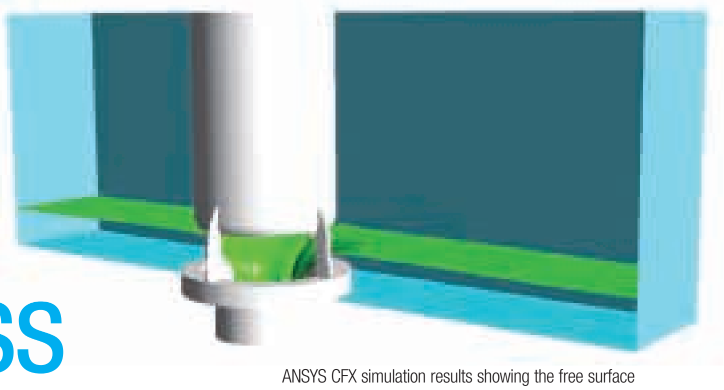
Flush with Success

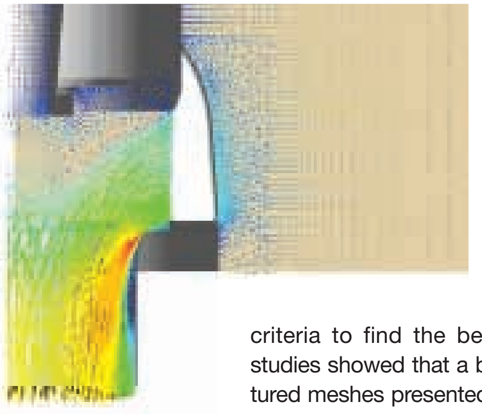


ANSYS CFX simulation results showing the free surface movement through the valve during discharge

A toilet discharge valve is optimized to reduce household water consumption and maintain performance.

By Miguel Francisco and António Gameiro Lopes, Mechanical Engineering Department, University of Coimbra, Portugal
Vítor Costa, Mechanical Engineering Department, University of Aveiro, Portugal

Environmental concerns have made reducing water consumption a requirement in many countries. Domestic water usage in toilets can be a significant contributor to overall consumption. A family of four makes an average of 20,000 toilet flushes per year, representing approximately 120,000 liters of water. To cope with the environmental demands of treating large amounts of sewage, legislation in many areas has become more restrictive with regard to water used per flush. This has led to the need to optimize the performance of toilet flushing devices.



Water velocity vector field analysis inside the discharge valve

With this in mind, engineers at the Department of Mechanical Engineering of the University of Coimbra and the University of Aveiro in Portugal helped Oliveira & Irmão S.A, a manufacturer of toilet flushing valves, to improve the performance of their products. Their collaborative effort focused on increasing the average water discharge velocity in order to optimize the washing efficiency in the toilet bowl. To achieve their goals, they used numerical simulation coupled with experimental validation and performed several shape studies aimed at achieving higher instantaneous flow rates.

As a first effort in the optimization process, the researchers employed steady-state computational fluid dynamics (CFD) simulations in order to estimate the pressure drop across the discharge valve. The team used ANSYS CFX software to perform their CFD analyses. In order to compute the volumetric flow rate, they analyzed the geometries from the initial simulations — which had exhibited lower pressure drops and, therefore, higher flow velocity — with full three-dimensional, unsteady, gravity-driven, free surface simulation models.

The challenge of this project was to determine the best modeling approach for the physical problem. For a reliable numerical estimate, a good definition of the free surface was crucial. The team carefully tested parameters such as boundary conditions, meshing types, modeling schemes and convergence criteria to find the best solution. Grid dependence studies showed that a blend of structured and unstructured meshes presented the most accurate results. The engineering team employed an unstructured mesh near the valve, but in the rest of the cistern where a detailed description of the water free surface is crucial, they adopted a structured mesh. They connected the two mesh types in regions of low-velocity gradients using a generalized grid interface.

The transient simulations used the compressive discretization scheme available in ANSYS CFX software for the volume fraction, so that the free surface would be resolved as sharply as possible as the simulation progressed. The k-ω–based shear stress transport (SST) model was chosen to capture the effects of turbulence. Experimental validation confirmed the fidelity of the numerical model, with relative errors below 5 percent when compared with experimental data. As a result of these efforts, the analysis team determined that ANSYS CFX software was a reliable tool for the simulation of this type of gravity-driven, free surface flow. ■

This work was funded by Fundação para a Ciência e a Tecnologia, Portugal, through the Research Project POCT/EME/46836/2002.