Dynamic Simulation of Electromechanical Systems using ANSYS and CASPOC

Jens Otto
CADFEM GmbH

Abstract
Electromechanic actuators are one of the widely used drive systems in the recent years. The number of such systems e.g. for automotive und biomedical applications is even more growing now. All together most of these systems must feature a very low power consumption, a small size (and even much smaller in the future) and the production costs should tend to zero using standard components.

The computation of the dynamic behaviour of a complete electromechanical system including moving parts and eddy current losses is an important requirement for a complete virtual prototype in an up to date development process. This characterization can be realized with an expensive magnetic-mechanical coupled field simulation. A reduced order system simulation based on the coenergy approach can be much more effective if some simplifications relating eddy currents can be made.

The following paper summarizes the possibilities and experiences at CADFEM with the finite element program ANSYS as a preferred analysis tool for electromechanical system simulation.

Introduction
Electromechanical Systems are in general energy converter transforming electrical energy into mechanical and vice versa. Some typical examples of such transducers are: loudspeakers, electromagnetic relays, electric motors and generators and also electrical measuring instruments.

The two electromagnetic effects producing an mechanical force used in such applications are the attempt to shorten magnetic flux lines, and the interaction between current carrying conductors and the magnetic field (Lorenz force). No matter which effect is used, the mechanical force always points into a direction such that the stored magnetic energy is minimized. For a typical (lossless) electromechanical system the energy conservation can be described as:

\[
\text{Input of Electric energy} = \text{Performed Mechanical work} + \text{Increased Magnetic energy}
\]

\[
\text{Figure 1 - Energy conservation principle}
\]

This conservation can be extended if losses must be included to cover eddy current and damping effects. The most simple way to predict the behaviour of an electromechanical system must include the mechanical and the magnetic aspects and can be realized with an electric circuit analogy. Starting here the next chapter explains the parameter extraction inside ANSYS and the system simulation using CASPOC.
Reduced order Simulation with a Coenergy based system description

The amount of electrical energy transferred to the magnetic field and converted into mechanical work can be expressed as:

\[ dW_{electric} = dW_{magnetic} + dW_{mech} = u_{ind} idt - i^2 R dt \]  

(1)

Starting with a single coil excited linear actuator as shown in figure 2. The mechanical work done by the force moving the anchor for a distance \( dx \) in a given time interval \( dt \) is:

\[ dW_{mech} = F dx \]  

(2)

Taken the movement into account the effective flux is a function of the excitation \( N \) as the number of windings) and the actual displacement. Following this the induced voltage is given to:

\[ u_{ind} = N \frac{d}{dt} \Phi(iN, x) = N^2 \frac{\partial \Phi(iN, x)}{\partial(iN)} i + N \frac{\partial \Phi(iN, x)}{\partial x} \dot{x} \]  

(3)

The energy stored in the magnetic field can be expressed:

\[ W_{mag}(\Phi, x) = N \int_0^i \Phi(iN, x) d\Phi = iN \Phi(iN, x) - W_{mag}^{co}(iN, x) \]  

(4)

Computing the variation of the magnetic coenergy with respect to the current \( i \) and the displacement \( x \) gives:

\[ dW_{mag}^{co}(iN, x) = \frac{\partial W_{mag}^{co}(iN, x)}{\partial(iN)} d(iN) + \frac{\partial W_{mag}^{co}(iN, x)}{\partial x} dx \]  

(5)

and also:

\[ dW_{mag}^{co}(i, x) = \Phi di + F dx \]  

(6)

Therefore the effective flux can be determined from the coenergy as:

\[ \Phi(iN, x) = \frac{\partial W_{mag}^{co}(iN, x)}{\partial(iN)} \]  

(7)

and the force is given with:
Using these expressions the equations of motion for the electromechanical system can be derived as:

\[ m\ddot{x} + \frac{kx}{2} = F(iN, x) = -\frac{\partial W_{\text{mag}}^c(x)}{\partial x} \]

\[ u_{\text{ind}} = U - iR = N^2 \frac{\partial \Phi(iN, x)}{\partial (iN)} i + N \frac{\partial \Phi(iN, x)}{\partial x} \dot{x} \]

\[ = N^2 \frac{\partial^2 \Phi^c(iN, x)}{\partial (iN)^2} i + N \frac{\partial}{\partial x} \left( \frac{\partial W_{\text{mag}}^c(x)}{\partial (iN)} \right) \dot{x} \]  

(9)

Transforming these second order system into a system of 3 first order differential equations allows an easy numerical solution if the initial conditions are known.

