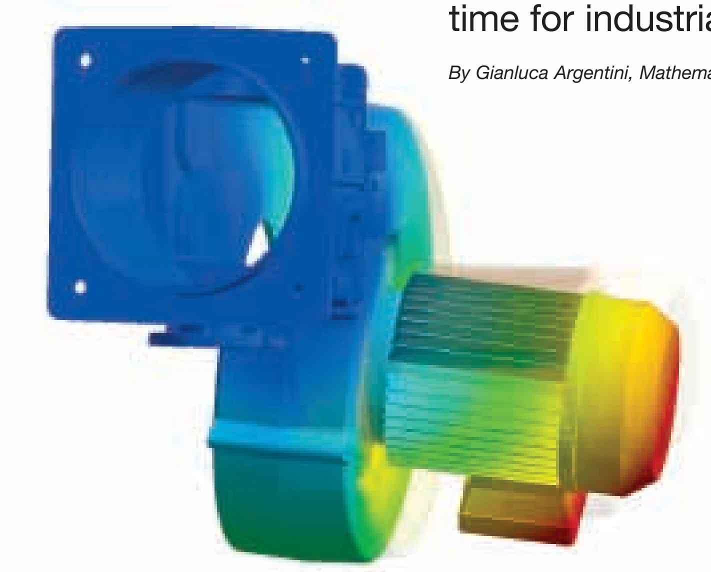
Predicting Vibrations in High Power Burners

Engineering simulation reduces development time for industrial burners by five months.

By Gianluca Argentini, Mathematical Modeler, R&D Department, Riello Burners, Legnago, Italy



Displacement at the first natural vibrational mode for the original Riello burner configuration

Industrial gas and oil burners are used for steam production in industrial processes and electric power generation plants. Reducing vibrations in these systems can increase component longevity and reduce maintenance on bolts and flanges. A gas or oil burner is comprised of an intake system that draws air into the burner and an ignition area where fuel is introduced, mixed with the intake air and ignited. In the air intake system, an electric motor drives an impeller, which pulls in air and propels it along the volute-shaped housing and into the ignition area. The ignition area contains a sleeve duct that encases a combustion head and attaches the entire burner to the associated combustion tube, or boiler. An industrial burner that is capable of producing more than five megawatts may exhibit structural vibrations due to interaction between the combustion chamber and the flame initiating from the burner itself.

Experiments on these kinds of burners at Riello Burners Combustion Research Center showed that the frequencies of the vibrations depended on the power of the flame, the dimensions of the combustion tube, and the features and design of the burner assembly itself. In order to reduce vibration in the burner system, engineers performed a detailed study of their structural properties, including natural modes of vibration and dependence on the geometry and the materials used in the system's components.

The usual mathematical description for problems of this type is based on the structural mass [M], stiffness [K] and damping [C] matrices of the system, which are related to the displacements of the structure {*U(t)*} by the standard set of differential equations of motion:

$$[M]\{\ddot{U}\} = -[C]\{\dot{U}\} - [K]\{U\}$$

While this equation can be manually solved for simple, linear, discrete systems, a numerical approach like finite element analysis is necessary for complex geometries such as in a burner-engine system. Engineers and researchers at Riello Burners have found that modal analysis using ANSYS Mechanical software within the ANSYS Workbench environment is extremely useful for a rigorous numerical treatment of alternate designs.

The engineering team applied a fixed constraint at the surface where the sleeve duct was anchored to the combustion chamber. For each body with a large mass, the engineers specified the appropriate physical constants, such as Young's modulus and Poisson's ratio, using the engineering data section of the ANSYS Workbench interface. When necessary, the team modified these values using the material property form, which allows users to specify suitable data in the physical characteristics field.

