EUROPEAN UNION
European Structural and Investment Funds
Operational Programme Research,
Development and Education

MINISTRY OF EDUCATION,
YOUTH AND SPORTS

Selected Topics in Electrical Machines

(KEV/VPS)

Modeling of vibration excited by magnetic forces in electrical machines

(Supporting materials for tutorials and student work)

ESF II project of the University of West Bohemia in Pilsen

reg. number: CZ.02.2.69/0.0/0.0/18_056/0013239

Jan Šobra

2021

EUROPEAN UNION
European Structural and Investment Funds
Operational Programme Research,
Development and Education

MINISTRY OF EDUCATION,
YOUTH AND SPORTS

List of content

List of content 2
Symbols and abbreviations 3
Introduction 5
1 Methodology of electromagnetically excited vibrations modeling 6
1.1 Magnetic transient analysis 6
1.2 Discrete Fourier transform 8
1.3 Harmonic response analysis 10
2 Modeling procedure in the ANSYS software package 13
2.1 Magnetic transient analysis 15
2.2 Mechanical analyses 18
Conclusions 26
References 27

Symbols and abbreviations

Symbol	Unit	Explanation
B	T	Magnetic flux density
$\boldsymbol{B}$	[-]	Damping matrix
D	C/m^2	Electric flux density
E	V/m	Electric field intensity
F	N	Force
$\boldsymbol{f}$	N	Excitation force vector
f_{max}	Hz	FFT frequency range
f_{vz}	Hz	Sampling frequency
H	A/m	Magnetic field intensity
J	A/m^2	Current density
$\boldsymbol{K}$	N/m	Stiffness matrix
$\boldsymbol{M}$	kg	Mass matrix
n	min^{-1}	Revolutions per minute
$\boldsymbol{n}$	[-]	Surface normal vector
n_{vz}	-	Number of samples for FFT
$\boldsymbol{q}$	m	Generalized coordinates vector
S	m^2	Surface
T	s	Time interval for sample recording
$\boldsymbol{T}$	[-]	Maxwell stress tensor
t	s	Time

$\boldsymbol{V}$		Modal matrix
V	m^3	Volume
$\boldsymbol{x}$		Modal coordinates vector
Δf	Hz	FFT frequency step
Δf	N/m^3	Volume force density
Δp	N/m^2	Surface force density
Δt	s	Simulation time-step
$\boldsymbol{\Lambda}$	[-]	Eigenvalue spectral matrix
μ	H/m	Permeability
ξ	rad	Angle position around statore circumference
ρ	C/m^3	Volume charge density
ω	$rad.s^{-1}$	Angular frequency
Ω	Hz	Natural frequency

Abbreviations

DFT	Discrete Fourier transform
FFT	Fast Fourier transform
FEM	Finite element method
MST	Maxwell Stress Tensor

Introduction

Stator system vibrations are linked to the machine noise. The spectrum and, to a certain extent, also the level of magnetically excited vibrations and noise can be preliminarily verified utilizing models already during the design of the machine. If necessary, measures to reduce vibration can be taken in time. When designing vibration reduction measures on existing machines, the models can help identify the root cause of the vibration as well as verify the effectiveness of the proposed measures. Electromagnetically excited vibrations are caused by time-varying magnetic forces which act on the stator teeth and are transmitted through the yoke to the machine frame. They can be further transmitted to other drive components and are also radiated to the environment in the form of noise.

The magnitude of the magnetic forces acting on the stator and rotor of the machine is derived from the square of air gap magnetic flux density waveform. Vibration level is influenced mainly by radial component of flux density or magnetic force respectively. However, certain vibrations are also excited by flux density / force tangential component, which also causes bending stress of the teeth. In the case of skewed stator or rotor slots, respectively permanent magnets skewing, an axial component of the force also originates. These forces originate at the interface of two materials with different permeability and can be calculated using Maxwell Stress Tensor (MST). Furthermore, vibrations of the core lamintion are caused due to magnetostriction – i.e. inside the ferromagnetic material. The Lorentz forces acting on the conductors inside slots are neglected. More information on the forces in the magnetic field can be found in [1], [2].

The aim of the document is to summarize the data for the course computer tutorials and to provide students with support for individual work solving. There is a guide to modeling magnetically excited stator vibrations using software for multiphysics analysis based on the finite element method (FEM) and describes the methodology on which the modeling is based.

The instructions below are performed in the ANSYS software package [3], but they can also be implemented in other software that allow the solution of the types of analyzes described below. Knowledge of the ANSYS Maxwell program environment and experience with modeling of electrical machines gained in previous studies is assumed. Familiarization with the user environment of ANSYS Workbench and Mechanical programs will take place within the course exercises and is not described in more detail in this text.

1 Methodology of electromagnetically excited vibrations modeling

Methodology of electromagnetically excited vibrations modeling is based on coupling of magnetic transient analysis (time variant magnetic forces) and harmonic response analysis (vibration). General block diagram of the model is provided in Fig. 1.1 a and it can be implemented by any software package having tools for solving the both mentioned analyses [4].

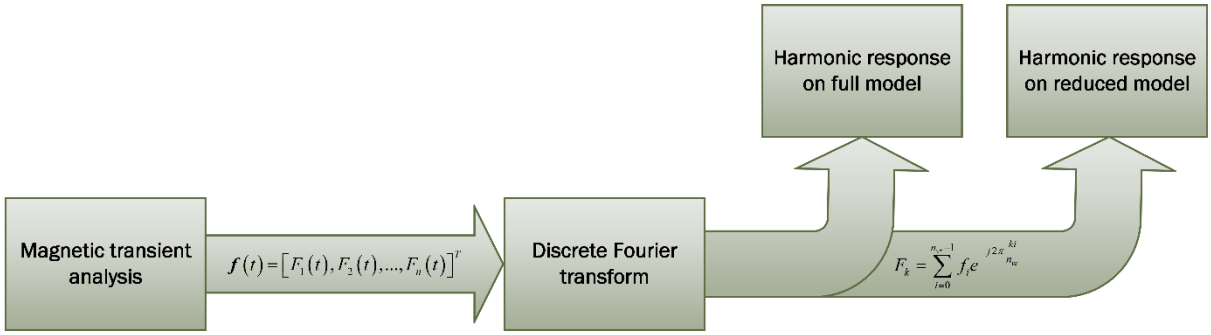


Fig. 1.1: Methodology block diagram.

1.1 Magnetic transient analysis

Magnetic transient analysis is based on solution of Maxwell's equations for a non-stationary electromagnetic field in a time domain and it is also possible to include the effects of revolving rotor. Maxwell equations in differential form for a low frequency time variant electromagnetic field are in (1) to (4). Displacement currents can be neglected in that case.

$$rot\,\boldsymbol{H}=\boldsymbol{J} \tag{1}$$

$$rot\,\boldsymbol{E}=-\frac{\partial \boldsymbol{B}}{\partial t} \tag{2}$$

$$div\,\boldsymbol{D}=\rho \tag{3}$$

$$div\,\boldsymbol{B}=0 \tag{4}$$

Analysis can be solved for the specific geometry in both 2D and 3D, however, resulting forces in 2D must respect the axial (effective) length of the lamination. Forces acting on individual teeth can be determined as an integral of surface force density for 2D analysis or volume force density for 3D

analysis respectively. Integrals are calculated on the geometry part relevant to the particular teeth. Surface and volume force densities are obtained via Maxwell stress tensor.

