ELECTRONICS

Electric Machine Noise and Vibration

Noise in electric machines can come from many sources and has direct effects on the perceived quality of a product. As one noise source is reduced, other sources become increasingly important. The design of a high-performance electric machine with good sound characteristics requires a holistic engineering approach involving both the magnetic design of the motor and structural/acoustic design of the housing and supports.

This design process can be analyzed with simulation, which can include simultaneous optimization of motor and housing parameters to achieve noise goals.

Noise and vibration in electric machines can cause both real and perceived failures in the performance and quality of a system driven by that electric machine. The noise can have structural (gear-slip, bearings, or other moving surfaces touching), aero-acoustic, load-induced or electromagnetic sources. Each of these physical mechanisms can interact and be fed back through the structural vibrations of the physical system, so it becomes difficult to separate noise sources in a meaningful way. Many structural vibrations can be reduced by additional supports and damping of the system at the expense of increased cost, size and weight. To provide a high-performance system within design constraints, we must engineer the structure and the system to produce the required performance while reducing vibration and subsequent noise. This can all be optimized simultaneously within the Ansys simulation platform.

There are several key physical aspects to bringing together the different noise sources with simulation. The two main aspects are representing the input forces with high fidelity, and then capturing the physical response of the system with high confidence.

Ansys Maxwell and Ansys Mechanical are used to simulate motor noise, vibration and harshness (NVH) accurately. The input forces are calculated from transient simulation data on the motor stator teeth and structural gear interfaces. The physical response of the structural system is performed in several ways (e.g, static pre-stress, modal free-response, harmonic forced-response), culminating in a forced harmonic analysis: a frequency-domain model in which the absolute frequency-dependent forces are used to calculate the absolute vibration magnitudes.

Ansys simulates electric machine performance within our magnetic FEA simulation software. In addition to the rated performance data (i.e., torque, loss, efficiency) that are normally available from the magnetic FEA simulation, we can choose a software option to automatically calculate the stator tooth forces. The only additional requirements are to further control the mesh on the teeth, and to set the transient time steps to produce the correct frequency data. The force calculation, the subsequent time-to-frequency transformation, and any periodic replication of the data is automatically done by the Ansys solver. There are two additional options for the force calculation: first, the rotor speed can be swept using a different excitation for each speed, and second, the force density on the entire stator object can be mapped in the frequency domain to the structural model. These new options work in the existing workflow but expand the capabilities to all machine types and produce waterfall diagrams of noise frequencies at different rotor speeds and during run-up. All these options are available for both 2D and 3D magnetic models, and for both full geometry or a fractional number of poles. The force data is naturally transferred to the structural model for the vibration response in the same way as the full 3D vibration response for all of these cases.

The structural response starts with a characterization of the pre-stress and modal response of the motor, housing, gear-drive and fixturing. The static pre-stress analysis can take into account the bolt pre-tension, gravity and static forces applied, which have an effect on the stiffness and frequency-response. A free-vibration analysis (modal simulation) provides the natural vibration frequencies and mode-shapes of the entire structure and can be used as modal-superposition inputs to the forced harmonic analysis. The forced harmonic analysis uses the frequency-domain input forces directly from the magnetic simulation, as well as other models for CFD pressures, gear and bearing forces. The magnetic forces are represented either as object-based forces per tooth, or directly as a force-density that is mapped on the entire stator surface, both in terms of real and imaginary components of absolute force magnitude. The fundamental result of the forced harmonic analysis is the absolute vibration displacement magnitudes (as in a frequency-based accelerometer measurement); these magnitudes can be used for many purposes related to vibration analyses, such as fatigue, acoustic, Equivalent Radiated Power (ERP) and optimization analyses. The fundamental benefit of this workflow is a single platform containing all noise sources with many choices of analysis options and automated data transfer, which enables seamless optimization.