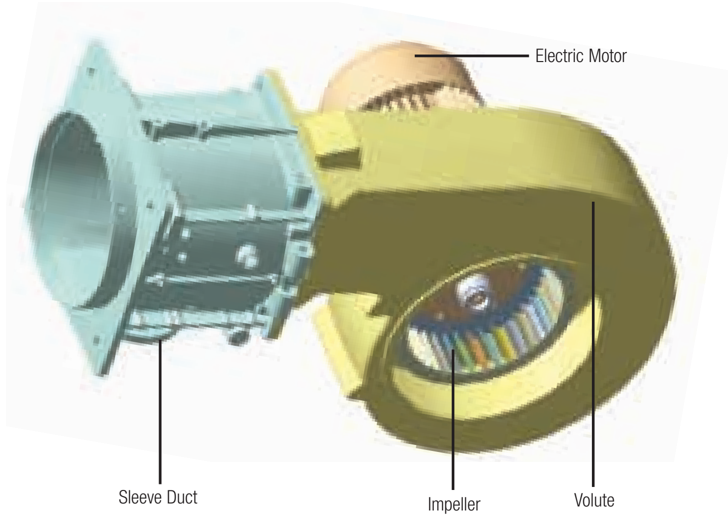
An initial simulation using the original burner design provided the same results as experiments with regard to the values of vibrational frequencies. In particular, as computed by the software, a value of 49 hertz demonstrated the need to accurately balance the electric engine to avoid whirling effects caused by a rotational speed of only 48 revolutions per second. Also, the smallest computed value of 29 hertz is almost equal to the natural frequency of the combustion tube (boiler). When designing and engineering a high power burner, the geometric and physical properties of the boiler — including the dimensions of the tube, the physical characteristics of the materials and the water mass flow — must be considered very carefully so that the boiler's vibrations have a natural frequency that is not close to that of the burner in order to avoid resonance problems.

To understand and include the influence of combustion on the burner's mechanical vibrations, the engineering team considered and improved a mathematical model in which the flame's perturbation and boiler's geometry were combined into a unique Fourier series with frequencies expressed by:

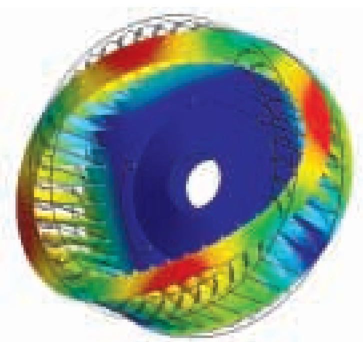
$$f_n = m\frac{c}{2L}$$

where *c* was the speed of sound in the tube's local environment (400 m/s), *L* the tube's length (8 m) and *m* the natural frequency of concern. Using this formula, f_1 was calculated to be 25 hertz, a value very close to the smallest vibrational frequency of 29 hertz computed by the software.

Using the harmonic response analysis module in the ANSYS Workbench platform, Riello engineers performed computations using sinusoidal loads with frequencies in the range of 5 to 100 hertz acting on the surface between sleeve duct and combustion tube. This module allowed researchers to set values for damping coefficients to improve the accuracy of the simulation. The simulation results were confirmed by the data obtained from experiments for both



The geometry of a burner assembly: Air is taken in through the impeller and directed into the sleeve duct, which houses a combustion head (which initiates the flame).



Impeller four-nodal diameter mode associated with its first natural frequency; this value, 141 hertz, is close to other vibrational frequencies of the global system.

Equivalent stress for impeller four-nodal diameter mode at 141 hertz; at the edge of the hub's central bearing surface, the local stress values are high and can be close to the creep coefficient of the material.

ANSYS Software for Linear Dynamic Vibration Simulation

The study of structural dynamics is critical for understanding and evaluating the performance of any product. It is also essential for solving noise and vibration problems with existing structures. ANSYS mechanical solutions, including ANSYS Structural, ANSYS Mechanical and ANSYS Professional software, provide all the necessary tools for such analyses.

Usually, vibration analysis begins with a modal analysis that estimates the natural vibration frequencies of a given structure. The frequencies of the structure can be determined from an unloaded state or from the loaded structure, as loads may shift the frequencies. Depending on the environment of the structure, several advanced types of analyses can be performed: harmonic, spectrum, random vibration or transient dynamic analysis.