This description reduces the characterization onto the main unknown of the electromechanical system - the excitation of the coil and the displacement of the anchor as a function of time. Thus it can be used as a basis of a reduced order simulation as long as the nonlinear behaviour of the magnetic circuit (saturation) is included correctly. This also allows the consideration of the anchor reaction correctly. The necessary computation of the coefficients can be done efficiently using a finite element simulation of the electromagnetic system.

**Electromagnetic finite element simulation**

As shown in equation (9) the main objective of a finite element simulation consists of the calculation of the magnetic coenergy as a function of a given excitation and the displacement. To do so a number of static magnetic field simulations must be done which involves the variation of those 2 (or even more) variables.

The simulation in ANSYS is ideally based on parametric model. If needed this can be build inside ANSYS with APDL (ANSYS Parametric Design Language). As the function of the magnetic coenergy should not contain to much noise, the mesh quality must be checked carefully. The mesh morphing opportunity suits well here as long as the displacements are small. Magnetic scalar potential formulation with SOLID 96 or SOLID 98 is preferred as the analysis are all static. This minimizes the modeling issues and allows an easy implementation of current sources with SOURCE36 elements. Figure 3 shows a typical model and the magnetic flux density.

![Figure 3 - Finite element model (left) of a linear actuator - flux density distribution (right)](image)
As the coenergy function must be derived two times (with respect to the excitation) for the system simulation, the number and position of calculating points must provide a good approximation of the complete coenergy function. Symmetric models as well as the symmetric character of the searched function can be used to minimize numerical simulation time. For the above shown example approximately 300 static simulations are needed to determine the coenergy as a function of the anchor position and the 2 excitations (source coil and eddy current coil). The used model consist of 2700 nodes and the simulation time was about 2.5 hours on a standard PC.

The resulting coenergy values can be obtained with the SENERGY macro or directly from the element table item (undocumented). For a two dimensional analysis this has to be checked as the magnetic field coenergy inside the coil region is not included. Once more the scripting language APDL can be used to handle and store the parameters and the coenergy as the result. Using the *vwrite command that information can be passed to an ASCII File for later use (within CASPOC e.g.).

**Eddy currents**

The shown simulation method based on equation (9) is only a basic characterization of an electromechanical system as it does not account the influence of eddy currents. For many practical application this effect must be included.

From the electrical point of view these currents simply represent another (not really known) current excitation due to the magnetic field change. The extended system of equations needs to consider the influence onto the flux and therefore to the induced voltage:

\[
\Phi(i, N_1, i_e, x) = \frac{\partial W_{mag}^{co}(i, N_1, i_e, x)}{\partial(i, N_1)}
\]

\[
u_{ind} = N_1^2 \frac{\partial \Phi(i, N_1, i_e, x)}{\partial(i, N_1)} i + N_1 \frac{\partial \Phi(i, N_1, i_e, x)}{\partial(i_e)} i_e + \frac{\partial \Phi(i, N_1, i_e, x)}{\partial x} \ddot{x}
\]

Also the resulting force includes this effect, the coenergy also depends on the eddy current.

Thus the influence of eddy currents can be included for the system simulation as long as the eddy current distribution and its amount can be estimated prior simulation. If no estimation can be done some harmonic magnetic analysis can help to predict the eddy current if the field change is reproduced correctly. Practical experiences shows that the exact eddy current distribution (and its exact position change within a moved anchor) can be neglected in many cases as long as the character of the distribution is stable.

**System Simulation**

The finite element simulation provides a set of discrete values of the coenergy function that can be used to describe the system behaviour according equation (9). With these values a twice continuous derivable function must be calculated. Many common system simulators such as MATLAB and SABER or even free tools (e.g. OCTAVE or SCILAB) can be used therefore.

As the design of the power electronics components is an integrated part of the electromechanical system development a simulation environment like CASPOC is preferred. This program with its easy to use GUI provides the broad functionality to realize the simulation of the power electronics together with the electromechanical system.

