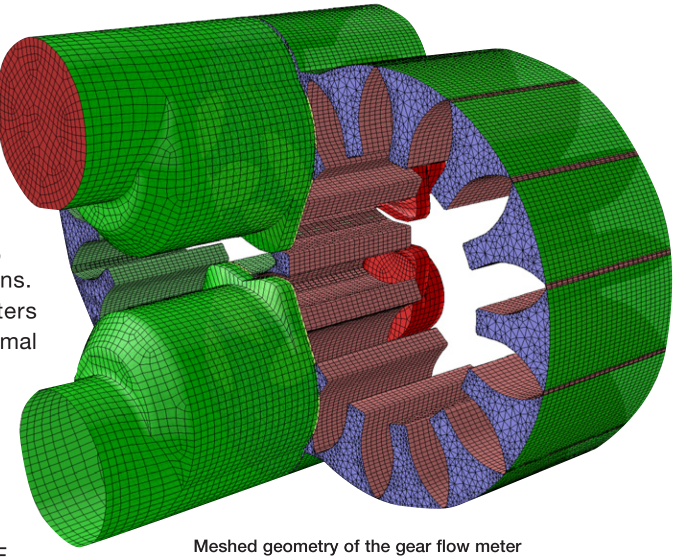
Gearing Up for Better Measurement

Using engineering simulation, a German flow meter manufacturer reduces vibration, noise and pressure loss to develop more reliable instruments.

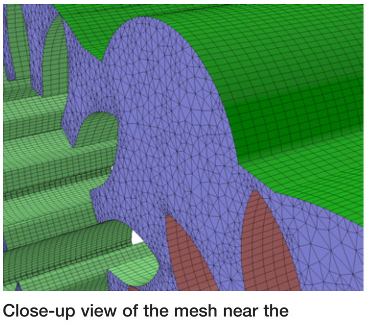
By Axel Vedder, Technical Director, VSE Volumentechnik GmbH, Neuenrade, Germany
Mourad Lotfey, CFD Engineer, ANSYS, Inc., Otterfing, Germany

Flow meters are measurement instruments used in airplanes, ships, medical devices, automobiles and homes as well as power-generation and process plants. These devices help companies around the world ensure that their systems and processes are running smoothly by measuring fluid flow characteristics such as volumetric flow rates, pressures, temperatures and material concentrations. Among the many industrial applications, flow meters indicate whether hydraulics are operating under optimal conditions, whether pipelines are running at maximum capacity, and whether additives are being properly injected into primary fluids. Such measurement systems must be robust and reliable, and they need to provide precise information.



Meshed geometry of the gear flow meter

While there are many ways to measure fluid flow, flow meters from German manufacturer VSE Volumentechnik GmbH (VSE) are based on the positive-displacement principle. The company's flow meters include a pair of gears that are continually rotated by the fluid flow. As these gears rotate, they produce an output frequency that corresponds to a specific flow rate. Gear-based flow meters represent a precise and efficient system for measuring volume flow rate of fluids but have traditionally included some drawbacks including noise, vibration and pressure loss created by the constantly spinning gears, especially with highly viscous material. This can cause discomfort for personnel working nearby as well as excessive wear on flow meter gears and bearings. The ability to determine the exact reasons these problems occur is quite limited when using traditional methods. By using fluid flow simulation software from ANSYS, engineers are able to model the fluid flow inside flow meters and are better able to understand the cause and to develop solutions for performance issues.



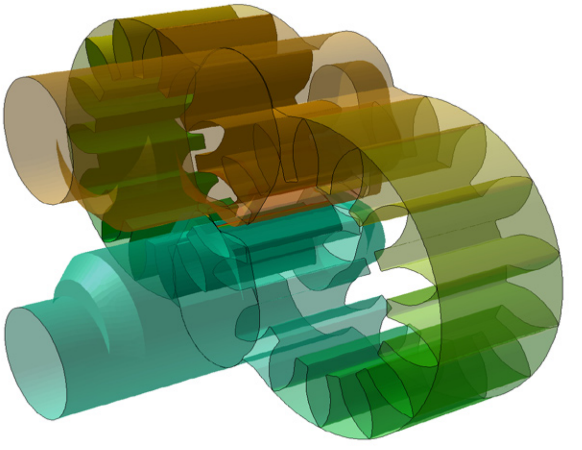
Close-up view of the mesh near the gear segments.

