Tutorial: DN400 Triple Offset Butterfly Valve Simulation

This tutorial is designed to help you complete your first valve flow simulation using Autonomous Valve CFD (AVC). It is a guided, do-along document that walks you through each stage of the workflow, from creating a new simulation to reviewing the final valve performance report.

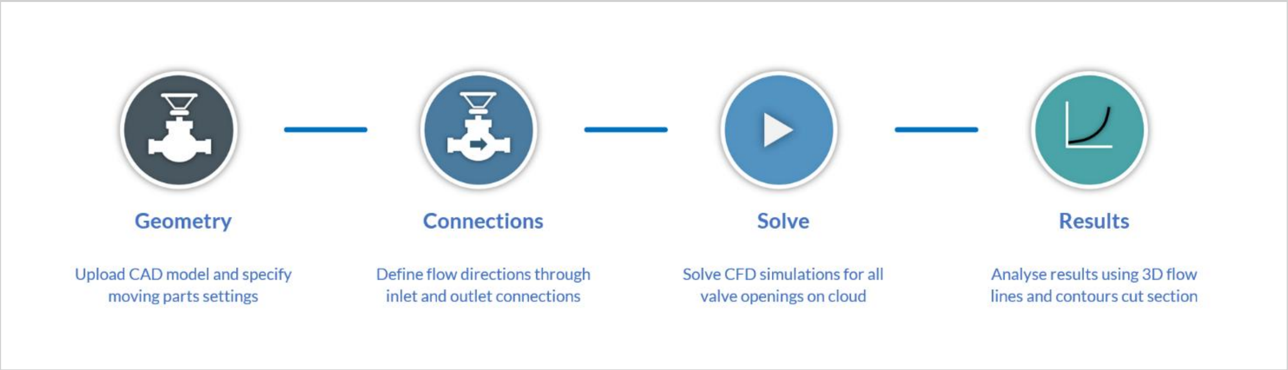
Before You Start

1. Valve CAD file: Download the CAD file "TOBV-DN400.step" and save it on your local machine.
2. AVC trial subscription: Activate your Autonomous Valve CFD trial subscription by following the steps provided in the trial activation guide.
3. Simulation credits: If you already have an active trial account, make sure you have sufficient simulation credits. This tutorial requires 8 simulation credits.

App Workflow

The AVC workflow is divided into four main stages:

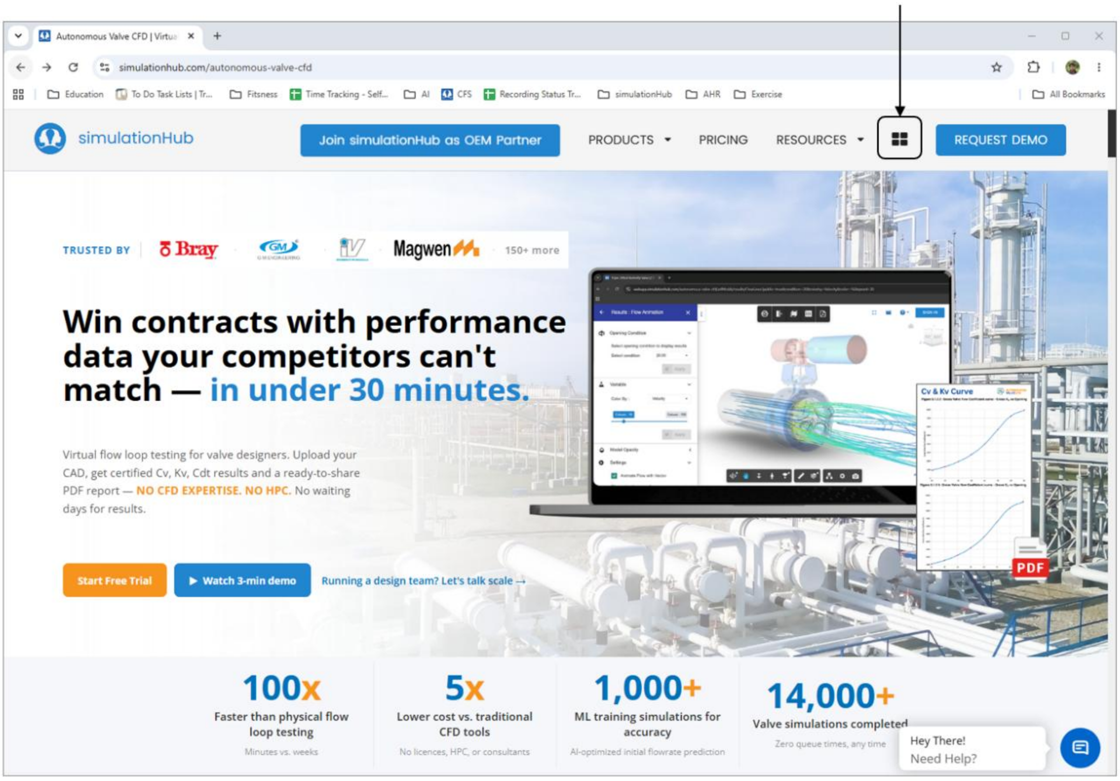
1. **Geometry**
   1.1. Upload CAD Model
   1.2. Moving Part Settings
2. **Connection**
   2.1. Define Valve Connections
3. **Simulation**
   3.1. Simulation Run & Status
4. **Results**
   4.1. Qualitative results: Flow Animation, Surface Pressure, and Contour Plots
   4.2. Performance data
   4.3. Report