There are currently two possibilities for using the forced harmonic simulation data to predict acoustic performance: through direct simulation of the acoustic domain or with the Equivalent Radiated Power (ERP). The acoustic simulation can be efficiently implemented by sequentially mapping the surface velocities of the structure to an acoustic simulation of the air. This model allows the air pressure to propagate as sound waves and produces an abundance of near-field and far-field sound pressure data vs. frequency, exactly as would be measured by microphones in an acoustic test. The ERP (or decibel ERPL) calculations are approximate, based on only the surface velocities of the structure, and represent the radiated noise based only on these surface velocities. The ERP is particularly well suited for sweeping the motor speed (RPM) to produce waterfall diagrams representing total radiated sound versus frequency and RPM.

The optimization process is facilitated by any parameters defined in the simulation model within Ansys Workbench. Workbench is the simulation platform tool that facilitates passing data between the different solvers, updating parameters for each solver, and connecting to optimization utilities such as Ansys optiSLang. optiSLang is particularly well suited for such a complicated multi-objective multiphysics analysis because it does a good job of creating a response surface with many input parameters; it efficiently distributes these simulations over many computer resources on a variety of scheduled environments. By combining all noise sources and optimizing with parameters in each solver, the simulation engineer can produce the best possible design instead of just a better design.

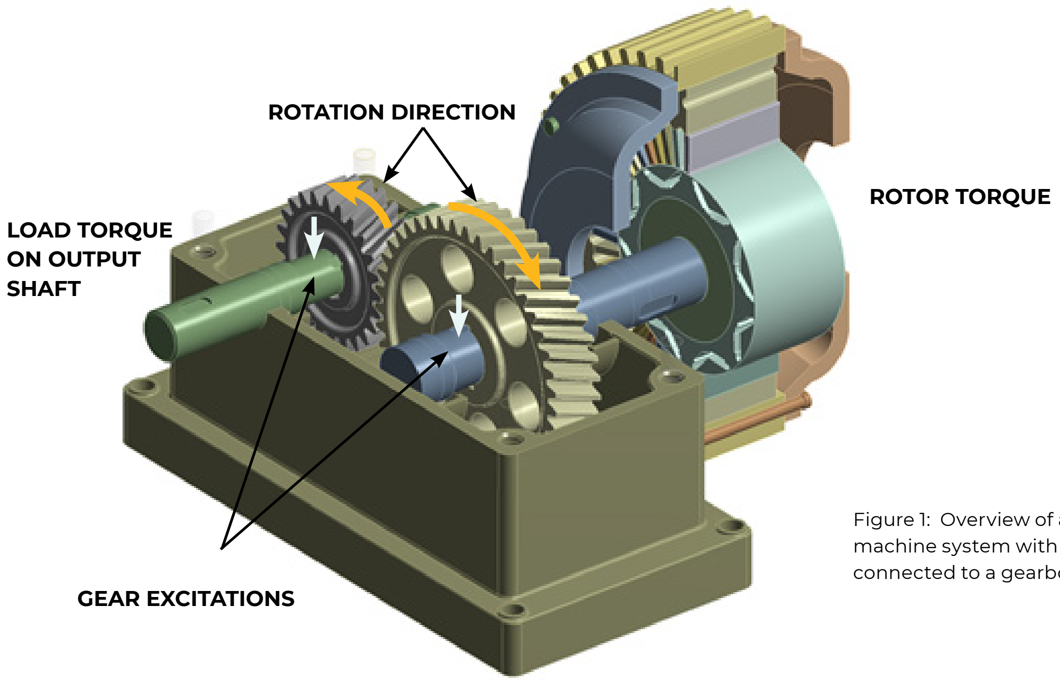

Figure 1: Overview of an electric machine system with the motor shaft connected to a gearbox.