Maxwell stress tensor (MST) is utilized for a force calculation on the borderline of two materials with different permeability. For analysis of forces acting on the stator and rotor teeth, the borderline is ferromagnetic material – air. In general case – for environment having permeability μ exposed to magnetic field with a flux density $\boldsymbol{B}=(B_x,B_y,B_z)$, MST can be expressed by symmetrical matrix [1], [5]

$$\boldsymbol{T}=\frac{1}{\mu}\begin{bmatrix} B_x^2-\frac{1}{2}|\boldsymbol{B}|^2 & B_xB_y & B_xB_z \\ B_xB_y & B_y^2-\frac{1}{2}|\boldsymbol{B}|^2 & B_yB_z \\ B_xB_z & B_yB_z & B_z^2-\frac{1}{2}|\boldsymbol{B}|^2 \end{bmatrix} \tag{5}$$

Integrals are calculated on the geometry part relevant to the particular teeth. Surface and volume force densities are obtained via Maxwell stress tensor.

Volume force density $\Delta\boldsymbol{f}$ can be expressed using MST as

$$\Delta\boldsymbol{f}=div\boldsymbol{T} \tag{6}$$

and total force $\boldsymbol{F}$ acting on a body with the volume V

$$\boldsymbol{F}=\int_V \Delta\boldsymbol{f}\,dV \tag{7}$$

Surface force density or a specific magnetic pressure (unit $N\cdot m^{-2}=Pa$) can be determined directly from MST

$$\Delta\boldsymbol{p}=\boldsymbol{T}\cdot\boldsymbol{n} \tag{8}$$

where $\boldsymbol{n}$ is surface normal vector.

Force acting on the face having surface S can be calculated as

$$\boldsymbol{F}=\int_S \Delta\boldsymbol{p}\,dS \tag{9}$$

For a 2D analysis of forces acting on the ferromagnetic material – air borderline, total force can be expressed by

$$\boldsymbol{F} = \frac{1}{\mu_0}\int_S \begin{bmatrix} B_x^2 - \frac{1}{2}|B|^2 & B_x B_y \\ B_x B_y & B_y^2 - \frac{1}{2}|B|^2 \end{bmatrix} \begin{Bmatrix} n_x \\ n_y \end{Bmatrix} dS \qquad (10)$$

where: $\boldsymbol{F}$ is total force acting on the face with the surface S

μ_0 is permeability of air

B_x, B_y are flux density components in the Cartesian coordinate system

$|B|$ is absolute value of flux density vector

n_x, n_y are components of the normal unit vector of the surface.

Transformation to radial and tangential components is often more suitable for results representation clarity.

$$F_r = F_x \cos\xi_z + F_y \sin\xi_z \qquad (11)$$

$$F_t = -F_x \sin\xi_z + F_y \cos\xi_z \qquad (12)$$

Equations (11) and (12), F_r a F_t are forces transformed into radial and tangentioal direction, F_x a F_y are forces acting on a specific tooth along *x* and *y* directions, and ξ_z is the angle of a specific tooth axis measured counter-clockwise from global *x* axis.

1.2 Discrete Fourier transform

Since the magnetic forces obtained as an output from the magnetic transient analysis are a function of time, it is necessary to transform them into the frequency domain in which the analysis of the harmonic response is solved. This is solved by means of discrete Fourier transform (DFT), where the time variant force wave is decomposed into real and imaginary components corresponding to individual superimposed harmonic waveforms.

The time vector of excitation forces can be written using the Fourier series according to (13), the discrete Fourier transform is then performed according to (14), where n_{vz} is the number of samples and $0 \le k \le n_{vz} - 1$. It is important that the DFT is performed on a time sample obtained at a steady state (both electrical and mechanical).

$$\boldsymbol{f}(t) = F_0 + \sum_{i=0}^{n} F_{ci} \cos(\omega_i t) + F_{si} \sin(\omega_i t) \tag{13}$$

$$F_k = \sum_{i=0}^{n_{vz}-1} f_i e^{-j2\pi \frac{ki}{n_{vz}}} \tag{14}$$

A very common algorithm for DFT is the Fast Fourier Transform (FFT). Below are the FFT parameters resulting from the magnetic transient analysis setup. The sampling frequency of the magnetic force wave at a constant simulation step is given by its inverse value according to (15)

$$f_{vz} = \frac{1}{\Delta t}, \tag{15}$$

where f_{vz} is sampling frequency and Δt is time step of the simulation.

The number of samples for FFT is then

$$n_{vz} = \frac{T}{\Delta t}, \tag{16}$$

where n_{vz} is the number of samples and T is time period for which samples are recorded.

Frequency step (resolution) Δf of FFT is given by (17)

$$\Delta f = \frac{f_{vz}}{n_{vz}}. \tag{17}$$

Frequency range $f_{\max}$ of FFT (Nyquist frequency) is

$$f_{\max} = \frac{f_{vz}}{2}. \tag{18}$$

From the above relationships, it is clear that in order to achieve a larger FFT frequency range with a sufficiently fine step, it is necessary to perform a transient analysis over a relatively long period of time with a very small time step. This, of course, leads to a long solution time and considerable demands on the free capacity of a drive where the result files are stored. Considering the duration of the calculations, it is therefore appropriate to use licenses for parallel calculations on multiple cores or workstations.

1.3 Harmonic response analysis

The analysis of harmonic response can then be solved using a full model, where the equations of damped forced oscillations are solved for each excitation frequency or using a reduced model, where the solution is narrowed only to the region of eigenvectors (modal superposition).

The vector of magnetic forces transformed to the frequency domain is used as an input (excitation) in the harmonic response analysis. The theory of forced oscillation and modal transformation below is elaborated according to [6], where a complete derivation of the equations below is provided.

The equation of motion of forced damped oscillations of a linear system with lumped parameters can be written in matrix form as

$$\boldsymbol{M}\ddot{\boldsymbol{q}} + \boldsymbol{B}\dot{\boldsymbol{q}} + \boldsymbol{K}\boldsymbol{q} = \boldsymbol{f}(t), \tag{19}$$

where $\boldsymbol{M} = \left[m_{ij}\right]$ is a mass matrix, $\boldsymbol{B} = \left[b_{ij}\right]$ is a damping matrix, $\boldsymbol{K} = \left[k_{ij}\right]$ is a stiffness matrix, $\boldsymbol{q}(t) = \left[q_1(t), q_2(t), \ldots, q_n(t)\right]^T$ is a vector of generalized coordinates and $\boldsymbol{f}(t) = \left[F_1(t), F_2(t), \ldots, F_n(t)\right]^T$ is an excitation vector.

