graphs showing the maximum conductor Y displacements over time. As the following picture shows, the maximum displacement is found at conductor 2 with about 33 mm.

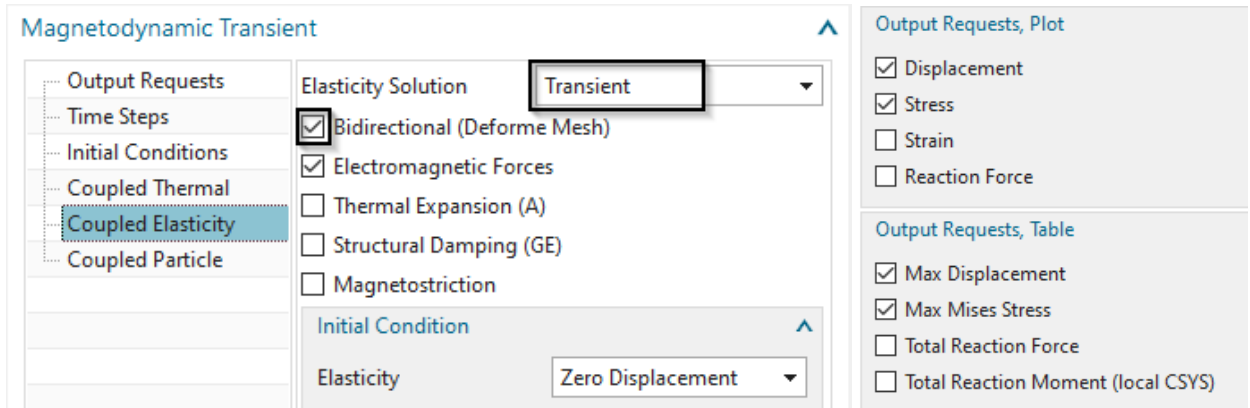


3.9 Solution: EM Transient Short Circuit, Elasticity Transient, Bidirectional

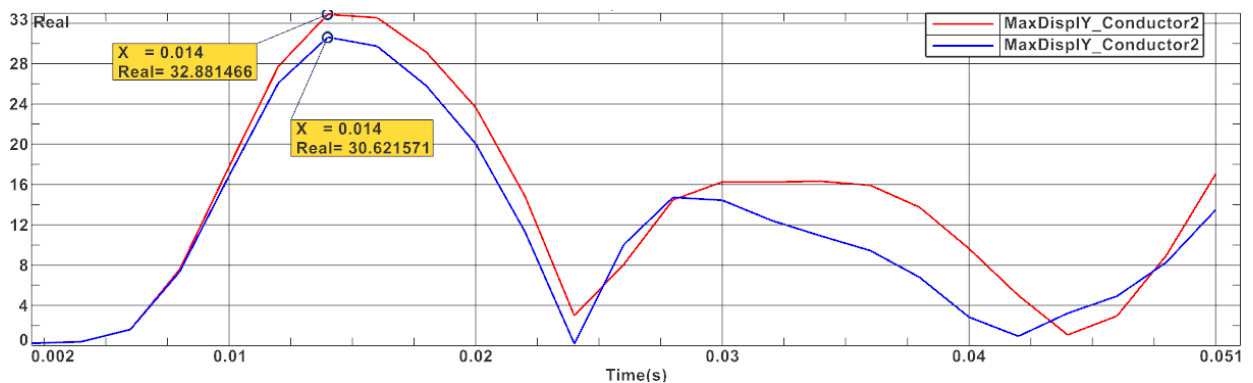
We want to use the same settings as in the prior solution except one setting: The Bidirectional option will be active here.

- Activate solution 'MagDynTrans_ShortCircuit_BiDir'.
- In register 'Output Requests' activate results as desired.

- In register 'Time Steps', set the 'Time Increment' to 0.05/25 and the 'Number of Time Steps' to 25.
- In register 'Coupled Elasticity', set the 'Elasticity Solution' to 'Transient'. Activate 'Bidirectional (Deforme Mesh)'. Also activate the 'Output Requests' as shown in the picture below.



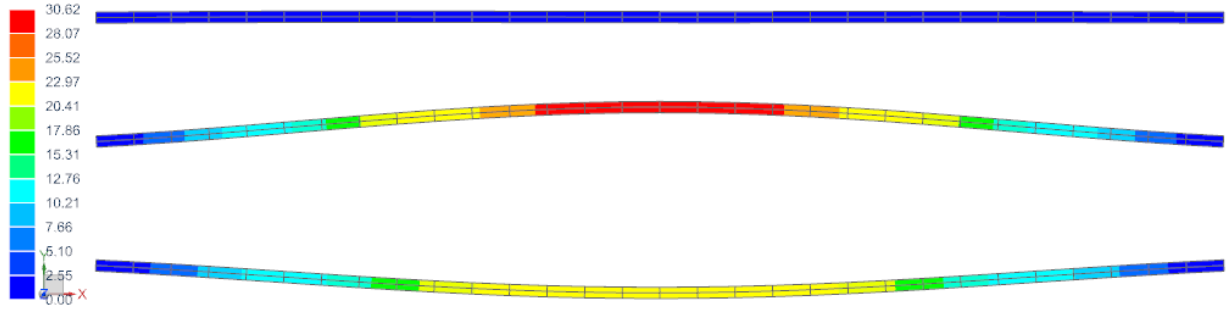
- Solve the solution.
- Display the maximum Y conductor displacements of conductor 2 of both Short Circuit solutions. As the following picture shows, the bidirectional solution shows slightly smaller displacements.



- Display the displacements at time 0.014 (maximum, as seen above). Because in this simulation the bidirectional effect is active, now both effects are captured:
 - The EM field increases the deformation,
 - The increased distance leads to smaller EM fields.

The solution is more realistic: The distance between conductors 1 and 2 increases, but therefore the fields become smaller and this again leads to a smaller distance.

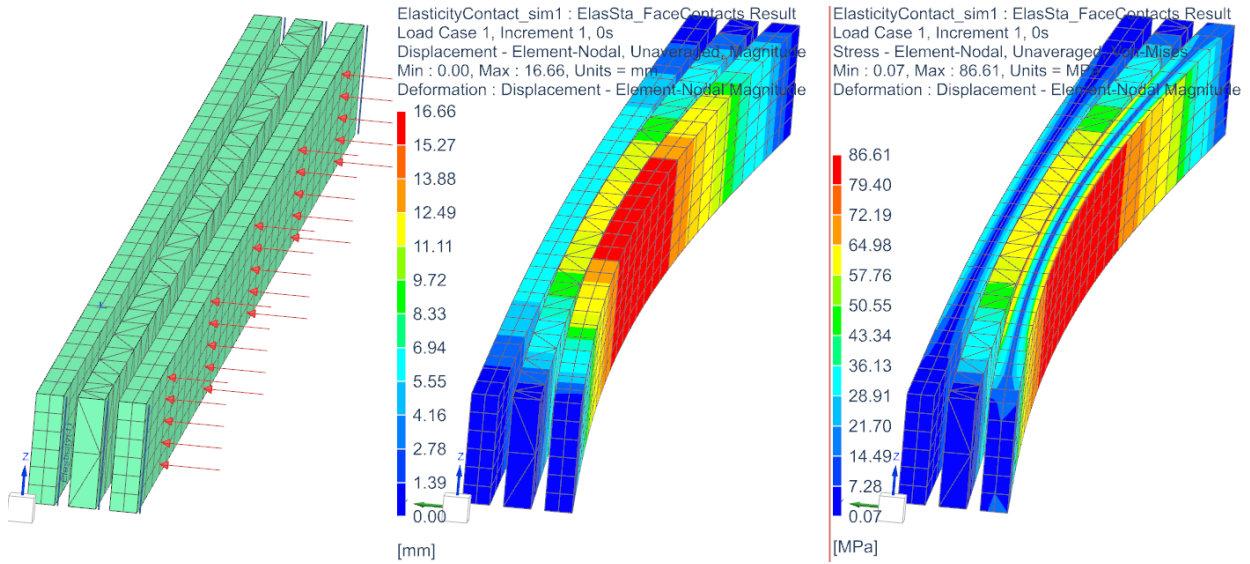
ThreeConductors_sim1 : MagDynTrans ShortCircuit BiDir Result
Load Case 1, Increment 8, 0.014 s
Displacement - Element-Nodal, Unaveraged, Magnitude
Min : 0.00, Max : 30.62, Units = mm
Deformation : Displacement - Element-Nodal Magnitude



The tutorial is finished.

4 Elasticity Contact

This tutorial shows how electric conductors contact while deforming. A later tutorial additionally shows how contacts can influence the electric current, what means if a gap arises, the electric resistance becomes very large. This example only shows the mechanical effects.

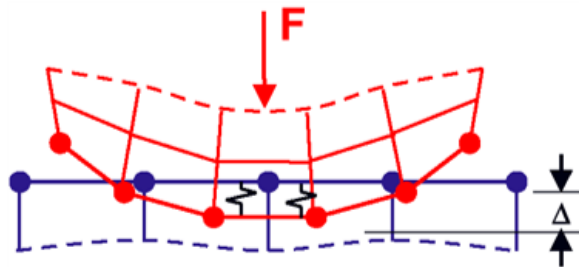


Follow the steps to see how the model is set up, solved and how the contact can be managed. The tutorial starts from an existing Fem and Sim file. Only the relevant features are added.

Estimated time: 1 h.

4.1 Contact Concepts

We use a classical elasticity contact feature, also called penalty contact algorithm. That feature introduces nonlinear mechanical effects what means that at each time step the solver must check for penetration in the contact area and if such penetration appears, there will be contact forces added. The contact forces must be adjusted in a way that the penetration disappears or falls below a given limit. To find such correct contact forces the solver must perform several iterations. Because of these extra contact iterations the computational costs become significantly larger when using contact.

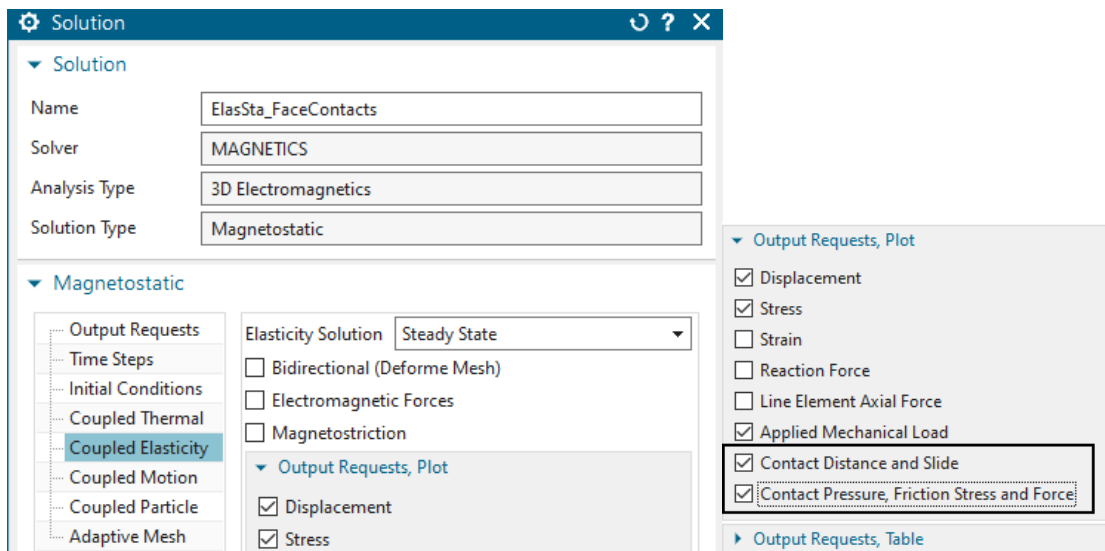


Two contact types are possible: 'Node-to-Node Contact' and 'Surface-to-Surface Contact'. The first type can be used to define single contacts between two points in the model. The second type is more advanced and can also handle coulomb friction. In this example we use both types for completeness.