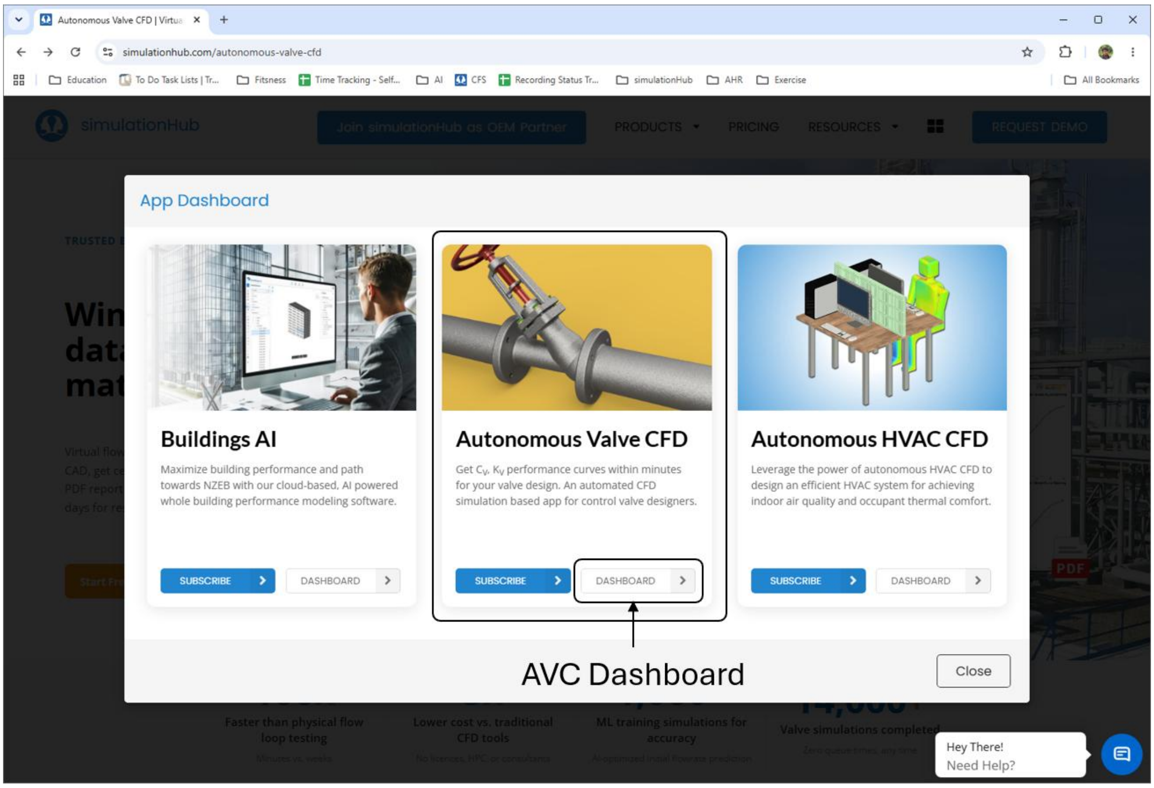
Step 1: Create a New Simulation

Visit the AVC product page at https://www.simulationhub.com/autonomous-valve-cfd and click the dashboard icon highlighted in the image below.