Model according to (19) is conservative if $\boldsymbol{K}=\boldsymbol{K}^T$ and $\boldsymbol{B}=\boldsymbol{0}$ or non-conservative if $\boldsymbol{K}=\boldsymbol{K}^T$ and damping matrix is in form (20) or (21).

Damping matrix $\boldsymbol{B}$ meeting the condition of proportional damping

$$\boldsymbol{B} = \sum_{j=0}^{r-1} c_j \boldsymbol{M}\left(\boldsymbol{M}^{-1}\boldsymbol{K}\right)^j \quad \text{for } 1 \le r \le n \tag{20}$$

or a commutative damping matrix

$$\boldsymbol{K}\boldsymbol{M}^{-1}\boldsymbol{B} = \boldsymbol{B}\boldsymbol{M}^{-1}\boldsymbol{K} \tag{21}$$

can be used for real systems (models) only in the case of weakly damped systems, where no functional damping members are present (shock absorbers, etc.) and the damping matrix $\boldsymbol{B}$ represents only material or weak structural damping.

When modeling the free oscillation of conservative systems, zero damping and excitation are considered, i.e. $\boldsymbol{B}=\boldsymbol{0}$ a $\boldsymbol{f}(t)=\boldsymbol{0}$. The equation of motion of free oscillation is in this case

$$\boldsymbol{M}\ddot{\boldsymbol{q}}(t) + \boldsymbol{K}\boldsymbol{q}(t) = 0\,. \tag{22}$$

The transformation of generalized coordinates $\boldsymbol{q}(t) = \left[q_1(t), q_2(t), \ldots, q_n(t)\right]^T$ to modal coordinates $\boldsymbol{x}(t) = \left[x_1(t), x_2(t), \ldots, x_n(t)\right]^T$ according to (23) is called a modal transformation. The solution of the dynamic response of systems based on modal transformation is called modal analysis.

$$\boldsymbol{q}(t) = \sum_{\nu=1}^{n} v_\nu x_\nu(t) = \boldsymbol{V}\boldsymbol{x}(t) \tag{23}$$

Substituting (23) into equation (22), the equation of motion of the free oscillation changes into shape

$$\boldsymbol{M}\boldsymbol{V}\,\ddot{\boldsymbol{x}}(t) + \boldsymbol{K}\boldsymbol{V}\,\boldsymbol{x}(t) = 0\,, \tag{24}$$

where $\boldsymbol{V} = \left[v_\nu\right] \in R^{n,n}$ is a modal matrix composed of eigenvectors that describe the eigenshape (natural shape) of the oscillation at the corresponding eigenfrequency (natural frequency). By multiplying the left-transposed modal matrix $\boldsymbol{V}^T$, equation (24) can be converted to a shape

$$\ddot{\boldsymbol{x}}(t) + \boldsymbol{\Lambda}\boldsymbol{x}(t) = 0\,, \tag{25}$$

where $\boldsymbol{\Lambda} = diag\left(\lambda_\nu\right) \in R^{n,n}$ is a diagonal spectral matrix with eigenvalues $\lambda_\nu = \Omega_\nu^2$ and the corresponding natural frequency can be determined as $\Omega_\nu = +\sqrt{\lambda_\nu}$.

By reducing the order of the matrix from the original value of *n* to *m* and considering non-zero excitation, it is possible to narrow the solution of the response of forced oscillations to the region of eigenvectors of predefined range (26) and thus simplify the model and significantly save computational time.

$$\ddot{\boldsymbol{x}}(t) + \boldsymbol{\Lambda}^{m}\boldsymbol{x}(t) = \boldsymbol{V}^{mT}\boldsymbol{f}(t) \tag{26}$$

If the time vectors of the excitation forces in (19) and (26) were replaced by the vectors of forces transformed into the frequency domain, the solution of the harmonic response according to (27) would correspond to the use of the full model and the solution according to (28) would correspond to the use of the reduced model.

$$\boldsymbol{M}\ddot{\boldsymbol{q}} + \boldsymbol{B}\dot{\boldsymbol{q}} + \boldsymbol{K}\boldsymbol{q} = \boldsymbol{f}(\omega) \tag{27}$$

$$\ddot{\boldsymbol{x}}(\omega) + \boldsymbol{\Lambda}^{m}\boldsymbol{x}(\omega) = \boldsymbol{V}^{mT}\boldsymbol{f}(\omega) \tag{28}$$

2 Modeling procedure in the ANSYS software package

The above methodology is applied in the ANSYS software package. Magnetic transient analysis is performed in ANSYS Maxwell program and harmonic response analysis is performed in ANSYS Mechanical program. Modal analysis and static structural analysis as a supplement are performed in ANSYS Mechanical as well. The general way of connecting magnetism (harmonic analysis of forces) and mechanics (harmonic response of vibrations) in the ANSYS Workbench environment is described in the help systems of individual products [7], [8]. The subchapters below provide specifics for rotating electrical machines.

The complete scheme of solved multiphysics problems depending on the chosen method of harmonic response analysis is shown in the block diagram in Fig. 2.1 or in Fig. 2.2 respectively. In ANSYS Mechanical, the solution of the harmonic response using the full model is called Full analysis, while the solution using the reduced model is referred as Mode Superposition analysis.

It can be seen from the diagram that the models for magnetic and mechanical analysis have their own geometry in the example given. This is due to the different simplification of the machine geometry and thus the required computational time and hardware requirements for both types of analyses.

Magnetism is calculated in 2D on the geometry of the magnetic circuit cross section (but can be solved in 3D as well) and with regard to the solved problem it is not necessary to consider the influence of the machine frame on magnetic field distribution. The mechanics is always calculated on 3D geometry of the solely stator stack or the stator packet set into the frame. The influence of the frame is crucial for modal analysis and harmonic response analysis. Conversely, the effect of the rotor on the vibrations of the stator stack can often be neglected due to oversized thickness of bearing shields. However, it depends on the specific circumstances of the model and especially the solution domain that designer is interested in the analysis.

If both analyzes (magnetism and harmonic response) were solved in 3D, it is also possible to use a common geometry and set negligible parts of the geometry in individual analyzes as *non-model objects*.