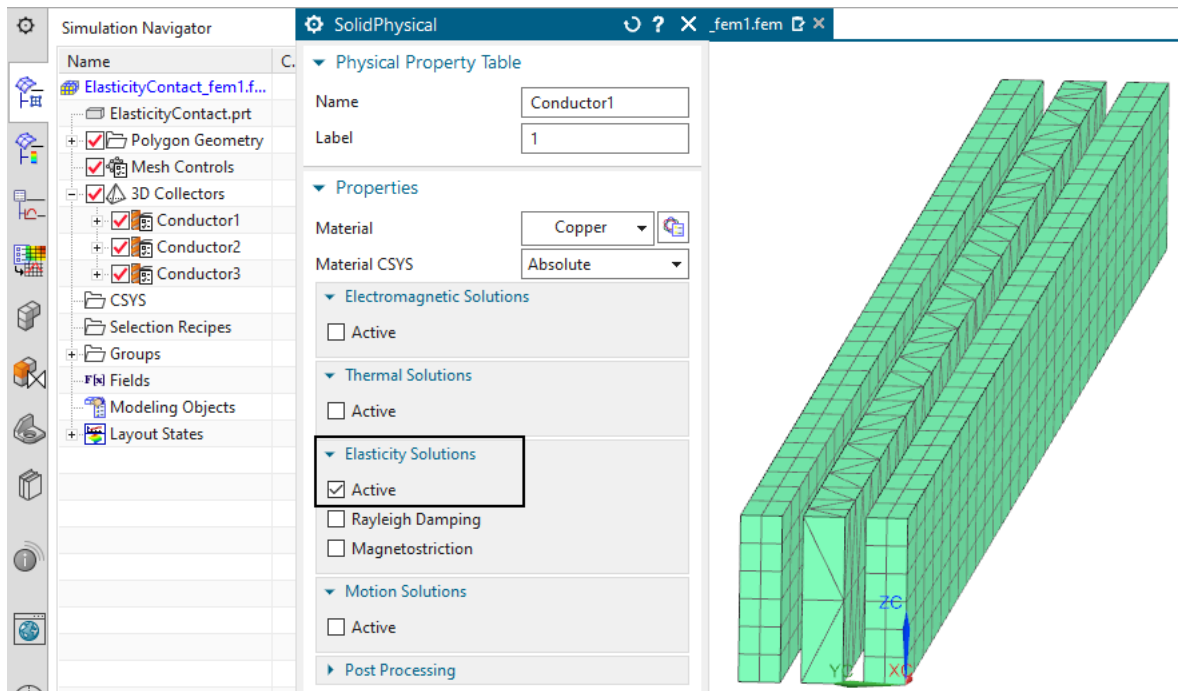
4.2 The Start Model

First check the existing files of the tutorial.

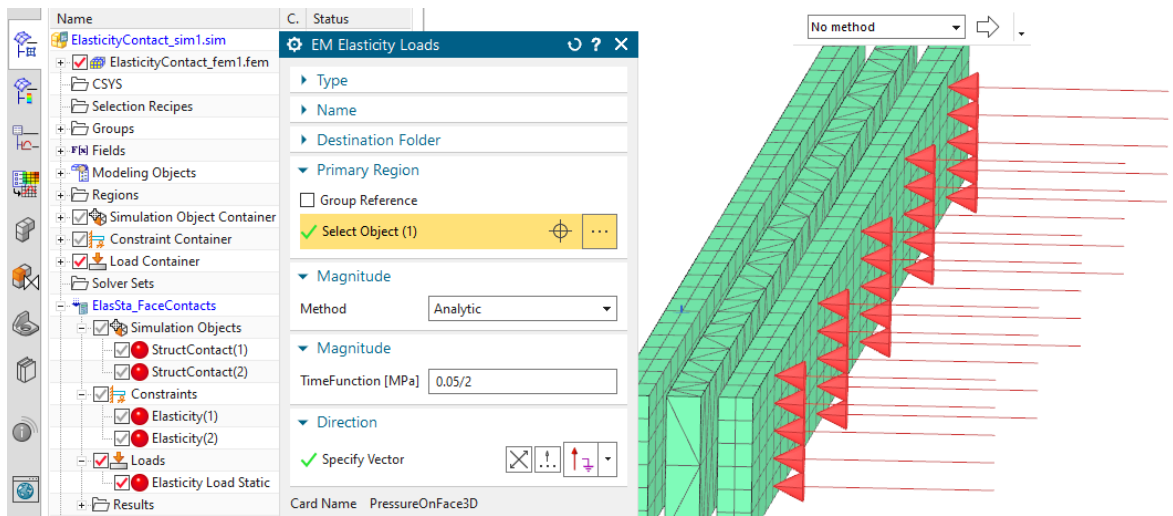
1. Download the model files for this tutorial from the following link:
<https://www.magnetics.de/downloads/Tutorials/8.CouplStructural/8.3ElasticityContact.zip>
2. Delete all existing 'Simulation Objects'. These contacts will be created in the following.
3. Check the existing model:
 - There are meshes of different types just to make sure contact work on all of them.
 - Two solutions are already there. One will be used for face contact and one for node contact. The solutions are of Magnetostatic type with zero time steps and we are only interested in the elasticity part.



- Notice there are two 'Output Requests' activated for this contact simulation: 'Contact Distance and Slide' and 'Contact Pressure, Friction Stress and Force'.
- The Physicals of all conductors also show that only the elasticity solution is of interest. All others are deactivated. The material is copper with an standard elastic modulus and poisson ratio.



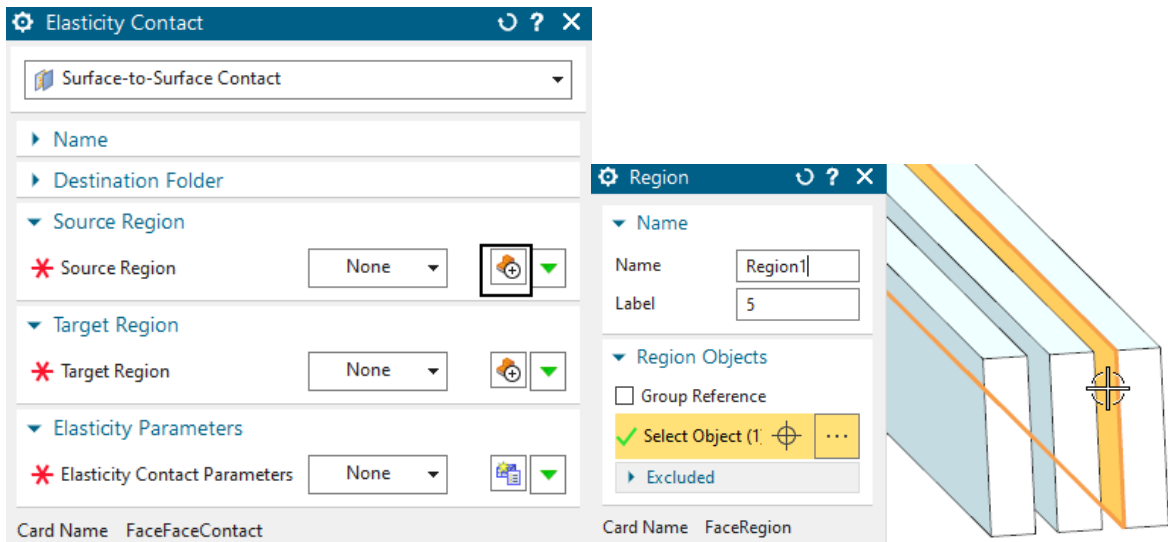
- There is a pressure load working on one side of the first conductor. Without contacts, this load would lead to a large penetration (Solve to check this). Also some elasticity constraints are defined to hold the three conductors in place.



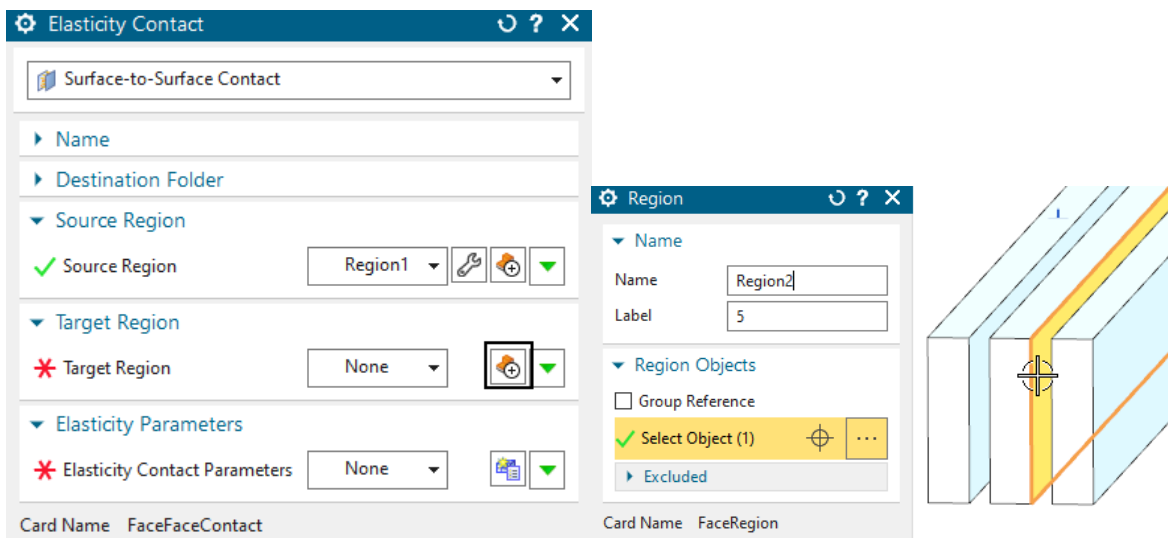
4.3 Create Surface-to-Surface Contacts

Create the first face contact as follows

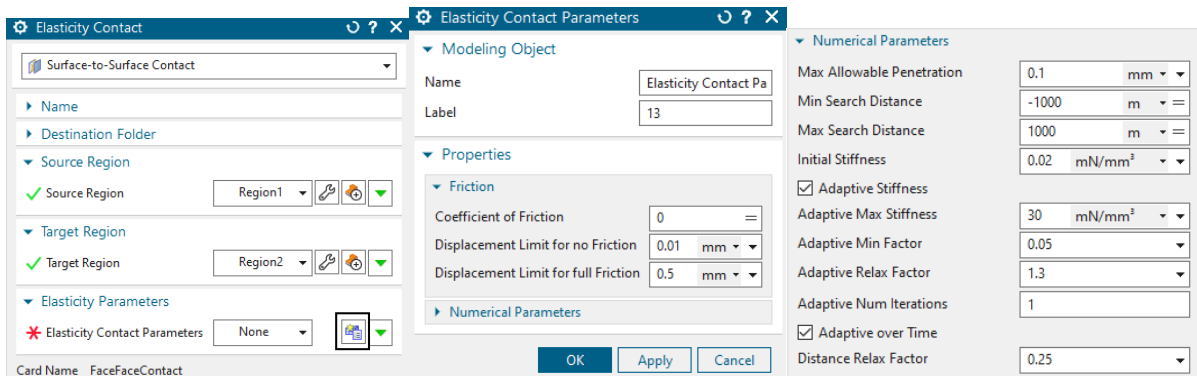
1. In 'Simulation Object', choose 'Elasticity Contact' and set the 'Type' to 'Surface-to-Surface Contact'. Now the two contact regions are defined. At 'Source Region', click 'Create Region' and select the face as shown below.