CASPOC supports the import of the coenergy function exported by ANSYS with several special block elements. After the values are imported into CASPOC a spline interpolation method calculates the necessary derivates at each position. This block element can be connected to any integrated circuit or block to characterize the electromechanical system. Figure 4 shows a very simple system simulation including a spring-mass-damper system and the electromagnetic actuator.
Within the simulation each nodal unknown (e.g. voltage, force, displacement) can be displayed as numerical value or within a scope. Also the motion of the magnetic device can be visualized with a nice animation tool.

Figure 4: Electromechanical system simulation in CASPOC

Beside the easy of use of CASPOC especially the minimized simulation time is very positive to be announced. This gives the opportunity to include several models representing damping.

The reduced order simulation based on the coenergy approach provides a fast and simple method to do a complete design study of an electromechanical system.

### Coupled field system simulation

Another way to characterize the electromechanical system would be a complete dynamic simulation inside a finite element program using a coupled field analysis. This allows to cover magnetic and mechanical nonlinearities (contact) as well as eddy current losses. Such a simulation supposes coupled field elements with a magnetic vector potential formulation (an additional electric scalar potential is necessary to represent eddy currents in 3D) as well as mechanical degrees of freedom to describe the motion.

\[
\{B\} = \nabla \times \{A\} \\
\{E\} = -\left(\frac{\partial A}{\partial t}\right) + \nabla V
\]

(12)

With PLANE13 and SOLID62 ANSYS provides coupled field elements for planar (or axisymmetric) and 3D applications. Using these elements for the complete model region, the transient simulation can be done with one unique model. On the other hand such a direct coupling leads to lengthy solution times due to the increasing number of unknowns.

### Model regions and meshing

For the solution of the equation of motion only the structural parts must be taken into account. The air region as well as the non moved parts can be neglected. The solution of Maxwell’s equation (with the finite element method) requires to model the structural parts and the surrounding air. If two separate
models can be used (one for the mechanics and one for the magnetics) the number of unknowns can be reduced and therefore the simulation time.

The use of a single element type with electromagnetic and mechanical degrees of freedom for the complete model also brings another problem along. Most of the electromechanical actuators have a small airgap which varies during the motion a few dimensions. This may deform the elements of the airgap region hardly Figure 5 (a).

![Figure 5 - Possible element shape distortion due to motion](image)

Some other applications such as rotating electrical machines have a constant airgap size but a large tangential motion that will also deform the provided mesh during the analysis Figure 5 (b). A third situation is conceivable if an anchor moves trough a small airgap and leaves this region or hits the ground. This would be a combination of the 2 mesh distortions of Figure 5 (a) and (b).

To overcome these problems ANSYS with its powerful APDL provides a variety of possibilities:

1) dissimilar meshes along the middle of the airgap and connect those regions using coupling and constraint equations
2) adjust the mesh in a sequential analysis using damorph, fvmorph commands
3) use anisotropic “dummy” material data for the non structural elements to adjust the deformation
4) remesh the non structural regions

As a transient analysis is needed to account the field coupling methods 1) to 3) can be easily implemented. But only the pure tangential movement (as for rotating electrical machines) allows larger displacements with method 1). Following this the large movement of a circuit breaker cannot be modeled. In such cases a remeshing is inevitable.

**Simulation**

The ANSYS user can realize this with the mesh morphing and its remesh option included. But this is not permitted in a transient analysis as the time integration scheme needs the previous solution onto the new created nodes. The procedure employed for the solution is an generalized trapezoidal rule:

\[
\{u_{n+1}\} = \{u_n\} + (1 - \Theta)\{\dot{u}_n\} \Delta t + \Theta \Delta t \{\ddot{u}_{n+1}\}
\]

(13)

where Theta is the transient time integration parameter.

A possible workaround can be found in a sequential analysis with an additional solution step that interpolates the solution. Doing so the movement of the magnetic part is simulated onto two separate steps:

1) remeshing and interpolation for the previous time step (change in position)
2) solution on next time step (change in time)
Realizing this approach inside ANSYS the solution time step size must be chosen carefully as the procedure does not satisfy an equilibrium state. Setting the time integration parameter to 1 results in a strong coupling to the actual solution step and minimizes the influence of the potential rise from the last solution.