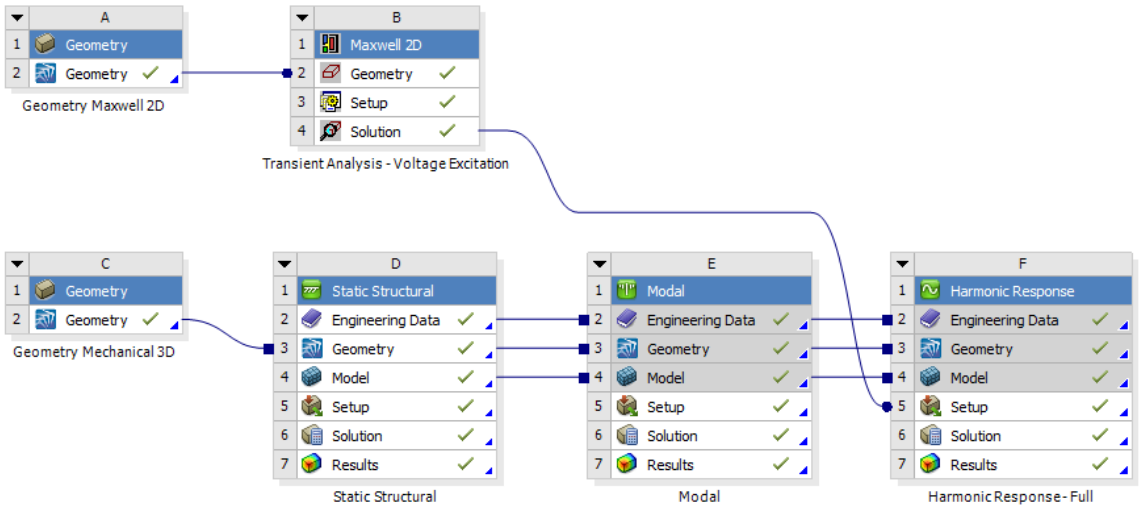


Fig. 2.1: Scheme of the solved problem in the ANSYS Workbench environment for the analysis of the harmonic response using the full model.

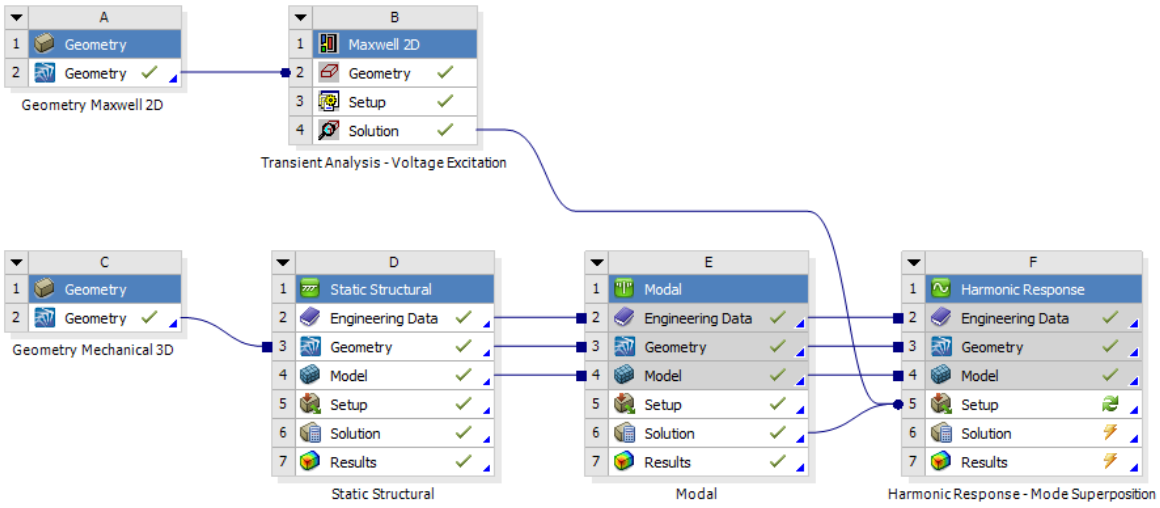


Fig. 2.2: Scheme of the solved problem in the ANSYS Workbench environment for the analysis of the harmonic response using the reduced model.

2.1 Magnetic transient analysis

In the ANSYS Maxwell program, it is necessary to perform a 2D or 3D magnetic transient analysis of the machine under the required operating state (load). In order to calculate the magnetic forces correctly, it is necessary to enter the axial length of the lamination stack in the case of 2D analysis. To save computing time, hardware requirements and data storage capacity, it is possible to use eventual symmetries in the geometry of the analyzed machine. However, this does not apply to modeling of faults or failures of the machine, which by their nature cause asymmetry of the magnetic field in the machine, or are caused by asymmetry of the machine structural configuration (eccentrically mounted rotor, etc.).

In the case of 2D analysis, stator and/or rotor slot or permanent magnet skewing in the axial direction is not usually considered. However, this can possibly be approximated by solving the analysis using the so-called *multi-slice model*. In this case, the machine is divided along a defined length in the axial direction into the required number of partial stacks, which are rotated by each other by such an angle that the angle between the first and the last packet corresponds to the total angle of the stator or rotors skewing.

As mentioned above, it is necessary to set the excitation and rotational speed corresponding to the solved load. At the same time, it is not necessary to consider a mechanical transient, as the solution of the harmonic response in a steady state is assumed. In practice, this setting corresponds to the case where a revolving unexcited machine is suddenly connected to a power supply. Then, at the beginning of the analysis, there is only an electrical transient, the fading of which can be significantly accelerated by selecting *Fast reach transient* and setting the supply frequency in the analysis settings.

It is possible to adjust the refinement of the finite element mesh in important areas of geometry using *Mesh Operations* tool. Because the Maxwell solver does not allow adaptive meshing for transient analyzes, it is necessary to set the mesh refinement manually to achieve the required calculation accuracy. In this case, the mesh must be refined, especially in the stator / rotor teeth and the air gap, where the force effects are calculated.

From the point of view of calculating magnetic forces acting on individual stator and / or rotor teeth, it is necessary to model each tooth as a separate object in the task geometry (see Fig. 2.3) and set the calculation of forces on selected geometry parts (teeth) in the *Parameters* item.

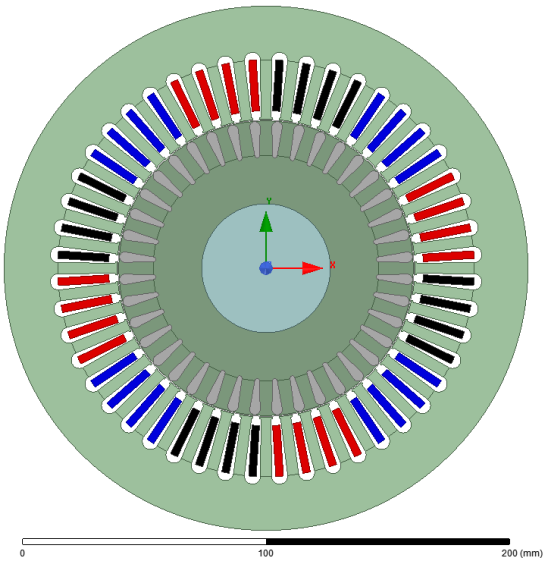


Fig. 2.3: Example of machine geometry – stator and rotor teetch modeled as a separate objects.

In the case of transient analysis, it is necessary to set the start and end time and the time step of the simulation in the *Analysis* item. The choice of the time step and the length of the simulation directly indicates the frequency range and frequency step of the fast Fourier transform (FFT) in the conversion of the forces waveform acting on the teeth from the time domain to the frequency domain - see the chapter 1.2.