Harmonic response analysis is a technique used to determine the steady-state response of a linear structure to loads that vary sinusoidally (harmonically) with time. The structure's response is calculated at several frequencies and a graph of a response quantity (usually displacements) versus frequency is generated. Peak responses are then identified on the graph and stresses reviewed at those peak frequencies.

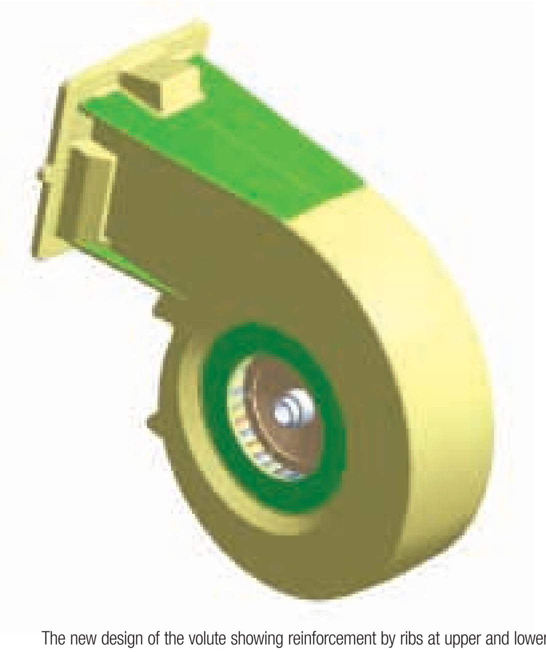
A spectrum analysis is one in which the results of a modal analysis are used with a known spectrum to calculate displacements and stresses in the model. It is mainly used instead of a time-history analysis to determine the response of structures to random or time-dependent loading conditions such as earthquakes, wind loads, ocean wave loads, jet engine thrust and rocket motor vibration.

A random vibration analysis is similar to a spectrum analysis technique but is based on probability and statistics. It is used to analyze loads that produce random time histories, such as acceleration loads during a rocket launch, that can be represented by a power spectrum density during each event.

Finally, a transient dynamic analysis is used to determine the response of a system under a given load variation over time.

ANSYS mechanical solutions allow the use of any of these techniques with various methods: direct analysis where the full matrices are assembled, mode superposition that reuses the results of a modal analysis, or reduced methods that condense the problem to a smaller set of degrees of freedom. The last two options help reduce the computational time. Another technique to reduce the computational time is the Component Mode Synthesis (CMS), which is used in reducing the size of the problem when large and complex assemblies are modeled.

– Pierre Thieffry, Product Manager, ANSYS, Inc.



The new design of the volute showing reinforcement by ribs at upper and lower parallel surfaces and by new material at the engine flange (modifications in green)

vibrational frequencies and displacement values. The peak displacement value for the mechanical system was computed at 30 hertz, confirming that the burner shows a structural resonance due to the first harmonic Fourier component of the flame.

To eliminate the structurally dangerous lower vibrational frequencies, the team performed modal analysis for a set of possible design modifications to the structure of the components. After iterating through four new designs, the engineers reached a virtual configuration where the first natural frequency is sufficiently high that displacement of the global structure does not occur during normal operation. In the end, by using the ANSYS Workbench platform, Riello Burners minimized the high cost for construction and testing of intermediate prototypes, reducing the time to develop an optimized model of the burner by approximately five months. ■

References

[1] Den Hartog, J.P., *Mechanical Vibrations*, Dover: New York, 1985.

[2] Strogatz, S.H., Abrams, D.M., McRobie, A., Eckhardt, B., Ott, E., "Theoretical Mechanics: Crowd Synchrony on the Millennium Bridge," *Nature* 2005, Vol. 438, pp. 43–44.

[3] Doria, A., "A Simple Method for the Analysis of Deep Cavity and Long Neck Acoustic Resonators," *Journal of Sound and Vibration* 2000, Vol. 232 (4), pp. 823–833.