OPENING THE VALVE TO IMPROVED DESIGN

Metso linear globe valve

Metso Flow Control engineers require several weeks to build and test a new valve. Using ANSYS AIM software developed for design engineers, Metso engineers can digitally explore the performance of a new design in a few hours. The ability to evaluate many more designs in less time makes it possible to substantially increase valve performance and get new valves to market sooner.

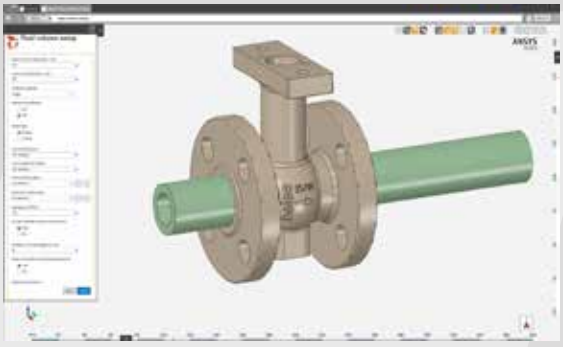
By **Tommi Bergström**,
Senior Research Engineer,
Metso Flow Control,
Vantaa, Finland

Valves are a vital part of almost every operation that includes fluid flow. Designing valves can be quite complex because they need to operate reliably for long periods in often adverse conditions and under exacting specifications. One of the most critical aspects of valve design is to provide a target flow rate at each point in its travel as the valve's trim (the moving parts of the valve in contact with the fluid) is opened or closed. The valve capacity coefficient (C_v) is defined as the flow rate in gallons of 60 F water that will pass through a valve in one minute at a 1 pound per square inch (psi) pressure drop. Metso previously determined the C_v value for any proposed valve design by building a prototype and testing with water. However, building a prototype and performing the test took about four weeks for each design iteration, and in most cases several iterations were required to hit the target specifications.

Recently, Metso engineers worked with EDRMedeso, an ANSYS Elite channel partner, to adopt an ANSYS AIM solution that enables design engineers to reliably and accurately simulate the performance of a proposed design in a few hours without

> *"Valve performance has been improved because engineers can evaluate many more alternatives than were possible in the past."*

involving a CFD expert. Design engineers import a valve design from CAD and spend a few minutes setting up the simulation using one of the guided workflows in AIM. After running the simulation, AIM produces the data required to evaluate the performance of the valve. This includes quantitative data such as pressure drop and flow rate, as well as visual information such as streamlines, contour plots and vector fields to help the engineer understand the flow behavior in the valve. Valve performance has been improved because engineers can evaluate many more alternatives than were possible in the past, making new valves available to customers sooner.

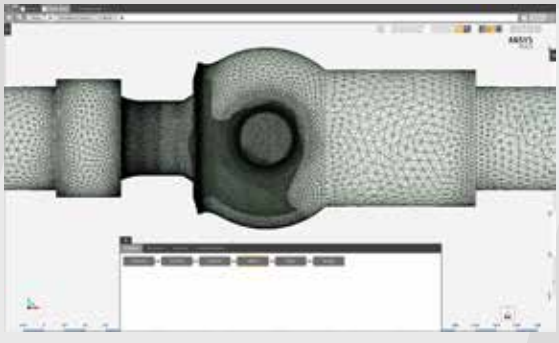


Importing CAD file containing valve geometry

WIDE RANGE OF INDUSTRIAL VALVES

Metso is a world-leading industrial company serving the mining, aggregates, recycling, oil, gas, pulp, paper and process industries. Among the Metso Flow Control business's leading product lines are control valves, on–off valves and emergency shutdown (ESD) valves. These valves are tested in a water test loop by measuring the flow while the valve is turned through its entire travel. The new valve development process often requires multiple design iterations to meet flow performance requirements.

About a decade ago, Metso began using computational fluid dynamics (CFD) to simulate valve performance. Experienced CFD engineers performed all simulation required, from design simulation for flow and pressure drop to complex analysis to mitigate noise and cavitation. Although still much faster than build-and-test, this slowed the design process as designers had to communicate design concepts to the simulation experts and then wait for them to simulate and evaluate the design.