2. At 'Target Region', again click 'Create Region' and select the opposing face.

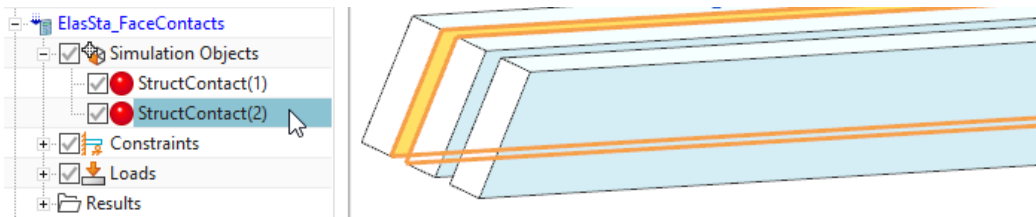


3. At 'Elasticity Contact Parameters', click 'Create Modeling Object'. The coefficient of friction, damping and several numerical parameters can be modified here. In many cases the defaults work fine, so there is no need to change anything now. Click OK two times to finish the contact creation.



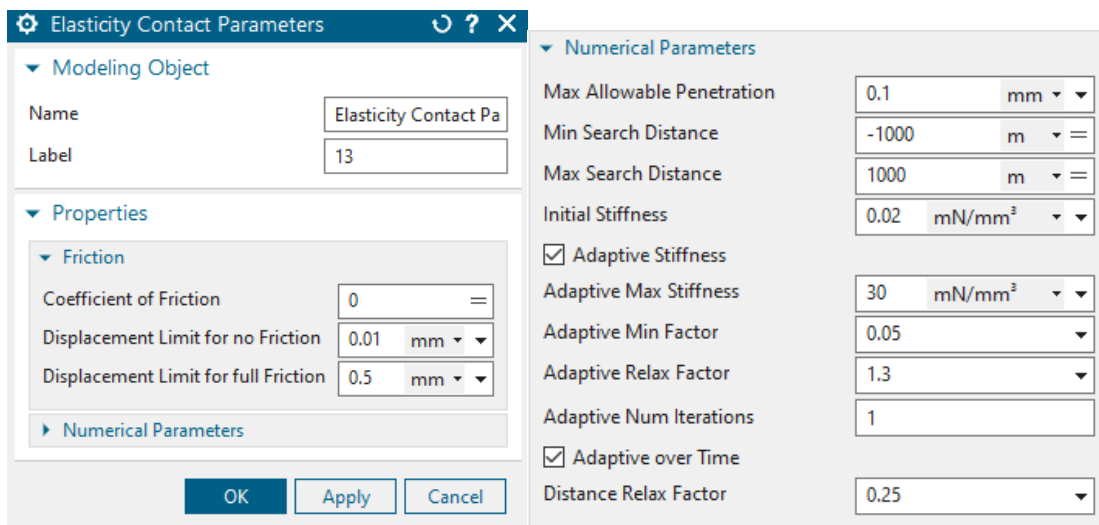
4. Create the second contact in the same way between conductor 2 and 3 and their corre-

sponding faces.



4.4 Contact Parameters Overview

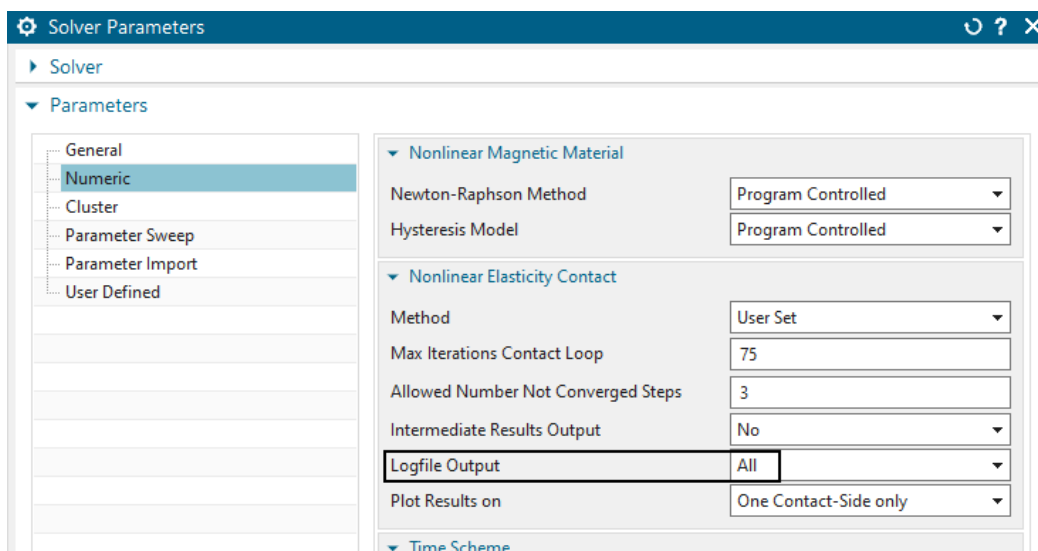
Even if we did not modify any, we will give a short information about the above contact parameters, because they definitely influence the simulation process. Although, many models run with the defaults, often solve time can be reduced by adjusting contact parameters.



- **'Coefficient of Friction'**: This value controls the coulomb friction forces that appear if this value is set larger than zero. Such friction forces act on the two contact faces in opposing directions, depending on the relative motion (slide). To control friction, there are also the two limits **'Displacement Limit for no Friction'** and **'Displacement Limit for full Friction'**. The assumption is that friction forces do not appear suddenly, rather than gradually increasing depending on the slide distance. Though, the first limit defines a small slide range that does not produce any friction forces. Then, if slide increases, the range between limit 1 and limit 2 defines the slide range in which the friction coefficient arises linearly between zero and the full given value. If slide becomes higher, full friction stays in effect.
- **'Max Allowable Penetration'**: This maximum penetration value defines the limit or residual that must be reached for a contact to converge.
- **'Min/Max Search Distance'**: The algorithm searches from the source to the target faces (and opposite) to find contact penetrations. These two distances define the limits of that search. They shouldn't be too small because often there is large penetration at the beginning of a contact process and even then the limits must fit.

- **'Initial Stiffness'** and **'Adaptive Stiffness'**: The contact algorithm finds penetration and calculates contact forces by the amount of penetration times a stiffness factor. If 'Adaptive Stiffness' is not activated, the used stiffness stays always the initial value given here. If the adaptive feature is activated, the stiffness factor will update depending on the current contact force, penetration and on the 'Adaptive Relax Factor'.
- **'Adaptive Max Stiffness'**: If the adaptive stiffness feature is used this value controls the maximum possible stiffness.
- **'Adaptive Min Factor'**: In case gaps arise during the contact iterations, the contact pressure will not suddenly become zero, rather it will decrease depending on the 'Distance Relax Factor'. As soon as the 'Adaptive Min Factor' is reached, the pressure is set to zero.
- **'Adaptive Relax Factor'**: This relax factor can be used to influence the adaptive stiffness computation. A smaller value, e.g. 0.1 leads to smaller stiffness, a higher value, e.g. 5. leads to more aggressive stiffness increase.
- **'Adaptive Num Iterations'**: This factor defines whether the adaptive stiffness is updated once at the beginning of a time step only (Default) or if it is updated again at following iterations.
- **'Adaptive over Time'**: If active, the previously adaptive stiffness is used at the beginning of a new time step. If not, it is initialized with the initial stiffness value.
- **'Distance Relax Factor'**: This factor controls how aggressive stiffness is decreased if gaps arise in contacts. Normally, the default should not be changed.

Some more settings for contact are found in the solver parameters under register 'Numeric' in box 'Nonlinear Elasticity Contact'. These are global settings, being valid for all contacts. If the setting 'Method' is switched from 'Program Controlled' to 'User Set' one can modify them. To see more info in the following simulation, we set the 'Logfile Output' to 'All'. Following the meanings of the settings.



- **'Max Iterations Contact Loop'**: Defines the maximum number of contact iterations. If the residuals of all contacts are not below their limits after these iterations, the whole step has not converged.
- **'Allowed Number Not Converged Steps'**: Sometimes some contact residuals do not fully reach their limit but the overall solution is valid. Therefore, this parameter defines how many of such not converged (connected) time steps are allowed until the solution stops. The solution monitor indicates such steps with 'NOT converged - Continue'.
- **'Intermediate Results Output'**: Setting this option to 'All' leads to a result output at all contact iterations. This can be used for debugging purposes mainly.
- **'Logfile Output'**: By default this is set to 'Mini', meaning the solution monitor shows information about each final contact iteration. Setting this to 'All' results in showing each iteration in the solution monitor.
- **'Plot Results on'**. By using the default option 'One Contact-Side only', contact results like pressure, force, sliding,... are shown on the first side of each contact only. Normally, the second side will have the opposing values and therefore it is not necessary to show such. Nevertheless, setting this option to 'Both Sides' will show the results on both contact sides.