Furthermore, it is possible to set the time instants in which the calculated quantities of the magnetic field, in particular the forces acting on the teeth, should be saved. Saving the results of magnetic field quantities for each solved step requires considerable free storage capacity and is more suitable for visualization (pictures, animations) of forces acting on the teeth during the analysis. In example in Fig. 2.4, which is solved on a four-pole induction machine with an output power of 18 kW, shows the waveform of magnetic forces acting on one stator tooth in the radial and tangential direction. At the waveform beginning, the above-mentioned short transient can be seen. An example of surface force density vectors acting on the same tooth at time step 60 ms is shown in Fig. 2.5.

It is not necessary to save magnetic field distribution results to convert the forces to the frequency domain using FFT. However, it is necessary to define the time interval and forces on which the FFT is to be performed. This can be done in *Enable harmonic force calculation* item, where only forces whose calculation was previously set in *Parameters* item can be selected.

In addition to the interval before the end of the simulation, it is also possible to define the time period for the FFT as the number of mechanical cycles before the end of the simulated time

period. In that case the time interval when the machine is already in a steady state should be selected. In the example shown in Fig. 2.4, this corresponds to an interval of approx. 60 to 360 ms. If a longer time window for FFT is required, it is possible to enter multiple repetitions of the selected interval or the number of mechanical cycles respectively.

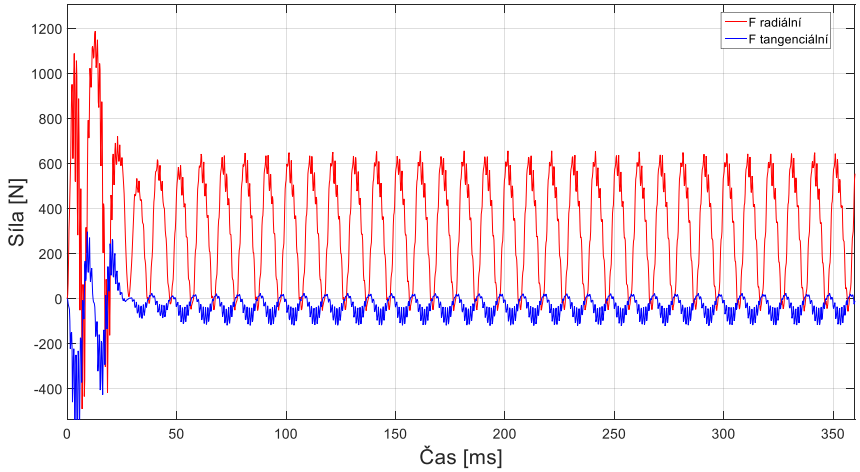


Fig. 2.4: Radial and tangential force component acting on a single stator tooth during transient analysis.

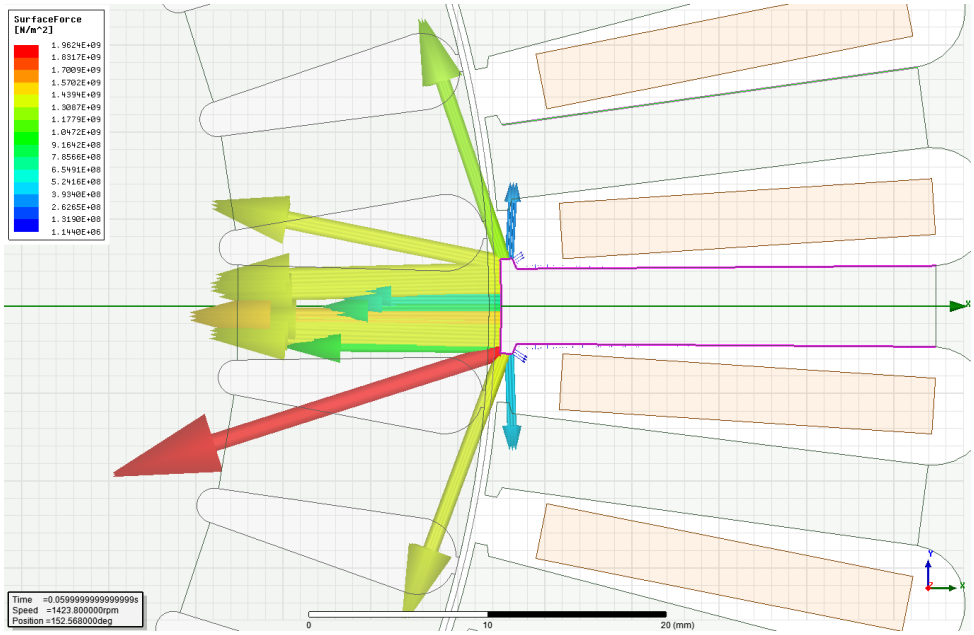


Fig. 2.5: Surface force density vectors acting on a single stator tooth.

2.2 Mechanical analyses

As can be seen in Fig. 2.1 and Fig. 2.2, the "mechanical branch" of models consists of static structural analysis, modal analysis, and finally harmonic response analysis – all solved in ANSYS Mechanical. However, it is not always necessary to include all the above analyzes in the model. Analyzes can share geometry, material parameters and mesh – so it is not necessary to define these for each analysis separately.

Static structural analysis should be preceded in the case of geometry consisting of a larger number of solids (e.g. teeth modeled as separate volumes). In this case, the purpose of the analysis is to verify the integrity of the model, i.e. whether the contacts between the individual bodies are set correctly. If the geometry is formed by a single solid body, the analysis is not necessary.

Preceding of modal analysis is necessary in the case of solving the harmonic response using a reduced model. The harmonic response is then solved only in the vicinity of the natural frequencies obtained from the modal analysis. If the harmonic response is solved on a full model, it is not necessary to precede the modal analysis. In this case, however, the modal analysis can serve as a useful complement, where it is possible to verify whether the increased vibrations are related to the proximity of stator stack or the stator system assembly natural frequency.

Harmonic response, as well as all the mentioned mechanical analyzes can be solved on a 3D model of either a solely stator stack (Fig. 2.6) or a stack mounted into the machine frame (Fig. 2.7), which can be further attached to the foundation or other structure. However, the basis of each variant is always a stator stack. Therefore, origin of the coordinate system should be placed in the center of stator stack and other components should be assembled on it after. This is important with regard to evaluation of the analysis. If the origin of the global coordinate system cannot be placed in the center of the packet, it is advisable to define your own coordinate system there.

Material of the stator packet is an anisotropic material exhibiting lumped parameters corresponding to the real behavior of this machine part. It is necessary to take into account the inhomogeneity of the stator stack, which is not an integral part of one material, but it is composed of thin electrical steel sheets provided with a thin layer of insulating varnish on both sides. It has been found [9], that the natural frequencies and natural shapes, and thus the harmonic response of a packet, cannot be modeled using the material properties of steel or a similar isotropic material.

By measuring the real stack and subsequent tuning of the finite element model, it is possible to determine lumped Young's modulus (modulus of elasticity) of the stator stack (or equivalent anisotropic material) modeled as a single part. By measuring and modeling in [9], a significant

difference of two orders in the Young's modulus magnitude in the axial direction was found. Lumped Young's and shear modulus of the stack determined in this way are summarized in Tab. 1 and can be used with relatively good accuracy in other machines of similar dimensions (outer diameter of the stator $D_e = 220\ mm$, inner diameter of the stator $D_1 = 129\ mm$, package length $l = 100\ mm$), but for significantly larger machines the results of modal analysis differ considerably.