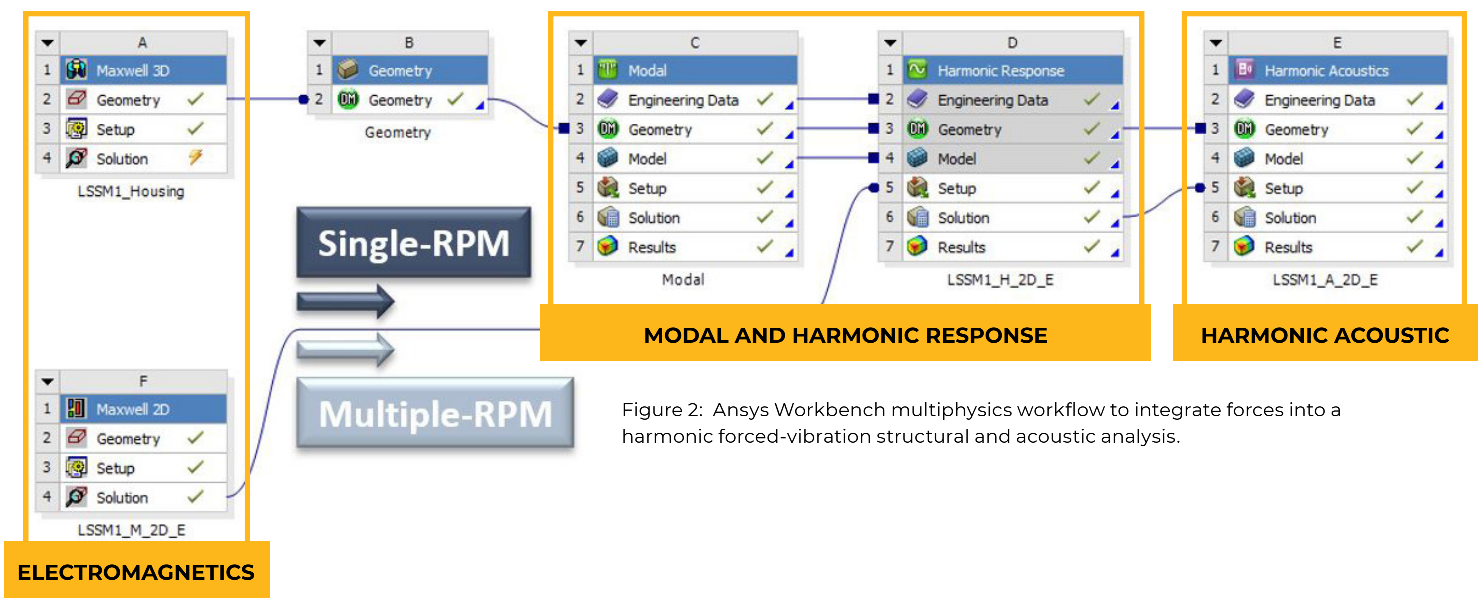

Figure 2: Ansys Workbench multiphysics workflow to integrate forces into a harmonic forced-vibration structural and acoustic analysis.

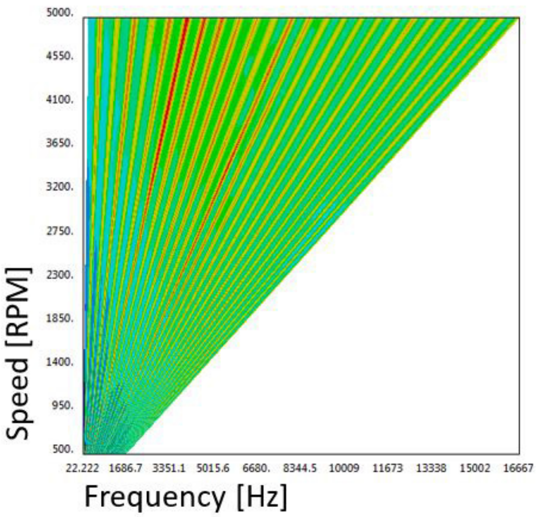

Figure 3: Equivalent Radiated Power (ERP) waterfall diagram showing ERP versus speed and frequency.

/ Summary

Using Ansys solvers, you can achieve high accuracy simulation results for vibration, noise and harshness (NVH) in electric machine systems.

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com