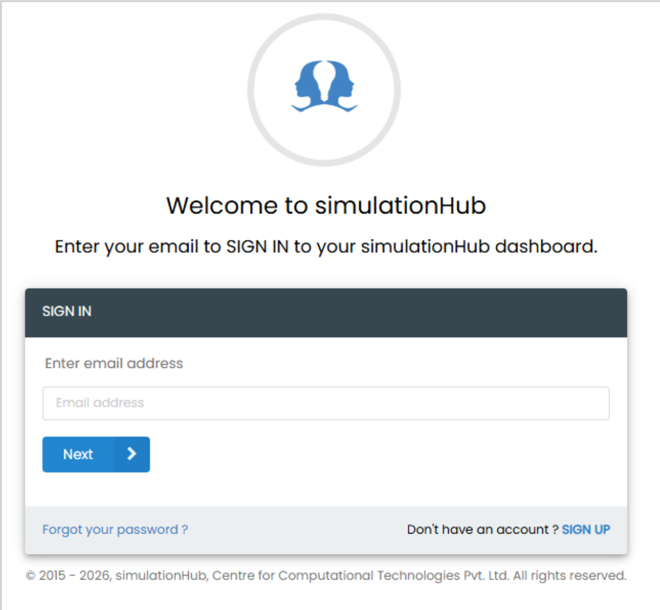
Dashboard



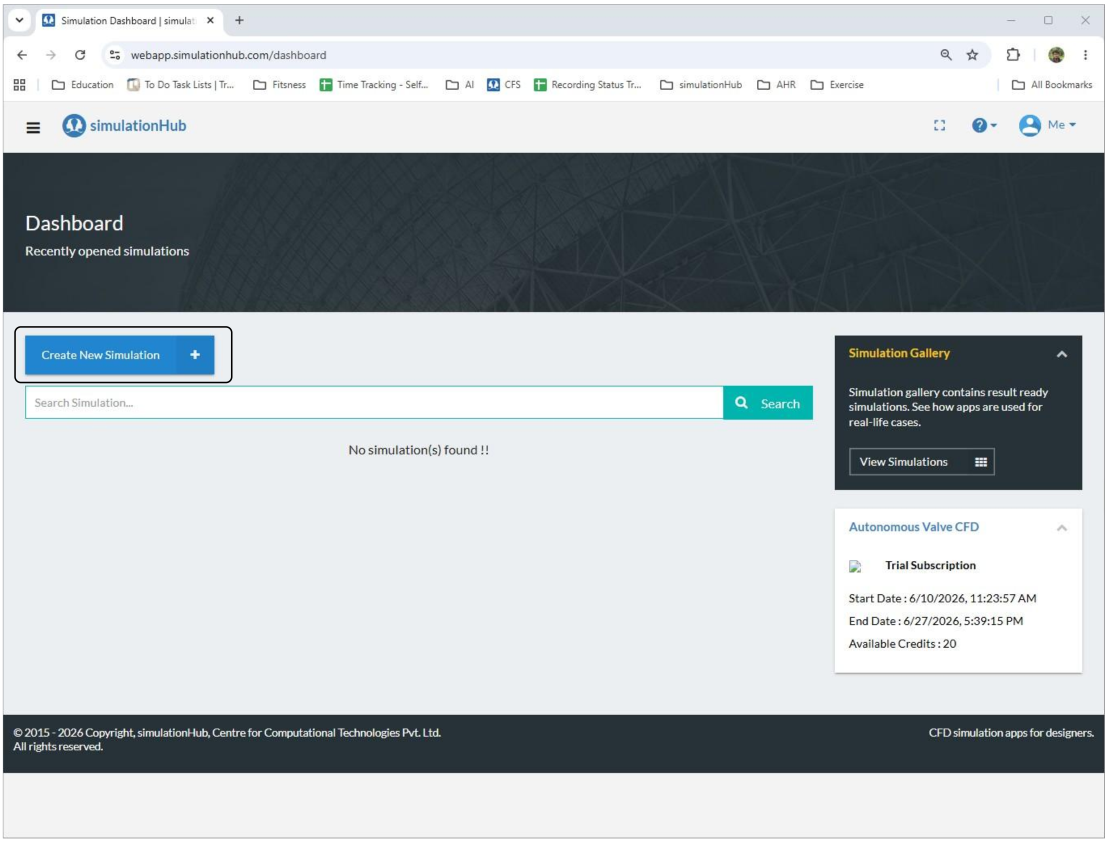
In the pop-up window, click the "**DASHBOARD**" button inside the Autonomous Valve CFD tile.