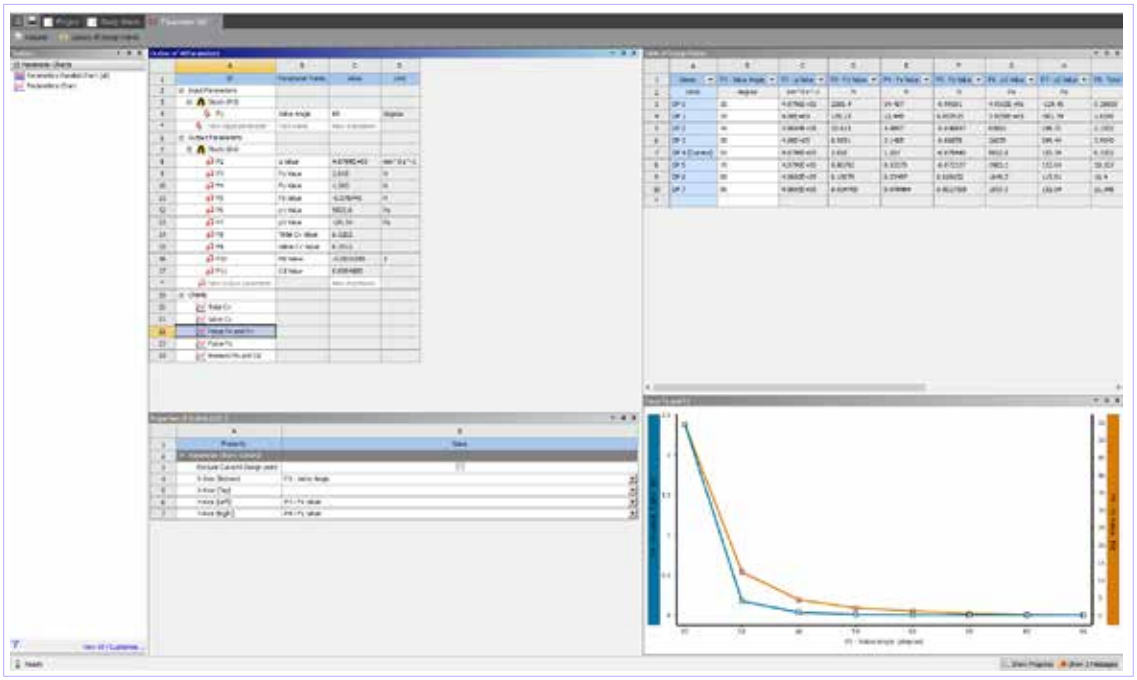


AIM solution automatically sets up flow analysis including meshing.

SOLUTION FOR DESIGN ENGINEERS

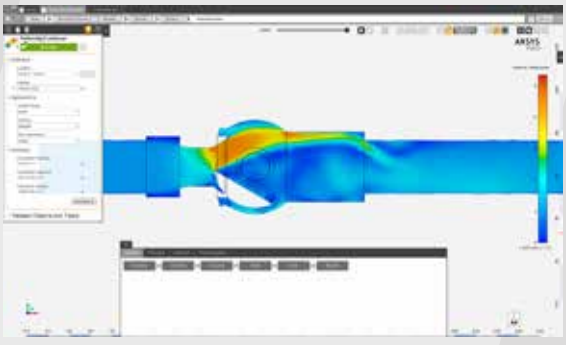
Metso addressed this challenge by implementing ANSYS AIM to enable design engineers to quickly simulate linear and rotary valves without assistance from a CFD expert. AIM was selected because its interface is intended for use by design engineers. AIM uses proven ANSYS technology to perform a wide range of multiphysics tasks, including fluid, structural, electromagnetic, vibration, thermal, durability and design optimization.

To assist the designers in performing the specific parametric simulation required and automate workflow, Metso Flow Control engineers used ANSYS ACT scripting tools to develop an application that automates the CFD process, including pre-processing, solving and post-processing. Only an hour and a half of training was needed to learn how to use the scripting tools. The design engineer begins by importing either the full or symmetric valve geometry into ANSYS AIM. Named selections are used to locate the inlet, outlet, trim and symmetry. The application guides the user in inputting the appropriate values, including pipe nominal diameter, trim diameter, feature caption, valve type, initial trim travel and maximum trim travel, etc., for a proposed design. The application determines the C_v values for a full range of trim positions. The result is a table with the C_v value at each trim position and a graph with C_v as a function



The ANSYS AIM solution generates parametric results at different valve angles.

> *"Metso CFD experts have been largely freed from routine valve design, and can instead focus on more challenging issues."*



Flow simulation results are presented to the user.

of trim position. The designer can then quickly determine the viability of each design.

BUILDING A BETTER VALVE

This ANSYS AIM solution is currently available for use by every engineer at Metso Flow Control. The Metso team can now develop multiple iterations to optimize the design of a valve in only a couple of days. The company still builds one prototype for every new valve to validate the CFD results. However, in the past, design engineers were usually limited to developing two or three design iterations because of time and cost constraints, but they can now evaluate many more design iterations to find the optimal solution in a much shorter time frame. ANSYS AIM has made it possible for the team to improve their product by, for example, reducing pressure drop or more closely matching the desired flow capacity curve. The company has also reduced time to market by decreasing the number of prototypes to one for nearly every project. Finally, the CFD experts have been largely freed from routine valve design so they can focus on more challenging issues, such as preventing cavitation and reducing noise.

ANSYS AIM: Try it Now!
ansys.com/tryAIM