4.5 Solving and Monitoring

1. Solve this solution 'ElasSta_FaceContacts'.

The solution monitor gives information about contact iterations. Each line 'Contact Loop' shows one iteration. The last line indicates that 62 iterations have been necessary for convergence.

```

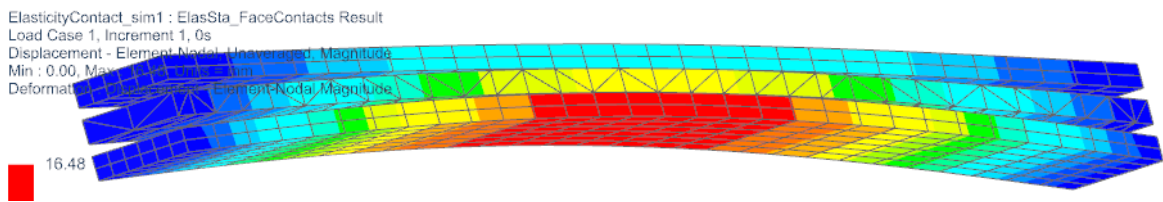
NX Magnetics - Solution Monitor - [ElasticityContact_sim1-ElasSta_FaceContacts.log LogSize=6,8 KB]
File Edit Window
ElasticityContact_sim1-ElasSta_FaceContacts.log
Info : Contact Loop, iter 54 Residuals (pid 4 -9.19616e-05) (pid 8 -0.00013103)
Info : Contact Loop, iter 55 Residuals (pid 4 -8.88192e-05) (pid 8 -0.000126128)
Info : Contact Loop, iter 56 Residuals (pid 4 -8.58863e-05) (pid 8 -0.000121508)
Info : Contact Loop, iter 57 Residuals (pid 4 -8.31469e-05) (pid 8 -0.000117143)
Info : Contact Loop, iter 58 Residuals (pid 4 -8.05858e-05) (pid 8 -0.00011301)
Info : Contact Loop, iter 59 Residuals (pid 4 -7.81754e-05) (pid 8 -0.000109088)
Info : Contact Loop, iter 60 Residuals (pid 4 -7.58618e-05) (pid 8 -0.000105392)
Info : Contact Loop, iter 61 Residuals (pid 4 -7.36512e-05) (pid 8 -0.000101884)
Info : Contact Loop, iter 62 Residuals (pid 4 -7.15447e-05) (pid 8 -9.85236e-05)
Info : All Contacts Converged with 62 Iterations. Residuals (pid 4 -7.15447e-05) (pid 8 -9.85236e-05)
Info : AFUWriter: Writing to Afu-Result File: ElasticityContact_sim1-ElasSta_FaceContacts_PostGraphs.afu
----- Successfully Ended -----
E n d   P r o c e s s i n g
Info : Stopped (Sat Mar 13 13:25:02 2021, Wall = 18.716s, CPU = 15.1562s, Mem = 439.871Mb)

```

Also, each contact is shown in brackets with its property identifier 'pid' and the residual. Negative values indicate penetration and positive values show distance gaps. Small penetrations below the limit have converged. The number of necessary iterations can be reduced by using larger contact stiffness, but in that case the risk for no convergence will arise.

4.6 Post Processing Contact Results

1. Open the plot results and display the result 'Displacements'. Verify that the overall solution looks correct.

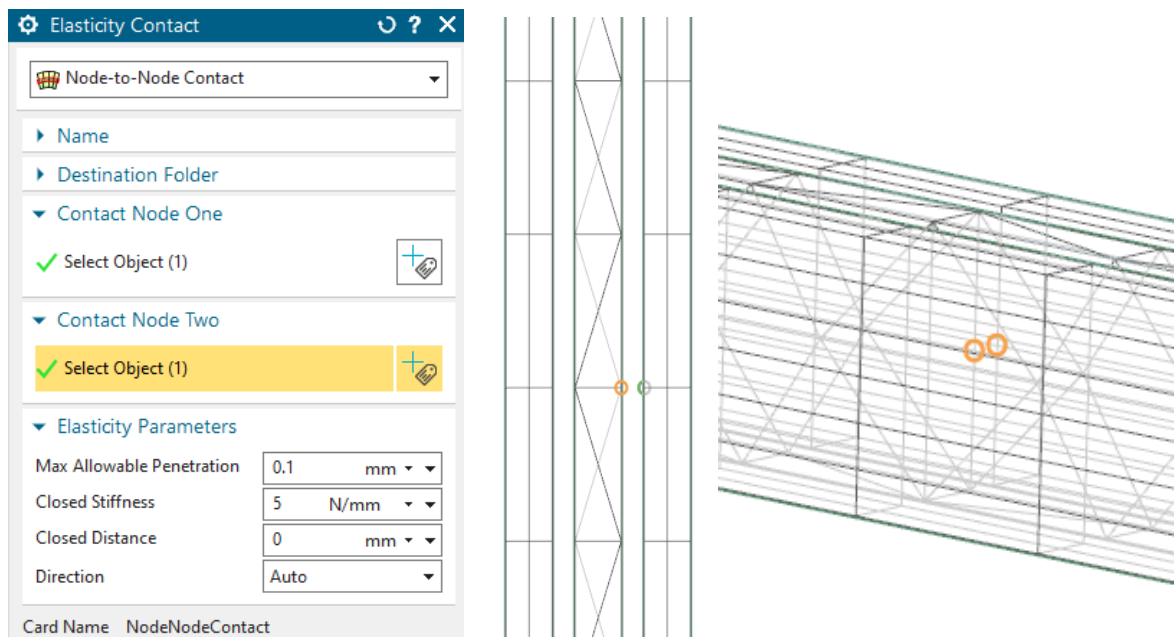


2. Also check the contact results. Display 'Contact Normal Force', 'Contact Pressure', 'Contact Slide', 'Contact Distance'

4.7 Create Node-to-Node Contacts

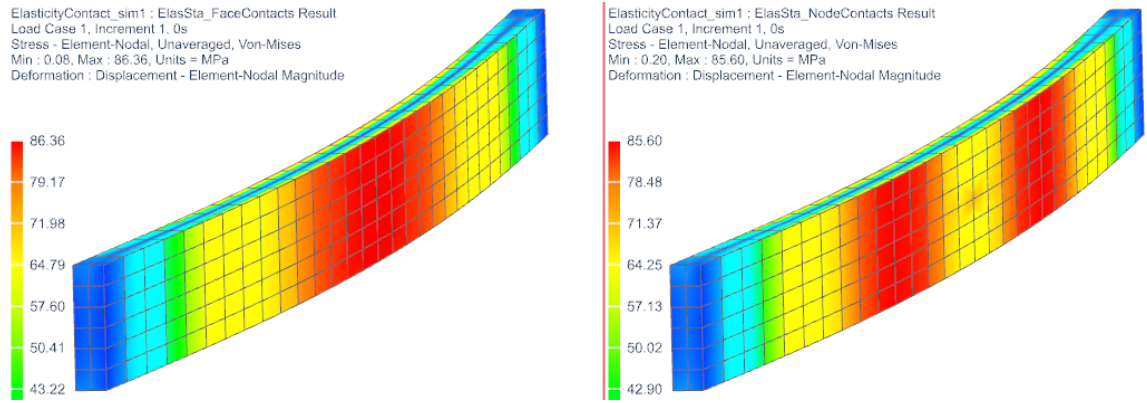
In the second solution 'ElasSta.NodeContacts' we want to use node contacts instead of face contacts.

1. Activate the second solution.
2. Create a new 'Simulation Object' of 'Elasticity Contact' and use the type 'Node-to-Node Contact'.



3. At 'Contact Node One' select a node from the face of conductor one quite in the middle.
4. At 'Contact Node Two' select a node from the face of conductor two quite in the middle. Click OK and the contact is created.
5. Create a second node contact between conductor two and three.
6. Solve the solution and post process the results.

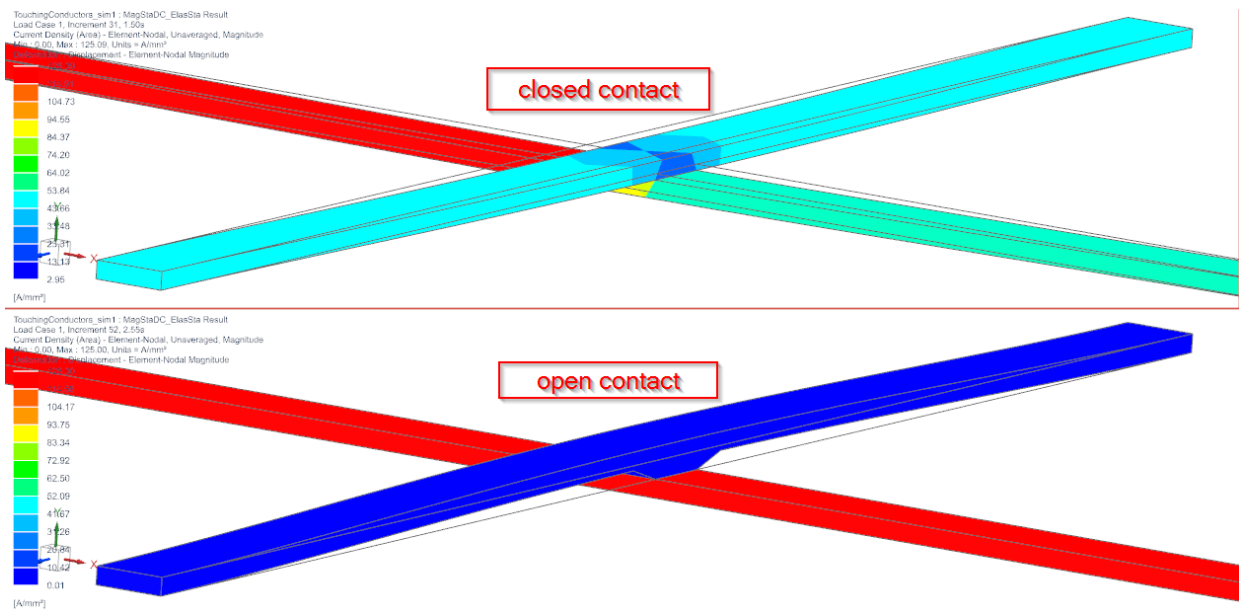
- 7. Deformation results will be similar to the face contact solution.
- 8. When comparing stress results, a difference is found. Because the node contact (picture below right) loads all contact forces on only the single node a different stress distribution results.



The tutorial is finished.

5 Touching and Releasing of Conductors

This tutorial shows a basic simulation principle of electric switches. Two conductors initially are in contact and electric current is flowing through both. Then, a mechanical force is acting on one of them forcing the contact to open and the current to stop. Even in the closed situation, there is a very small air gap in the contact area and this does produce some electric resistance. While the contact gradually opens, this gap enlarges and the resistance quickly becomes very high, so the electric current stops crossing the contact.



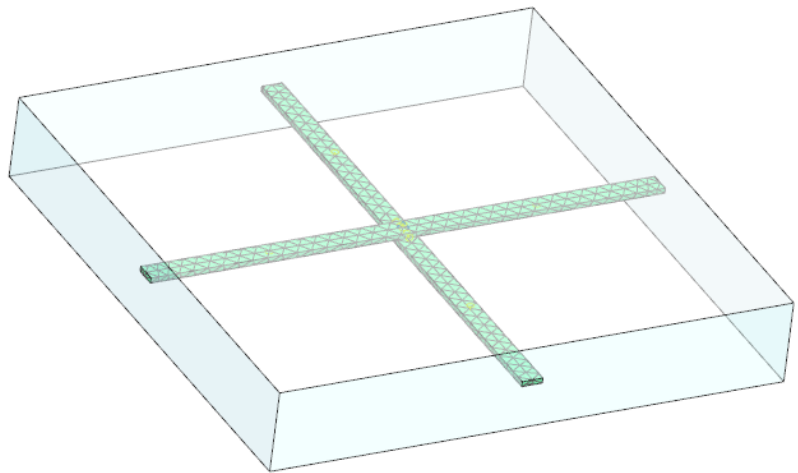
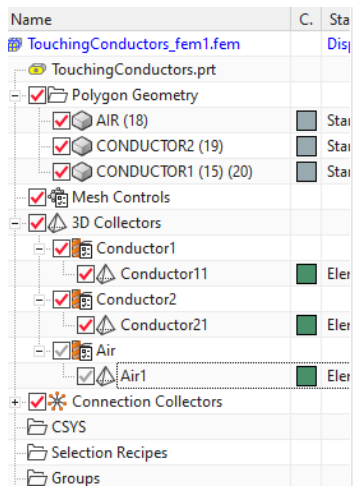
Estimated time: 0.5 h.