Tab. 1: Lumped stator stack parameters according to [9].

Parameter	***Value***	***Unit***
Young's modulus E_x	$2{,}05E^{11}$	Pa
Young's modulus E_y	$2{,}05E^{11}$	Pa
Young's modulus E_z	$1{,}5E^{9}$	Pa
Shear modulus G_{xy}	$2{,}5E^{10}$	Pa
Shear modulus G_{yz}	$1{,}5E^{9}$	Pa
Shear modulus G_{xz}	$1{,}5E^{9}$	Pa
Poisson's ratio ν_{xy}	0,3	-
Poisson's ratio ν_{yz}	0,3	-
Poisson's ratio ν_{xz}	0,3	-

When defining lumped material parameters of the stack, it is also appropriate to include in the mass density of the lumped material a contribution from the mass of the winding. The mass of the stator system affects vibration damping and the modeling of the winding as a separate part is a problematic issue in the point of view of turn separation, binding and fastening of endwindings or defining contacts between the winding and the slot.

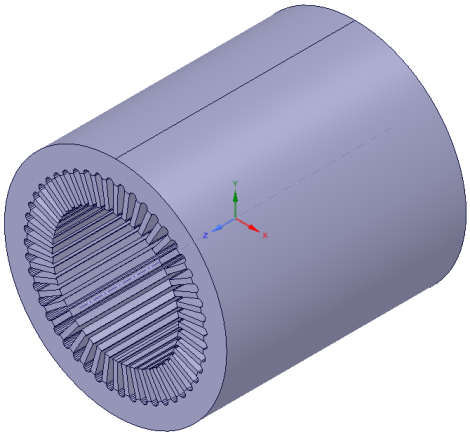


Fig. 2.6: Solely stator stack geometry.

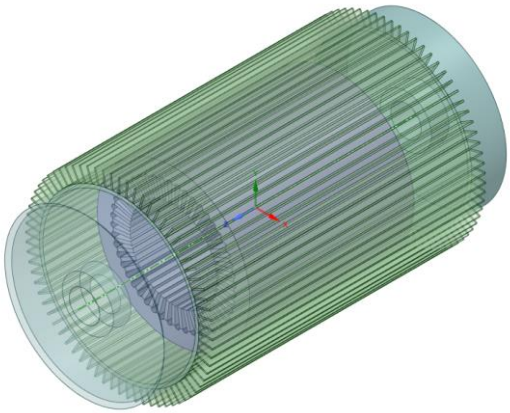


Fig. 2.7: Geometry of stack mounted into the machine frame.

2.2.1 Static structural analysis

In this case, static structural analysis serves only to verify the integrity of individual parts of the model and its implementation is therefore not absolutely necessary for vibration analysis (unless it is necessary to consider the pre-stress of some structural parts). A certain part of the geometry, usually some face, is fixed and only the gravitation acts on the whole model. Then it is verified whether the individual parts of the geometry are still firmly connected in one assembly, i.e. whether the connections between the individual geometry objects (e.g. between the teeth and the stator

yoke, stator plate and frame, etc.) are set correctly and the assembly does not split into individual parts.

In Fig. 2.8 an example is given for the model of the solely stator stack, where the individual teeth are geometrically separated from the stator yoke, but in the model a fixed (*bonded*) contact is set between them and thus there is no deformation of the contact surfaces. Therefore, from a physical point of view, the stack behaves as a single body. The face of the tooth facing the air gap is fixed and gravity acts in the negative direction of the *z*-axis. This also corresponds to the stress of the individual bodies shown in Fig. 2.8, where the most stressed is the tooth with the fixed face.

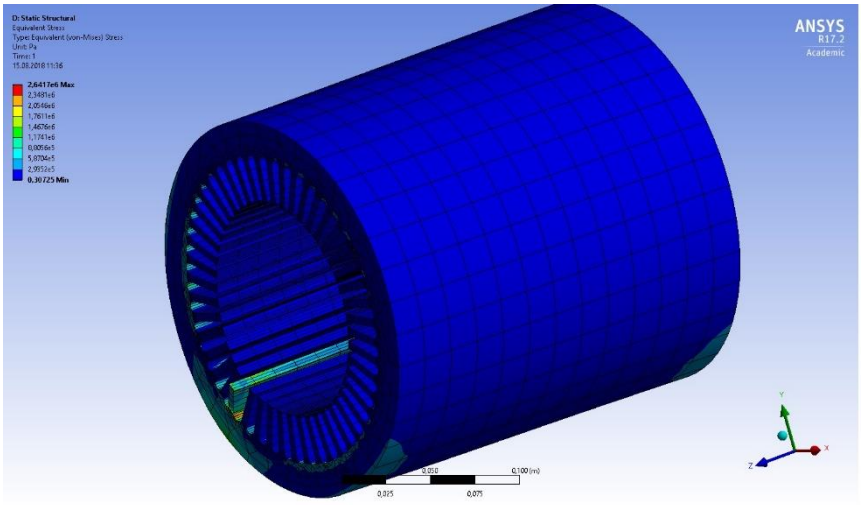


Fig. 2.8: Stress of a stator stack model in static structural analysis.

2.2.2 Modal analysis

In modal analysis, the *fixed support* of certain parts of the geometry with regard to the expected mounting of the stack into the machine frame or the motor fastening to the foundation or other supporting structure should be set as a boundary condition. If only the solely stack is modeled, the end faces of the stator yoke are usually fixed at the both ends of the package (it is necessary to geometrically separate the yoke from the teeth – see Fig. 2.11), or analysis can be performed for a free suspended stack (without fixed support). On larger machines, where the stack is contracted using annulus plates with contracting fingers on individual teeth, it is possible to fixed the entire front surfaces of the pack – i.e. the yoke and the teeth. In the case of modeling a stack mounted into the frame, it is possible to embed the bearing surfaces of the feet, or screw holes in the feet or flange on the frame – depending on the frame design.

In the analysis settings, it is necessary to set the maximum number of natural shapes (*modes*) and corresponding natural frequencies to be identified. Also the frequency range, in which the modal analysis should be solved, can be limited. It is also possible to set whether the analysis should be solved as damped (damping, stiffness and mass coefficients must be set) or undamped.

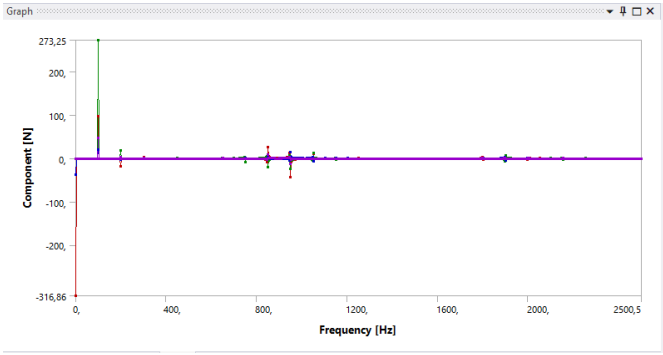
When the analysis is solved, the mode shapes can be visualized for each detected natural frequency. The results can also be used as input (analysis range setting) for harmonic response analysis on a reduced model.