To perform the simulation, VSE engineers began with geometry from a CAD file. After importing geometry and cleaning up the flow domain appropriately, they created a numerical grid of hexahedrons and wedges for study with fluid dynamics analysis. Using ANSYS FLUENT software, the engineering team conducted a transient fluid simulation of the gear flow meter. With the ANSYS FLUENT dynamic mesh method, the initial mesh was automatically adjusted and regenerated during the simulation process as the volume mesh was deformed geometrically by the rotational movement of the gears. The software performed fluid dynamics calculations of the rotating flow volumes between the gears after each incremental angular movement until the pair of gears completed a full rotation.

VSE simulated the gear flow meter in three dimensions with the fluid defined as incompressible, laminar, isothermal and Newtonian. Since the real flow conditions are very complicated, the simulation contained several simplifications to reduce the computational expense. In VSE's simulation, the frictional forces between the gear pair and the axis as well as gear inertia effects were ignored. Additionally, the friction and leakage between the side faces of the gears and the adjoining housing were disregarded.

The simulation provided access to unique detailed 3-D flow information inside the flow meter, including dynamic distribution of pressure, velocities, residence time and viscous heating of the fluid. Recirculation regions could be identified inside the gear flow meter, by using 3-D flow path lines. ANSYS FLUENT technology also calculated the spatial distribution of velocity, the shear stress gradients, and the wall friction forces and moments.

VSE subsequently calculated the torque of the gears as the cross product of the surface forces (pressure and shear) and the distance from the gear segment to the rotational axis. By analyzing the contribution of different gear surface segments to the total torque, the engineers identified the segment that produced the greatest torque and modified it to minimize the pressure loss (and consequently to reduce energy consumption of the gear flow meter). The dynamic mechanism of cooling, mixing, pressure loss and degasification could then be studied in detail.

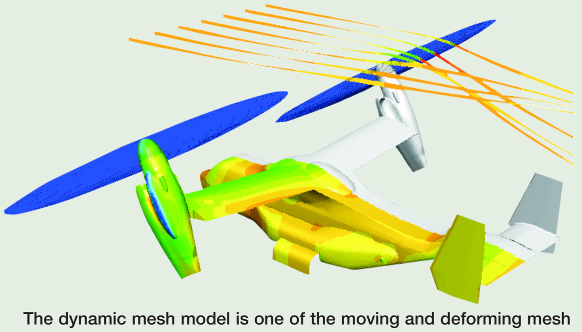


Contours of pressure distribution on the model walls

Using fluid flow simulation, the cause of the noise development was localized, and VSE was able to develop proposals for reducing vibration, noise and pressure loss. VSE engineers intend to optimize the operating characteristics further using software from ANSYS, which they have determined to be cost efficient, easily integrated into the design system and helpful in accelerating their design cycle. ■

Dynamic Mesh Model

The dynamic mesh capability in ANSYS FLUENT software meets the needs of challenging applications that involve unsteady moving geometry. In such cases, the dynamic mesh model is used to move boundaries, objects or both as well as to adjust the mesh accordingly. Several different mesh rebuilding schemes, including layering, smoothing and remeshing, can be used for different moving parts within the same simulation as needed. Only the initial mesh and a description of the boundary movement are required. The process of moving and deforming the mesh according to the specified boundary motion is handled internally by the ANSYS FLUENT solver, since specific factors govern when a mesh cell should split and when two cells should merge. Available solver options that allow the software to automatically adjust the time step dynamically during the simulation can be used to accelerate the calculation and ensure a stable flow simulation. Autosave options permit intermediate files, which include the current state of the mesh and solution, to be saved at each time step for later analysis. The saved files can also be used to create animations of the moving mesh and the transient behavior of many flow parameters (for example, pressure profiles or path lines). Dynamic meshing is compatible with a host of other models available in ANSYS FLUENT software, including a suite of spray breakup and combustion models and multiphase models such as those for free surface prediction and compressible flow. Example applications in which the dynamic mesh model is necessary to capture motion include passing cars, moving flaps on an aircraft wing, moving piston in an engine cylinder, store separation from an aircraft, ship motion, and the opening and closing of valves.



The dynamic mesh model is one of the moving and deforming mesh capabilities available in ANSYS tools for fluid flow simulation. The moving mesh technology in ANSYS FLUENT software can be used to model the changing tilt of the rotorcraft's wings while changing flight modes, illustrated with this Osprey aircraft.

Chris Wolfe, Senior Product Manager, ANSYS, Inc.