Follow the steps:

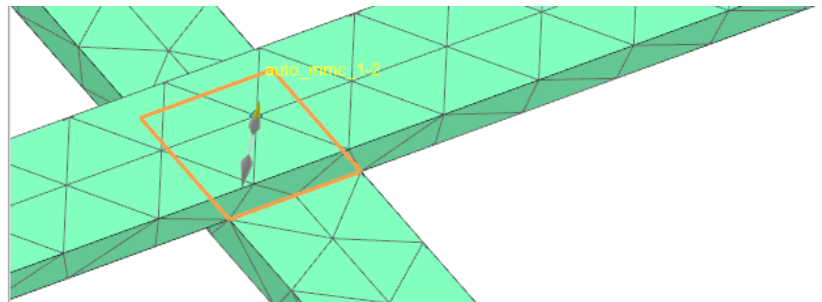
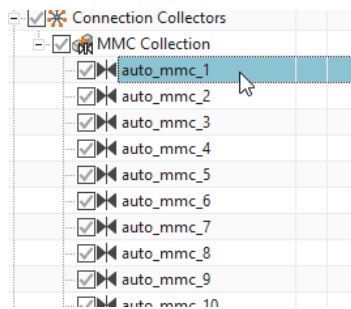
5.1 The Start Model

First check the existing model files of this tutorial.

1. Download the model files for this tutorial from the following link:
<https://www.magnetics.de/downloads/Tutorials/8.CouplStructural/8.4TouchingConductors.zip>
2. Open the file 'TouchingConductors_sim1.sim'
3. Delete the Simulation Object 'Thin Gap Condition(1)' and also the Modeling Object 'Elasticity Contact Parameters1'. We will newly create these in the following.
4. Check the existing model: First set the Fem part displayed to see the content.

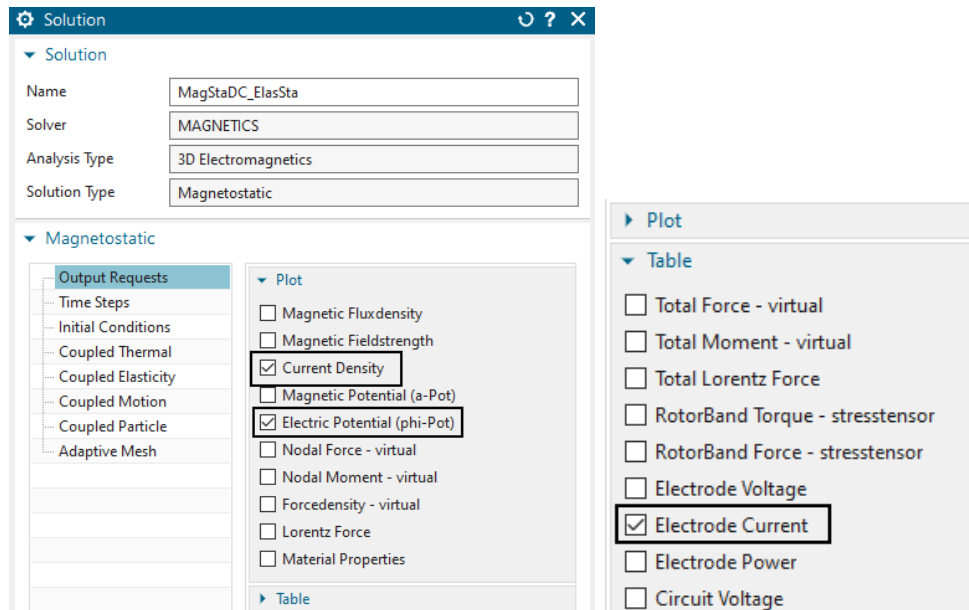


- There are two conductor meshes and one air. Nothing is special. It is very similar to the previous tutorials with electric conductors.
- One thing should be noticed: The Mesh Mating Condition between the two conductors is also a usual condition of type 'Glue Coincident'. Until now the two conductors are treated as if they were one. Later in the Sim file, we will define a 'Thin Gap Condition' with coupling to elasticity here. Thus, this face will become a contact that allows to open.

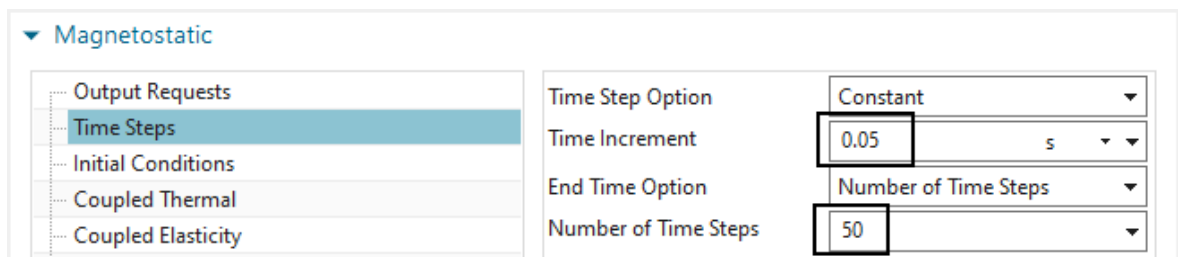


5. Set the Sim part displayed and check the solution.

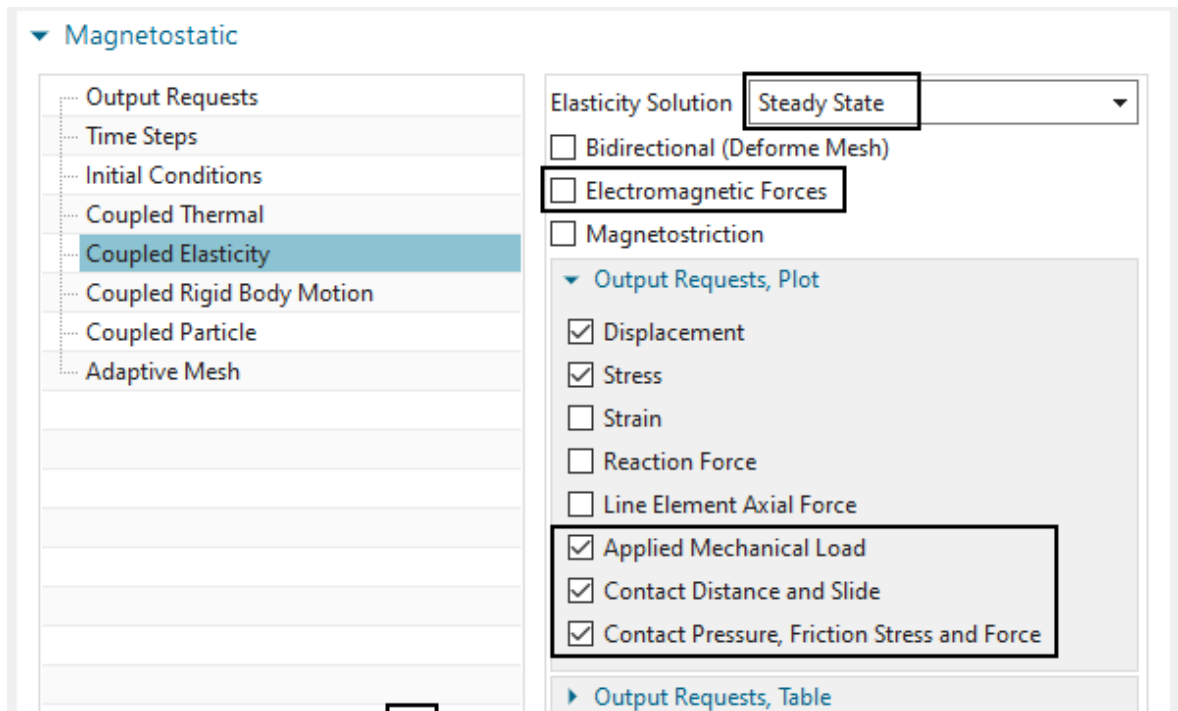
- Edit the existing solution 'MagStaDC_ElasSta' to check the settings. It is a magnetostatic solution type with some basic output requests activated to check the current.



- in 'Time Steps' there are 50 steps with 0.05 sec defined. During this time period the mechanical force will act, first forcing the closing direction, then altering to the open one.

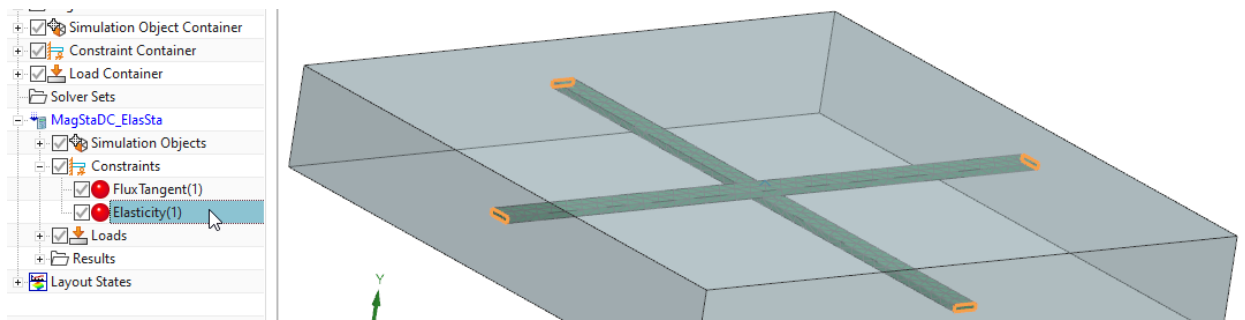


- In 'Coupled Elasticity' the 'Elasticity Solution' is set to 'Steady State' and some output request for contact are activated. Although this is a static solution, all these features are also possible in transient solutions.
- Notice, 'Electromagnetic Forces' is deactivated. Of course, this can also be active. But in this tutorial we want to see the effect of mechanical forces only.