2.2.3 Harmonic response analysis

In the case of harmonic response solved using a full model, it is necessary to set the table of excitation force frequencies and boundary conditions (e.g. fixing a certain parts of geometry - see section **Chyba! Nenalezen zdroj odkazů.**). In the case of harmonic response solved using the reduced model, the frequencies of excitation forces and initial conditions are given by the modal analysis solution (it is necessary to connect analyzes according to Fig. 2.2) and if necessary, selected table-set frequencies for full model solving can be entered (significant rise of solution time). If the solution using the full model will be compared with the results of modal analysis, it is appropriate to set the same boundary conditions for both tasks.

To import forces from magnetic transient analysis, right-click *Imported Remote Loads* (available when Mechanical is linked to Maxwell) and then select *Generate Remote Loads*. For each frequency from the FFT, the forces are decomposed in the real and imaginary components acting in the *x* and *y* axes (in the case of 3D magnetic analysis also in the *z* axis direction) and the moments (again decomposed in the real and imaginary components) acting in the *z* axis (or other axes for 3D magnetic analysis) – see Fig. 2.9.

In the harmonic response settings, it is necessary to link each tooth force and moment to the face normal to the air gap on the corresponding tooth tip – see Fig. 2.10. This can be done by selecting the face directly (*Geometry Selection*) or by predefined name of the face (*Named Selection*). An example of imported forces and moments acting on several selected teeth is shown in Fig. 2.11. It is clear from the arrows orientation that the forces do not act only in the radial direction, but the influence of the tangential component is evident. In the same way, a forces must be applied to all stator teeth.



Tabular Data

	Frequency [Hz]	X Component - Real [N]	X Component - Imag [N]	Y Component - Real [N]	Y Component - I
1	0,	-316,86	0,	-37,813	0,
2	3,3333	2,6198e-002	-5,9657e-002	-7,1089e-003	-1,9332e-003
3	6,6667	-3,0812e-002	-2,4217e-002	-2,7663e-002	2,5249e-002
4	10,	-4,017e-002	-2,5154e-002	-7,9958e-003	2,1532e-002
5	13,333	-6,2702e-002	-2,0848e-002	-3,0253e-003	2,1108e-002
6	16,667	-6,0705e-002	-2,0044e-002	1,9856e-003	8,463e-003
7	20,	-6,6439e-002	-2,381e-002	-3,0924e-002	-5,2717e-003
8	23,333	-7,2675e-002	-1,6146e-002	7,3445e-003	1,4708e-003
9	26,667	-7,6335e-002	-1,0393e-002	5,2741e-003	-1,4337e-002
10	30,	-8,7785e-002	-6,0381e-003	-2,9123e-004	-8,4807e-003
11	33,333	-0,10137	-2,101e-003	7,7445e-003	-2,1964e-002
12	36,667	-0,13013	8,9169e-003	-1,08e-002	-1,2954e-002
13	40,	-0,11487	5,2391e-002	-3,696e-003	-1,0901e-002
14	43,333	-4,7344e-002	1,9063e-002	-4,9683e-003	-1,1088e-002
15	46,667	1,9002e-003	3,2187e-002	9,6904e-003	1,5601e-002
16	50,	-0,15716	4,8284e-003	3,1522e-002	-4,2857e-002
17	53,333	-5,342e-002	6,3592e-002	-1,5964e-002	2,3436e-002
18	56,667	-3,2297e-002	0,10281	2,9038e-002	-1,6247e-002
19	60,	2,2608e-002	8,8518e-002	3,2714e-002	-2,465e-002
20	63,333	4,0532e-002	7,1713e-002	3,2711e-002	-3,6059e-002
21	66,667	5,4113e-002	4,3551e-002	4,0715e-002	-6,8354e-002
22	70,	5,8248e-002	2,1572e-002	2,2344e-002	-5,1346e-002

Fig. 2.9: Decomposition of the force vector into real and imaginary parts in the direction of the x and y axes when 2D magnetic analysis and FFT is performed.

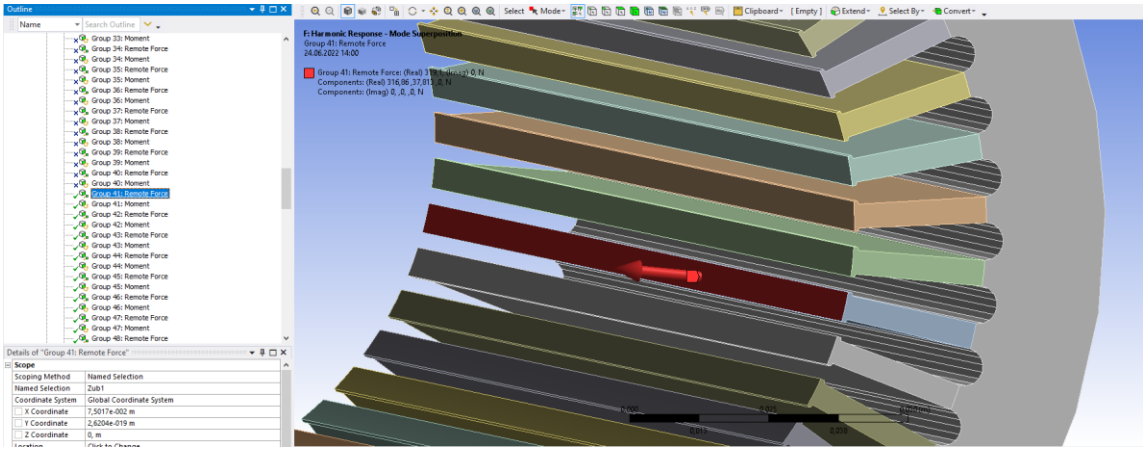


Fig. 2.10: Assignment of a lumped force vector to the particular tooth, respectively to the face on the tooth normal to the air gap.

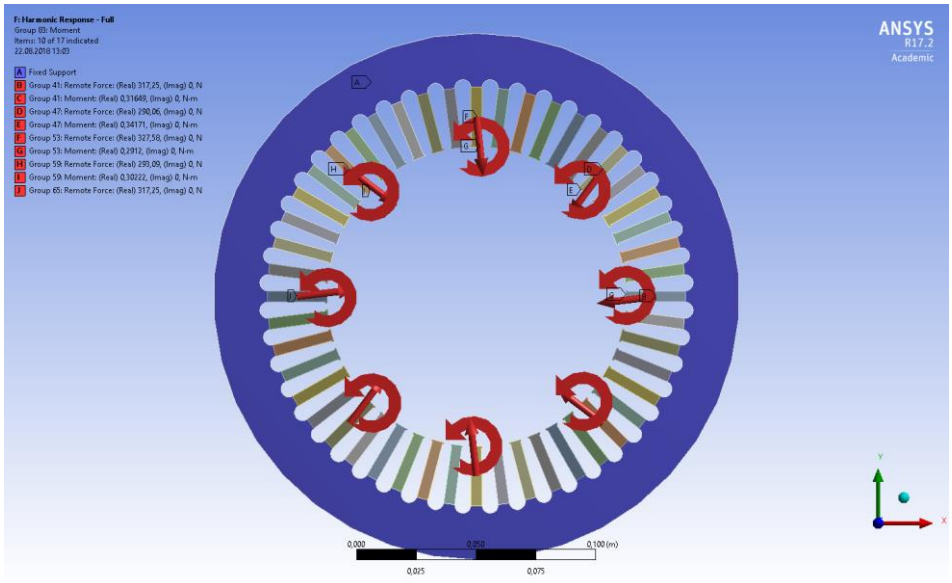


Fig. 2.11: Magnetic forces and moments assigned to the respective teeth in a harmonic response – red arrows. Fixed face of the stator yoke is highlighted in blue.