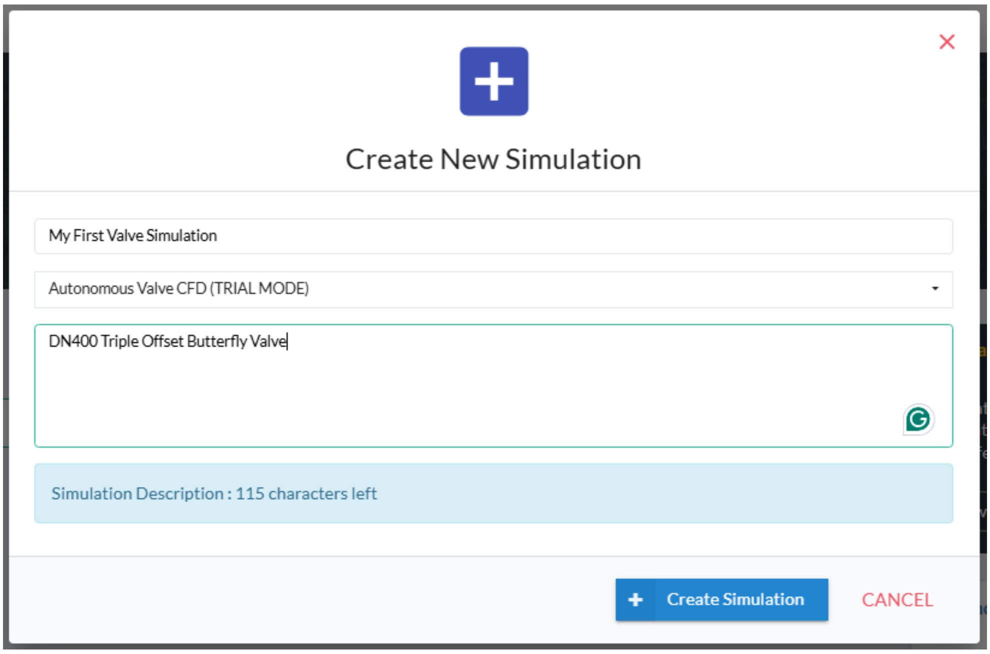
Sign in using your simulationHub login credentials. (Please check the trial activation guide if you don't have the account yet)



After signing in, your simulation dashboard will open. The dashboard is where you access all your simulations and review subscription details such as validity and available simulation credits. If you are signing in for the first time, or if you have not created any simulations yet, the dashboard may appear empty. Click '**Create New Simulation**' to create a new simulation project.