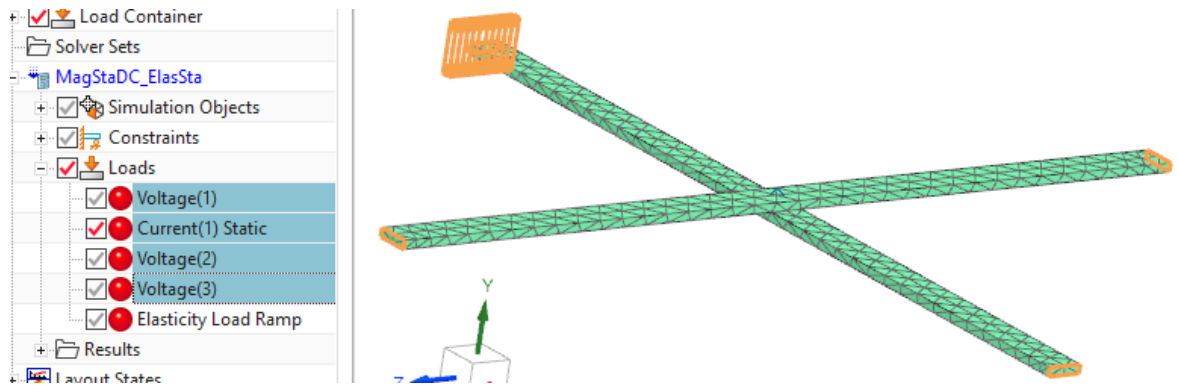
6. Check the constraints.

- Like always in electromagnetics, there is a 'Flux Tangent' condition at the outside walls of the meshed volumes.
- An elasticity constraint 'Elasticity(1)' fixes all four electrode faces of the two conductors.



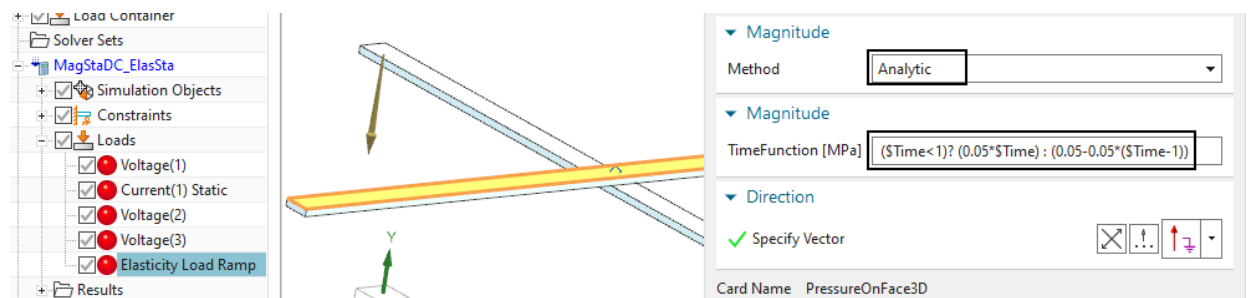
7. Check the loads.

- There is a current load 'Current(1) Static' applied to one of the electrode faces. This injects an electric current of 50000 amps into the conductor system. All three remaining electrode faces have a voltage zero condition applied that allows the current to enter or leave.



- A mechanical pressure load named 'Elasticity Load Ramp' is of 'Analytic' type and uses the following time dependent condition:

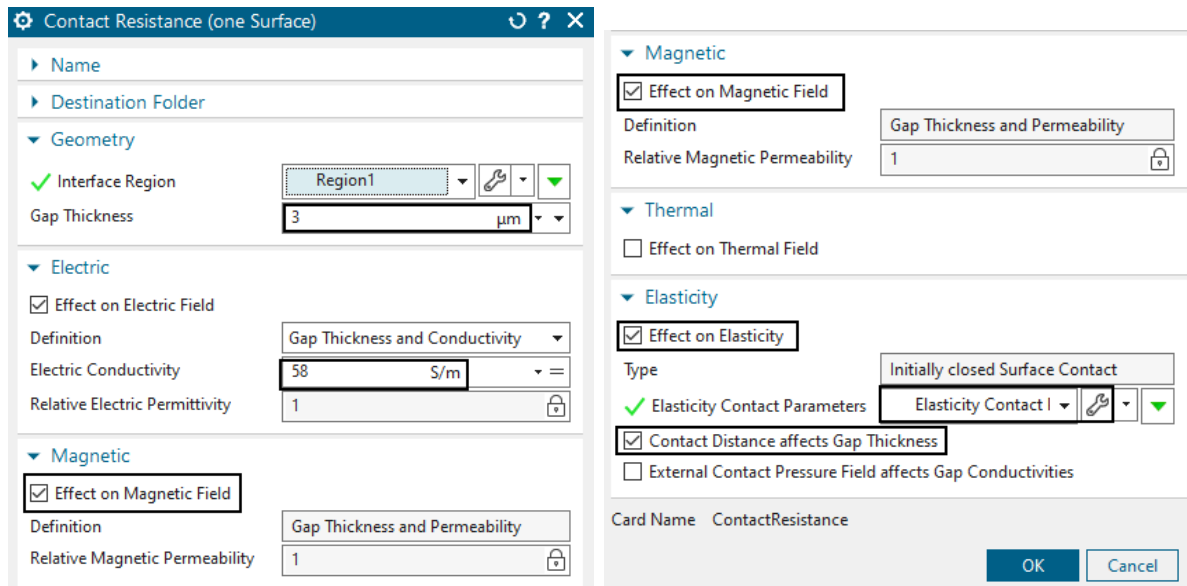
$$(\$Time < 1) ? (0.05 * \$Time) : (0.05 - 0.05 * (\$Time - 1)).$$
This form reads as: In case time is smaller 1, use the form $0.05 * Time$. In other cases, use the form $0.05 - 0.05 * (Time - 1)$. These are two linear functions, one for the increase the other for the decrease of the mechanical pressure.



- Such were the existing features in the model. Nothing very new for the readers of this document. Following the new condition is created.

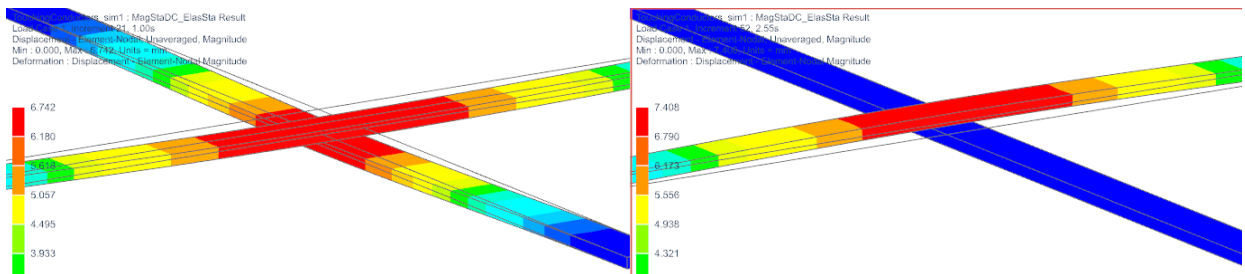
5.2 Creating a Thin Gap with EM-Elasticity Coupling

1. Create a new Simulation Object 'Contact Resistance (one Surface)'.
2. At 'Region', select the contact face between the two conductors.
3. Set the 'Gap Thickness' to 0.003 mm. This value represents the initial gap opening. There is already a small electric resistance because of this small gap.
4. Activate 'Effect on Electric Field' and set the 'Electric Conductivity' to the value 58 S/m. Notice, this is much less than copper with $58e6$ S/m. Thus, as long as the gap remains small there is very small resistance.
5. Also activate 'Effect on Magnetic Field'.
6. Now activate 'Effect on Elasticity' and create a 'Modeling Object' for contact parameters (accept all defaults here).
7. Finally, activate 'Contact Distance affects Gap Thickness'. This leads to the desired effect of stopping the current when opening the contact.

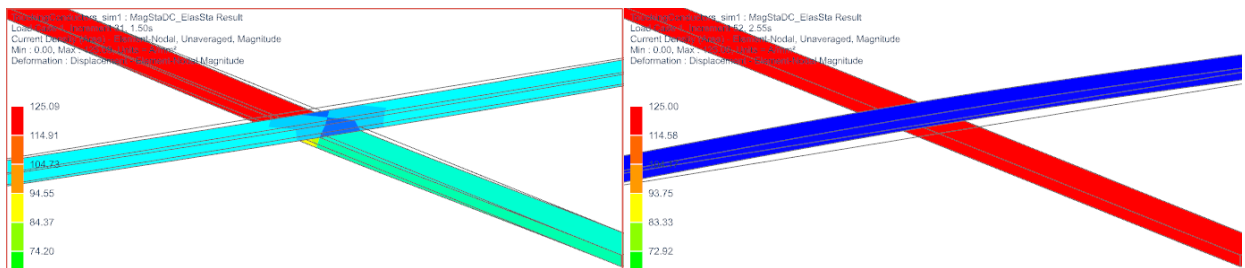


5.3 Solving and Post Processing

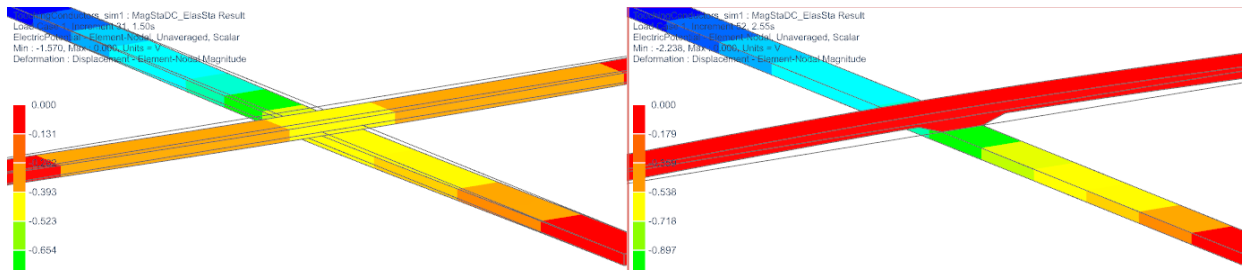
1. Solve the solution. The 50 steps with contact need about 2 minutes solve time.
2. Display the Displacement results. Tip: For easier display, hide the Air mesh and also set the edges to 'Feature'. In the below picture there are additionally the undeformed edges shown. The left side shows the time of full force in contact direction. The right side shows the time of full force in opposite. The gap arise can be clearly seen.



3. Set the result to 'Current Density' now. Again, compare the two time steps as in picture below. Left side is the closed situation and the electric current being injected (top left) is split in three parts. Right side shows the open situation. The whole current now flows through one conductor.



4. Finally, set the result to 'Potential' and again compare. Left side (closed) shows there is no jump in the potential at the contact. Right side (open) does show the potential jump.