Vibration frequency response calculated in ANSYS Mechanical is given by the amplitude and phase of for displacement, velocity or acceleration in the *x*, *y* and *z* axes – graphs with frequency spectra of individual quantities can be plotted. The frequency response can be evaluated in a certain volume, face or vertex (point) of the geometry or in a mesh node (not in an element) as the minimum, maximum or mean values of individual quantities. The most appropriate solution is to evaluate the response at a vertex (point) in the geometry or a mesh node, as the result is not distorted by calculating the mean value of the vibrations for the whole area or volume. That corresponds to vibration measurement, where the accelerometer is attached to a small area on the surface of the measured object (in the case of electrical machines most often the frame or bearing bracket).

For the analysis purposes, it is therefore appropriate to evaluate the frequency response in the mesh nodes on the outer circumference of the stator (see Fig. 2.12) or frame, located as close as possible to the axes of the selected teeth. Number of teeth (or "measuring spots") should be chosen with regard to the number of poles of the analyzed machine, in order to avoid response evaluation in the node of a natural shape (mode). An example for a four-pole machine with 48 stator slots is shown in Fig. 2.13.

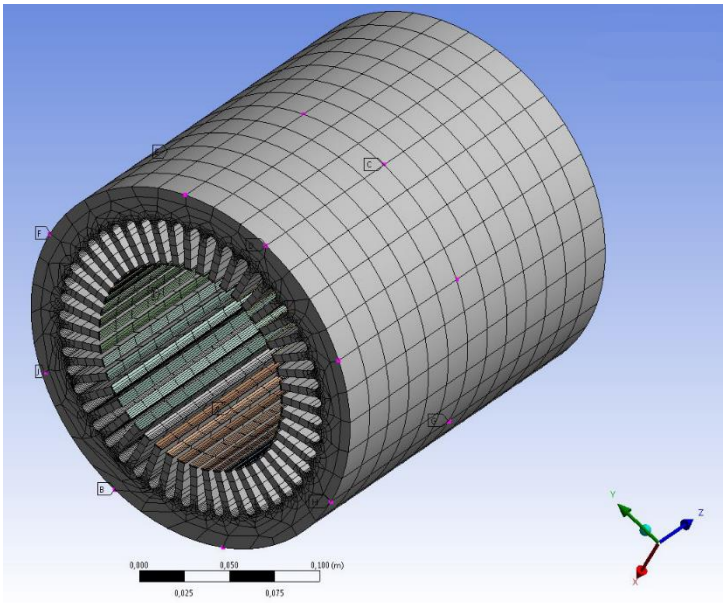


Fig. 2.12: Mesh nodes at the end and in the middle of the stator stack in which the frequency response is detected (marked in pink).

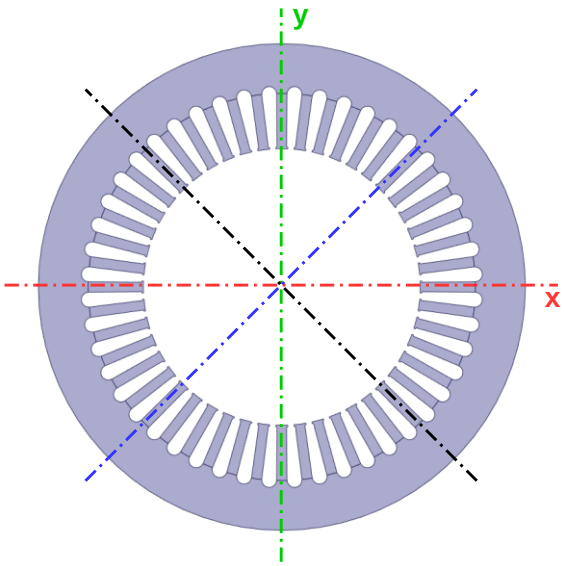


Fig. 2.13: Tooth axes of a four-pole, 48 stator slot machine in which the response is determined.

Conclusions

Supporting materials for computer tutorials and individual work of students in the course Selected Parts of Electrical Machines contain theoretical basis for modeling internally excited vibrations of electrical machines due to magnetic forces. Furthermore, the process of modeling vibrations – harmonic response – due to magnetic forces via ANSYS software package based on the finite element method utilization is described in the document. As supplementary, or necessary in some cases, modal analysis and static structural analysis are described.

The main advantage of harmonic response analysis using a reduced model is the solution speed. However, the results are rather informative in the sense that some of the excitation frequencies do not coincide with the natural frequency of the assembly and therefore the response to the whole spectrum of excitation forces is unknown. In the analysis settings, it is possible to enter other frequencies to be solved into the setup table, but this prolongs solution speed as mentioned above. The result of this analysis is certainly sufficient in many applications or in the initial analyzes during the machine design. However, for a more accurate result of the harmonic response to the whole spectrum of excitation forces, it is more appropriate to perform the analysis utilizing the full model.

References

[1] D. Mayer, *Aplikovaný elektromagnetizmus*. České Budějovice: KOPP, 2012.

[2] J. F. Gieras, C. Wang, and J. C. Lai, *Noise of Polyphase Electric Motors*. Boca Raton: CRC/Taylor & Francis, 2006.

[3] "ANSYS® Academic Research Mechanical and CFD, Release 17.2." ANSYS, Inc.

[4] J. Šobra, "Dynamické chování konstrukčních částí elektrických strojů," Západočeská univerzita v Plzni, Fakulta elektrotechnická, Plzeň, 2018.

[5] M. Promberger, *Použití matic a tensorů v theoretické elektrotechnice*. Praha: SNTL, 1955.

[6] V. Zeman and Z. Hlaváč, *Kmitání mechanických soustav*, 2. Plzeň: Západočeská univerzita, 2004.

[7] "ANSYS® Electromagnetics Suite, Release 17.2, Help System, Maxwell to ANSYS Harmonic Force Coupling," 2016.

[8] "ANSYS® Mechanical, Release 17.2, Help System, Importing Data into a Harmonic Analysis," 2016.

[9] L. Bouzek, "Elektromagnetické pole, síly, chvění a hluk v elektrických strojích," Západočeská univerzita v Plzni, 2014.