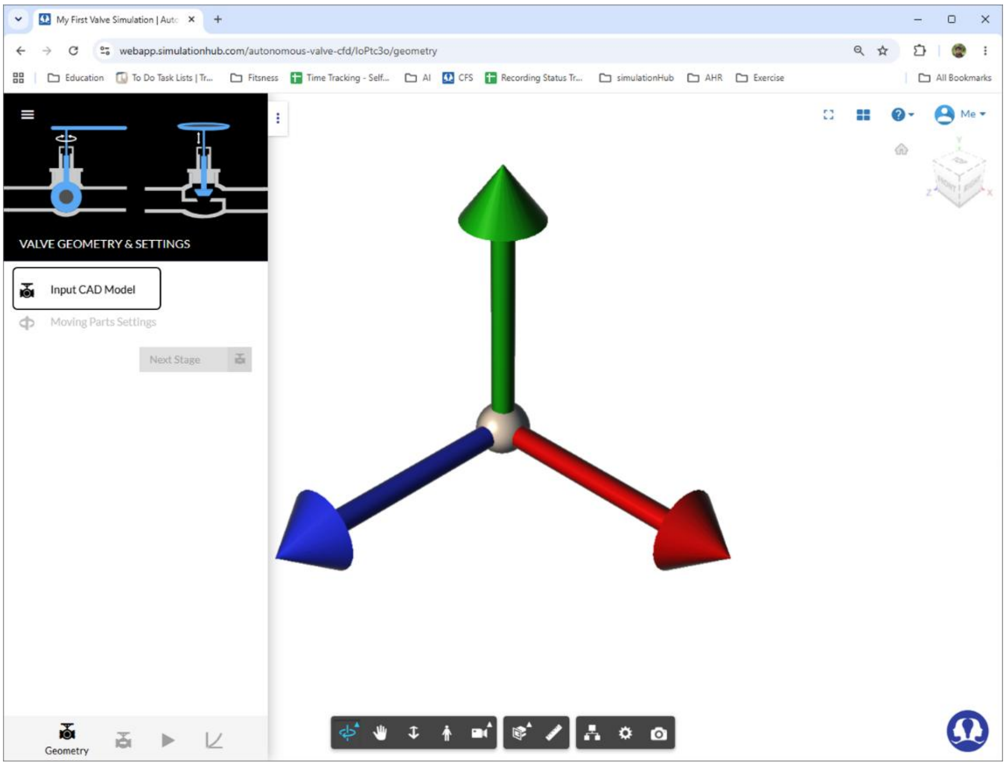


In the New Simulation window, enter a simulation name, such as "My First Valve Simulation", and add a short description for future reference. Then click "**Create Simulation**".

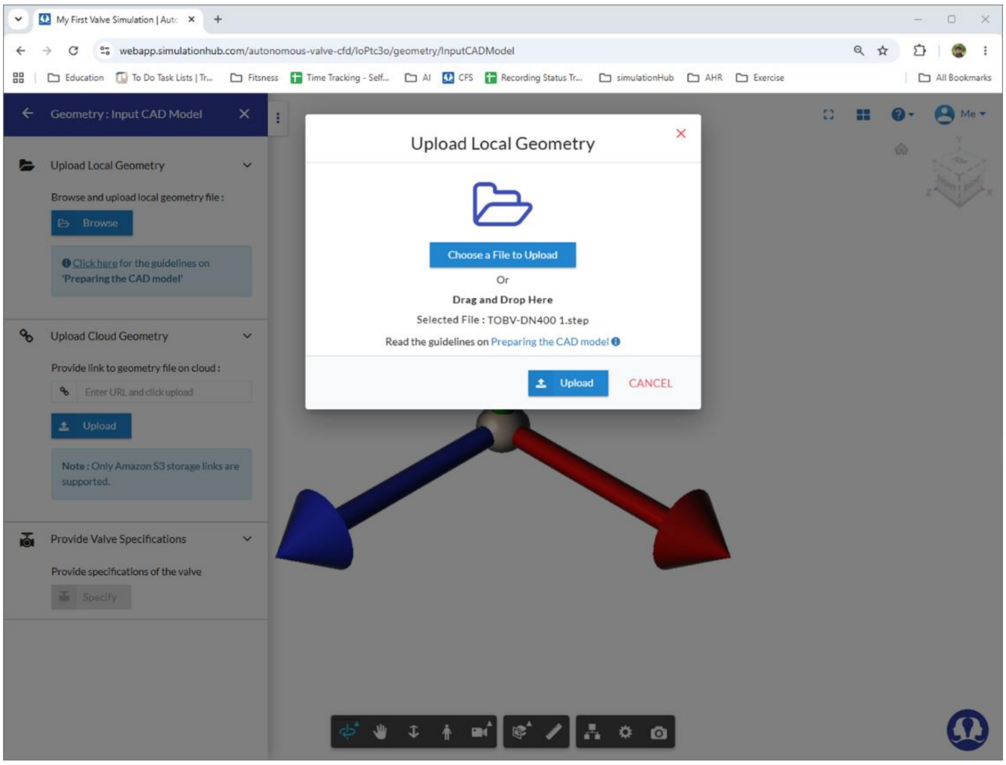


Step 2A: Upload the Valve CAD Model

Once the simulation is created, you will enter the Geometry stage. Click "**Input CAD Model**" to view the options for uploading the valve CAD model.