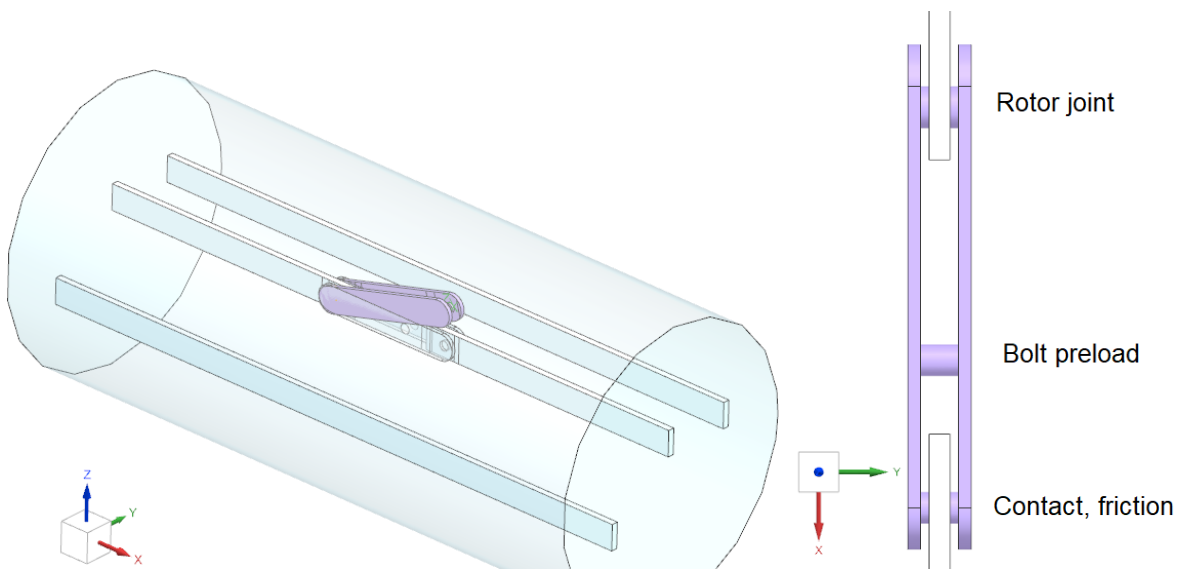


The tutorial is finished.

6 Electromechanical Switch

This tutorial shows a simple electro mechanical switch and its simulation principle. Three conductors are arranged as shown below (left) and loaded by three phase AC currents. For simplicity in this tutorial, only one of the three has a switch. The switch is shown in detail below right. A free rotating joint connection without friction couples the switch to the conductor (top). On the other side (bottom) a clamped contact with friction connects to the other conductor side. In between there is a cylinder with mechanical pre-load, for instance by a screw. The pre-load is given, also the friction coefficient and the electric currents.

Lorentz forces acting on the conductor system will maybe open up the switch and therefore this simulation shall give an estimation about expected movements of the switch.



Estimated time: 1 h.

Features to learn:

- Elasticity Contact with effect on electric and magnetic field,
- Elasticity joint connections,
- Elasticity Bolt pre-load.

6.1 The Demo Model

First check the existing model files of this tutorial. This model is completely set up with all necessary features. In the tutorial we walk through those features of interest and discuss them. Of course, a good idea for learning would be to start from the CAD model (in folder 'start') and set up the whole model.

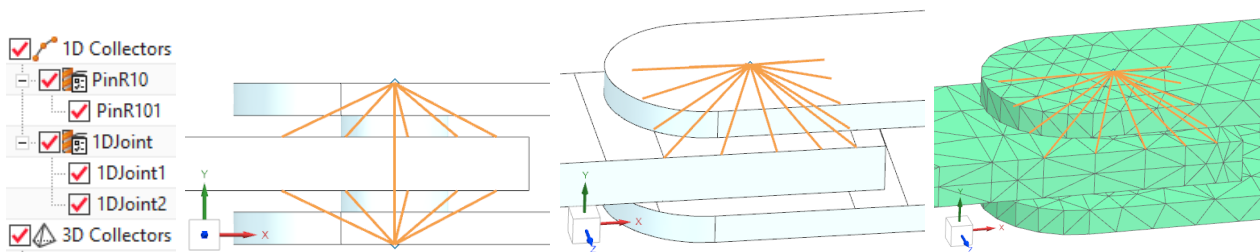
1. Download the model files for this tutorial from the following link:
<https://www.magnetics.de/downloads/Tutorials/8.CouplStructural/8.5SimpleSwitch.zip>

2. Open the file 'SimpleSwitch2_sim1.sim'.
3. Check the existing model: First set the Fem part displayed to see the content.

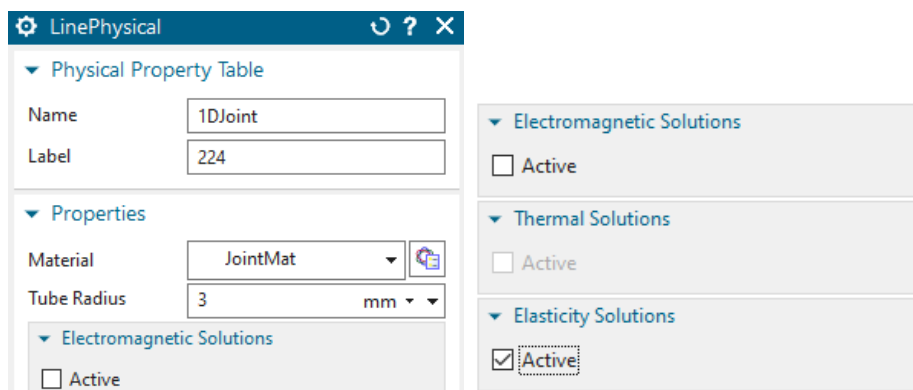
6.2 Elasticity Joint Connection

The first thing to draw the attention on is the rotating joint connection and how this is managed in the model. Often in electro-mechanical devices there are components being coupled in that way: only rotation is possible what is also called a revolute joint. One possibility for such a connection can be using several contacts. But another, simpler possibility is the construction of several line elements like we do here. Line elements in elasticity behave like rods, that means they do not transfer moments.

The pictures below show that type of joint model that is a build by two 'Spider' type 1D connection meshes, each from a point to some edges. Additionally there is one direct 1D connection from point one to point two. By using such a construction the parts are connected by the lines, but rotation is allowed.



The material 'JointMat' and physical properties for such joint elements are chosen in a way that they do not influence the dynamics: they have a very small mass and the elastic modulus is taken from steel. The 'Tube Radius' is chosen in a way that the elastic forces will be strong enough to keep the parts in position. We would notice by the resulting displacements if these are chosen too small.

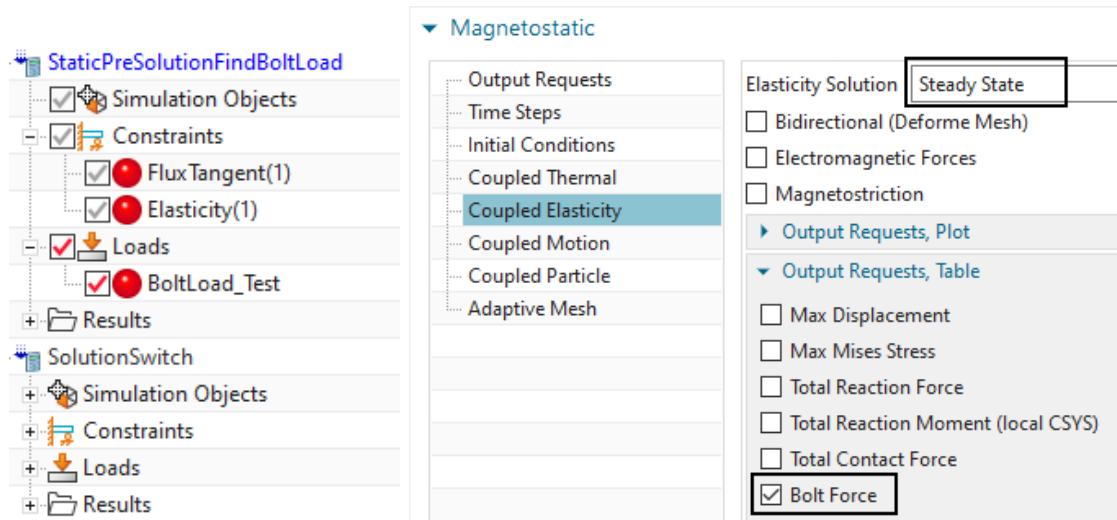


For the lines all solutions but 'Elasticity' are deactivated, so they do not play a role for the electromagnetic solution.

6.3 Handling of Elasticity Bolt Pre-Loads

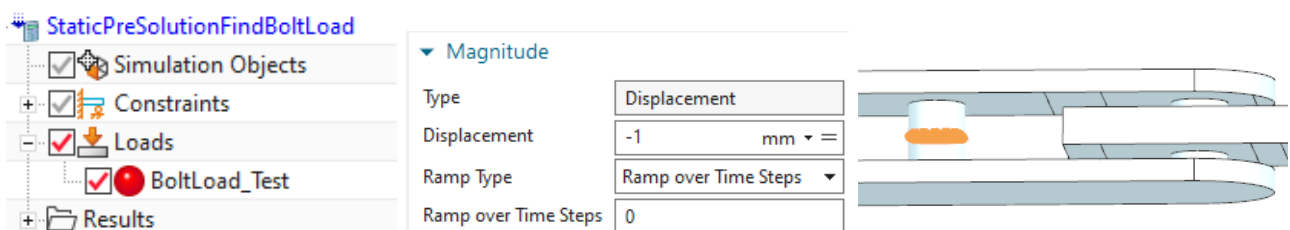
- Lets now check and discuss the bolt pre-load, so set the Sim part displayed.

- Notice, there is a solution named 'StaticPreSolutionFindBoltLoad'. This is a simple Magnetostatic solution with all constraints, but no contacts. Coupled Elasticity is set to 'Steady State' here.
- There is also a load 'BoltLoad_Test'. This is an 'EM Elasticity Load' of type 'Bolt Pre-Load on 3D Elements - Cut Plane'.

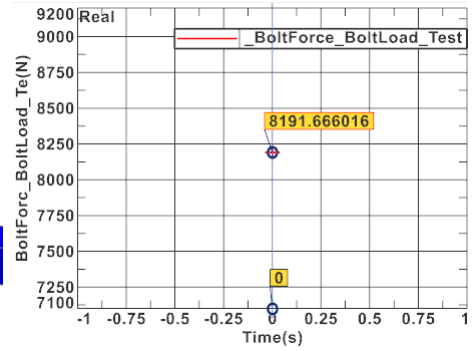
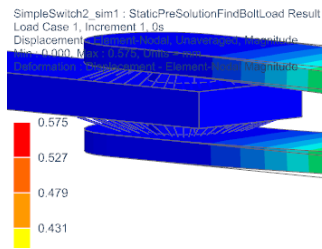


As the names announce, we use a pre solution. The bolt load feature currently cannot directly use a force as input, instead it can use a given displacement (similar to negative strain) on the bolt. That enforced displacement or strain will result in force. Thus, the strategy is applying an initial displacement, finding the resulting force and scaling the displacement to the desired bolt force. The corresponding displacement leading to that force will be the needed one for following simulations.