The following command sequence represents the necessary solution scheme which has been tested on a simple magnetic induction loop.

```plaintext
! cbdof nodes (only once)
*if,key,eq,0,then
 esel,s,type,,2,3
 nwrite,nodes,inp
 alls
 ! start solution
 /solu
 antype,trans
 timint,off
 time,t_akt
 nsubst,2,2,2
 kbc,1
 solve
 *endif

! every next solution
*if,key,eq,1,then
 /output,scratch
 /nopr
 /nolist
 /inp,result,inp
 alls
 ! result interpolation
 /solu
 antype,trans
 outres,all,all
 timint,off
 kbc,1
 nsubst,nss,nss,nss
 time,t_alt
 solve
 *endif

! delete constraint
 alls,below,area
 nsel,u,ext
 ddel,all,all
 alls
 ! acutal time step
 time,t_akt
 timint,on
 nsub,ns2,ns2,ns2
 kbc,1
 neqit,25
 tintp,,,,1,,,,,1
 solve
 *endif
```

**Figure 6 - Transient solution routine with remeshing**

The tested model consists of a moved cylindrical permanent magnet and a coil winding (Figure 7). A high end resistance was used to minimize the reaction of the induced voltage. The induced voltages inside the coil shows a very good congruency with analytical results.

**Figure 7 - Test model for induction loop and analysis results**
With the prescribed coupled field solution inside ANSYS the correct field solution with respect to eddy currents can be determined. But as these solution must be transient, and the model has to include structural and magnetic part, the solution time is much higher compared to the reduced order simulation based on a the magnetic coenergy approach.

**Application Example (Medical Actuator)**

As a practical example a magnetic actuator for a medical application has been characterized. Figure 8 shows the part for the model used for a 3d magnetic analysis. The model consist of a primary coil excited with a controlled current, a fixed magnetic part (blue) and a moveable structure (red).

![Figure 8 - Model (left) and magnetic flux (right) of an electromagnetic for medical application](image)

The 3-dimensional model realized with the scalar potential formulation was used to compute the coenergy function with respect to the displacement of the moving part, the exciting current and the eddy current in the anchor. Simulation time was about 3 hours for more than 250 variations on a standard PC. Based on the calculated values a system simulation was done in CASPOC including prestress effect and eddy currents. Figure 9 shows the displacement vs time characteristic of the actuator. The computed results are close to the measurements in this region as damping effects are not important.

![Figure 9 - Displacement vs time characteristic of the actuator](image)
The compared transient coupled field simulation (magnetic-mechanical) was performed using a 2d-axisymmetric model using PLANE53, CIRCUIT124 as well as MASS21, COMBINE14 and CONTACT52 for the mechanical part. As the large movement needs to be taken into account a remeshing method was used. The overall simulation time for the system with its sequentially coupling included eddy currents, prestress effect and contact was about 40 minutes on a actual PC. Obtained results shows a good agreement to the measured curve with a small error (10%) in the first part of the curve.

**Conclusion**

The characterization of an electromechanical system can be realized with either of described methods – the reduced order based on a coenergy approach with CASPOC for the system simulation and also the coupled field simulation inside ANSYS.

As long as the eddy current distribution can be estimated and the model needs to realized in 3d, the reduced order method should be preferred. Doing so the parameter extraction for the coenergy function must be done once but the electrical circuit simulation within CASPOC can be realized with a minimum of effort and time (and this for many different electrical signal characteristics). The method also allows to cover eddy current effects with a minimum of overall simulation time.

On the other hand if the eddy current distribution could not be estimated prior simulation or the analysis can be done within 2d modeling – the coupled field simulation inside ANSYS provides an alternative way and leads to correct simulation results. This simulation method could be effectively realized using the power of APDL – but the simulation effort could be improved in the future to minimize users input.

Combining those two methods ANSYS provides an practical software simulation environment for electromechanical system simulation. The analysis strategy can be adapted to the task requirements and the users desire.

**References**

2) Dynamiksimulation im Relais: Projektion auf drei Freiheitsgrade mit Hilfe der FEM, Martin Hanke, Jürgen Breitlow-Hertzfeld, SIEMENS AG, Berlin, 1997
3) CAD of Electromechanical and Electromagnetic Devices , Dr. Stephane Dufour, Dr. Gerard Vinsard, Nancy 2001
4) Modelling of electromechanical Systems, Werner Haas, Kurt Schlacher und Reinhard Gahleitner, Johannes Kepler University Linz, 2000