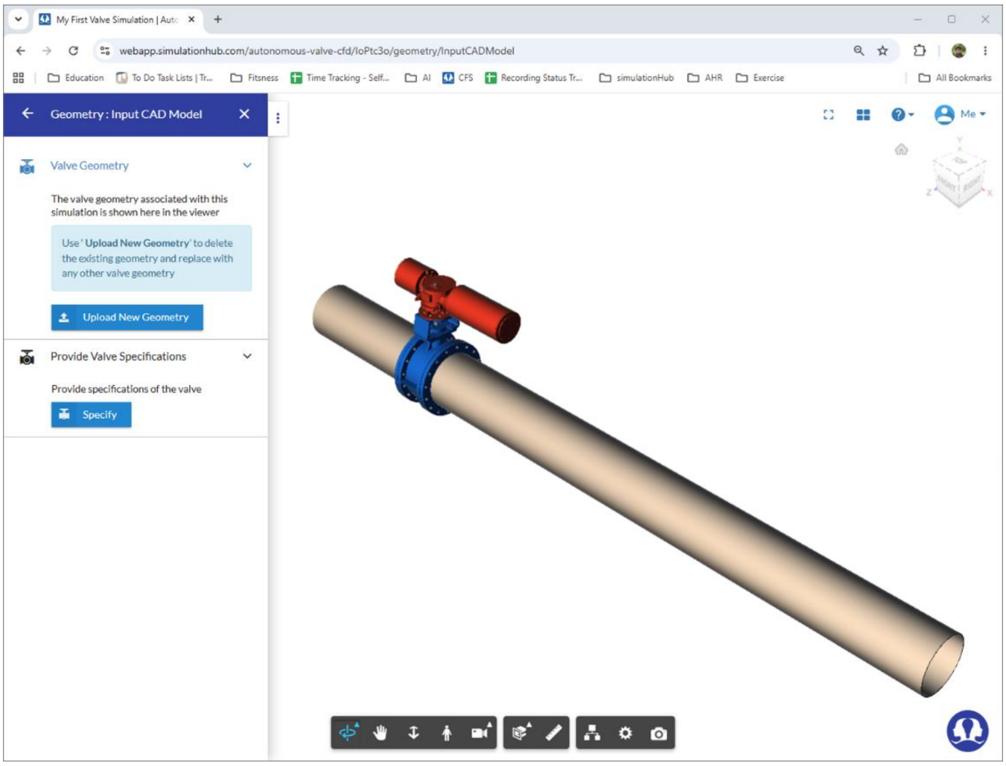


In the pop-up window, click "**Choose a File to Upload**" and select the "TOBV-DN400.step" file from your local directory.



For this tutorial, the CAD model has already been prepared according to the required guidelines and is ready to use. For future simulations, refer to the "Preparing the CAD Model" guidelines before uploading your own valve geometry.

Click "**Upload**" to upload the valve geometry. This may take a minute or two.



Once the geometry is uploaded, the butterfly valve model will appear in the 3D viewer.