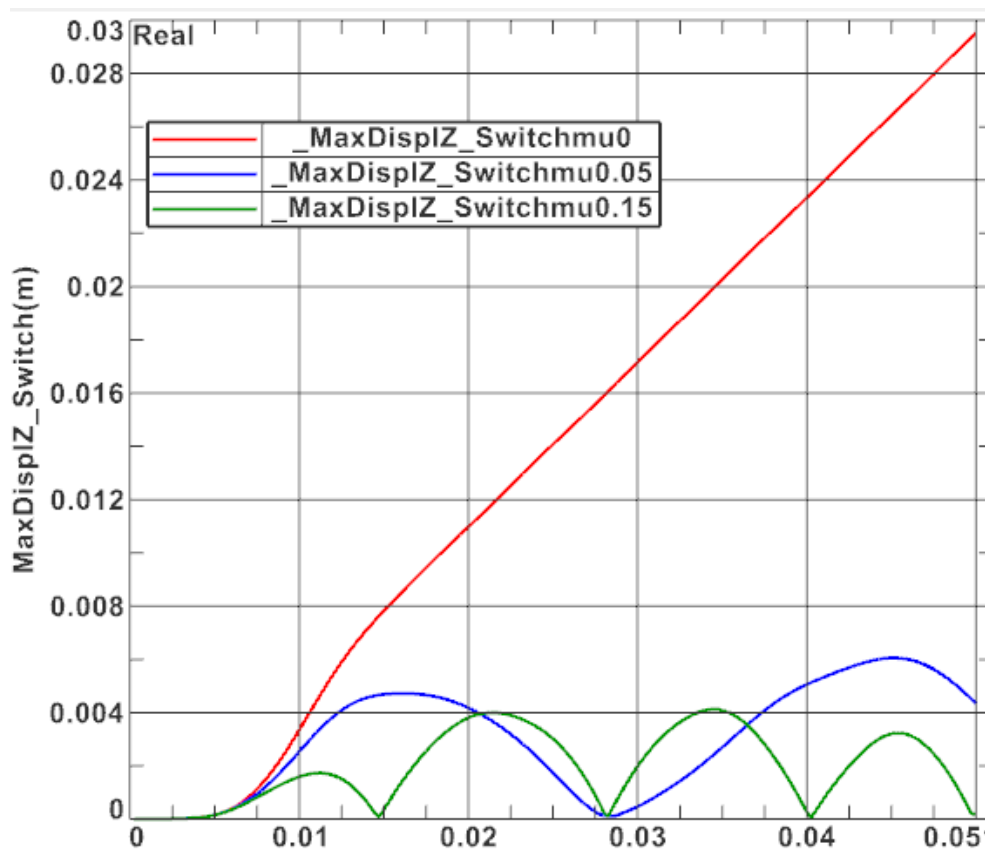
In this tutorial the desired bolt load is 5000 N. In the pre-solution we initially apply a displacement of -1 mm. Notice, the 'Ramp over Time Steps' is zero. This option is useful only in dynamic solutions, to prevent unwanted oscillations.



Thus, after solving, there results a deformed bolt and the tabular result shows the bolt force is 8191.6 N (the plot below shows an over scaled deformation).

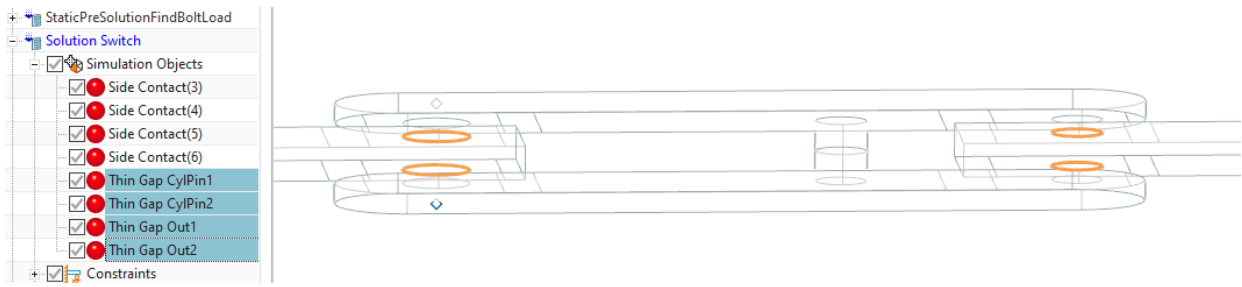


To find the necessary scaled displacement we divide the desired 5000 by 8191.6 what results in a scale of 0.6104. Thus, in the main solution 'SolutionSwitch' we use -0.6104 mm as applied displacement. Also, in that dynamic solution, we use a 'Ramp over Time Steps' set to 5 to avoid oscillations.



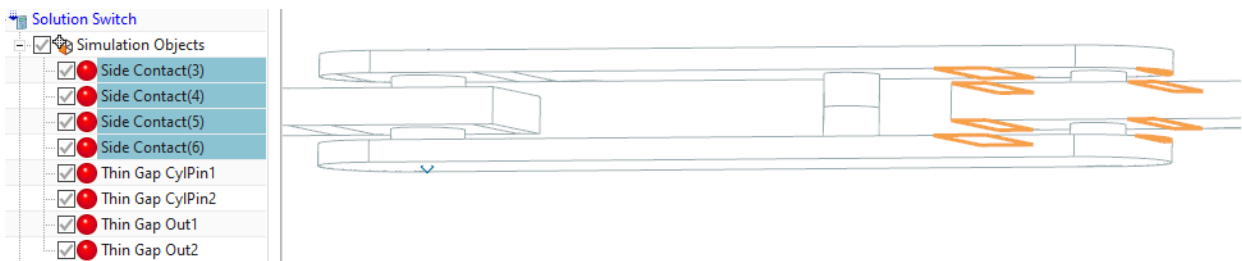
6.4 Contacts with EM Elasticity Coupling

Notice the Simulation Objects named 'Thin Gap CylPin' and 'Thin Gap Out'. These define faces at which electric current passes over a gap and that gap may change its thickness and size depending on the elasticity solution. These contacts work as already demonstrated in the previous tutorial '8.4TouchingConductors'.



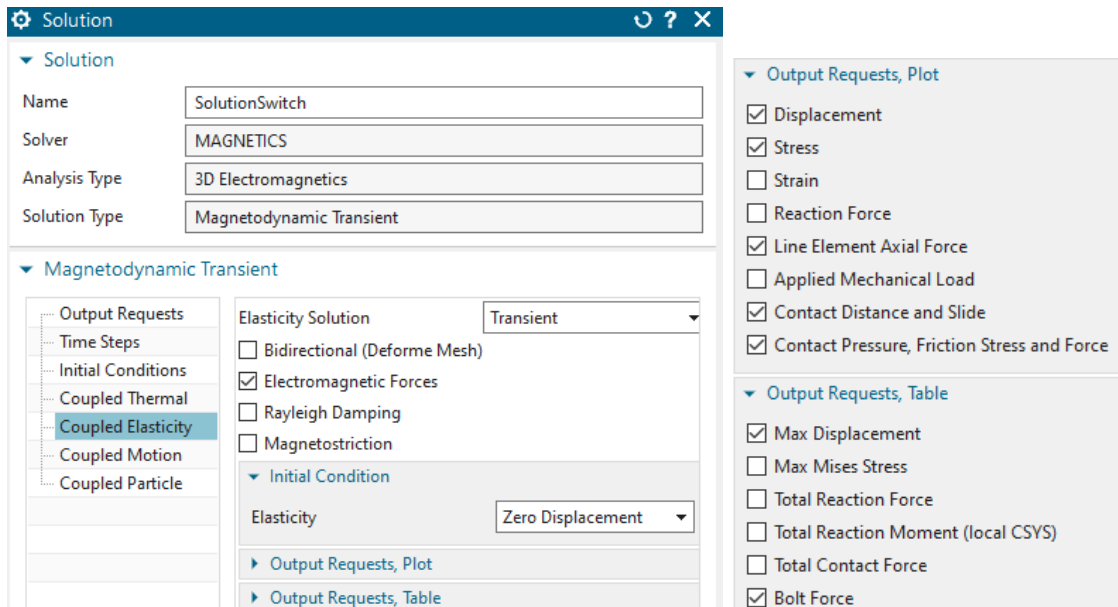
6.5 Additional Contacts to Avoid Penetration

Beside the contacts with EM Elasticity Coupling there are also some other contacts that simply avoid penetration of the geometry while deforming. These are the Simulation Objects named 'Side Contact'. They are of type 'Elasticity Contact' and they work as already demonstrated in the previous tutorial '8.3ElasticityContact'.



6.6 Dynamic Solution for EM-Elasticity

Finally, check solution 'SolutionSwitch'. There is nothing special in it. It is a Magnetodynamic transient one with Elasticity coupling also set to transient.



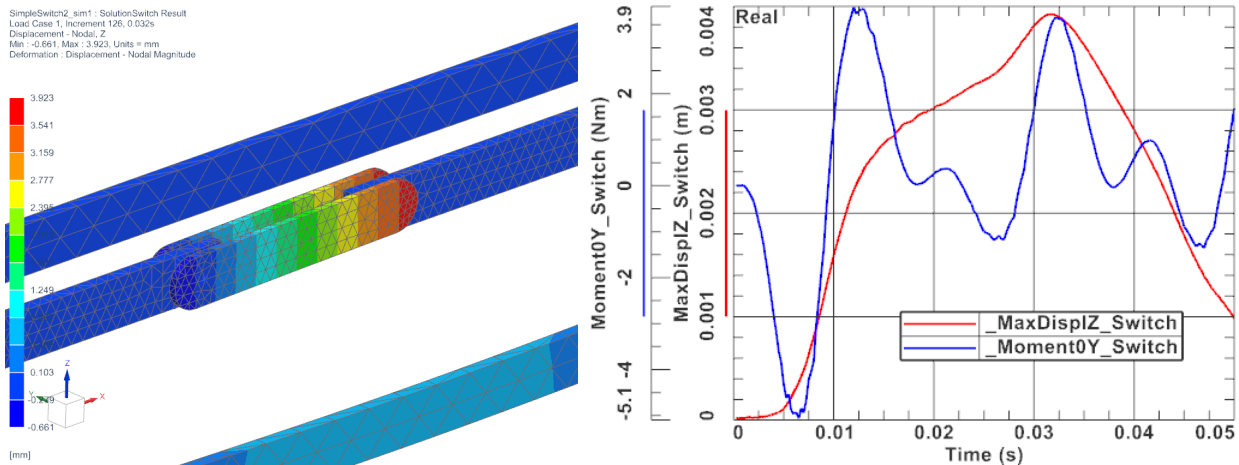
Time Steps and time increment are shown below. Set the 'Number of Time Steps' to 200 to solve the full range. Some usual output requests (Plot: Current Density and Table: Electrode Current) are activated.

▼ Magnetodynamic Transient

Output Requests	Time Step Option	Constant
Time Steps	Time Increment	0.00025 s
Initial Conditions	End Time Option	Number of Time Steps
Coupled Thermal	Number of Time Steps	200
Coupled Elasticity		

6.7 Solve and Post Processing

The solve of these 200 time steps with EM Elasticity coupling will take about 30 minutes. Main results of interest are displacement over time steps. Also the moment that results from Lorentz forces and acts on the switch is of interest. The following graphs show the Z displacement of the switch (red curve) and the moment around the axis (blue curve). The left picture shows the maximum displacement of the switch.



The above simulation shows relatively small displacements of the switch. That can be explained by the behaviour of the moment what results from the given oscillating voltage load.

The tutorial is complete.