Mouse Controls for 3D Viewer Navigation

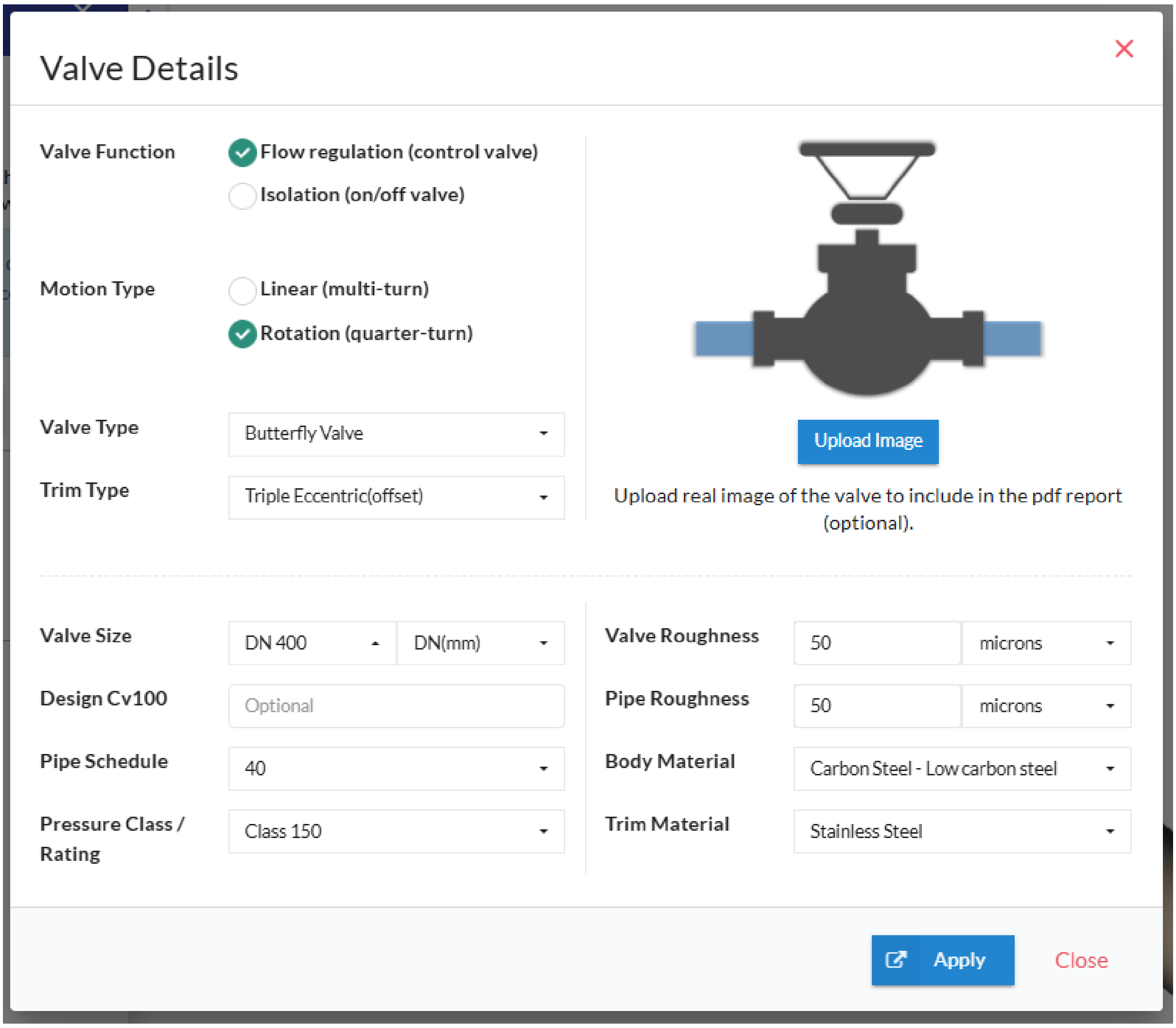
Use the following mouse controls to inspect the valve geometry:

- Left click: Orbit/rotate the view
- Right click: Pan the view
- Scroll: Zoom in or out

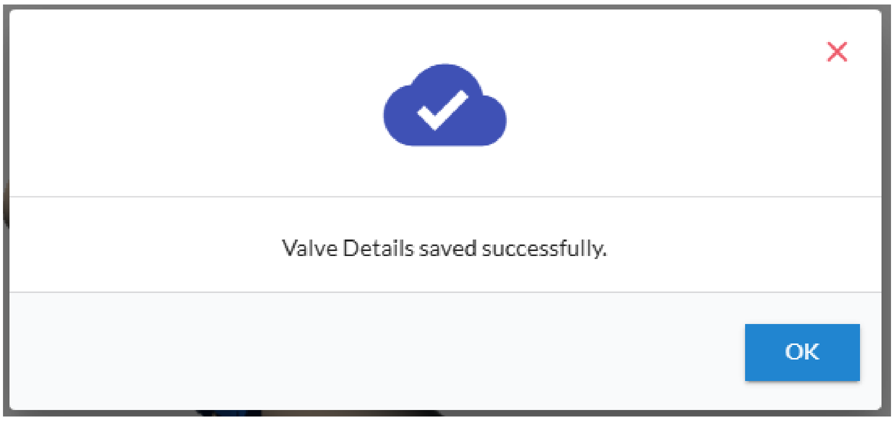
Use left click to select a single body part of the valve model. Use Ctrl + left click to select multiple parts. After selecting parts, right click in the viewer to access options such as "Isolate" or "Hide". These options help you inspect the valve model in detail. You can also explore other options from the toolbar at the bottom of the viewer.

Step 2B: Provide Valve Specifications

Next, provide the valve specification details. Click "**Specify**" and complete the details in the pop-up window. These include valve function, motion type, valve type, trim type, valve size, valve roughness, pipe roughness, and other relevant inputs.

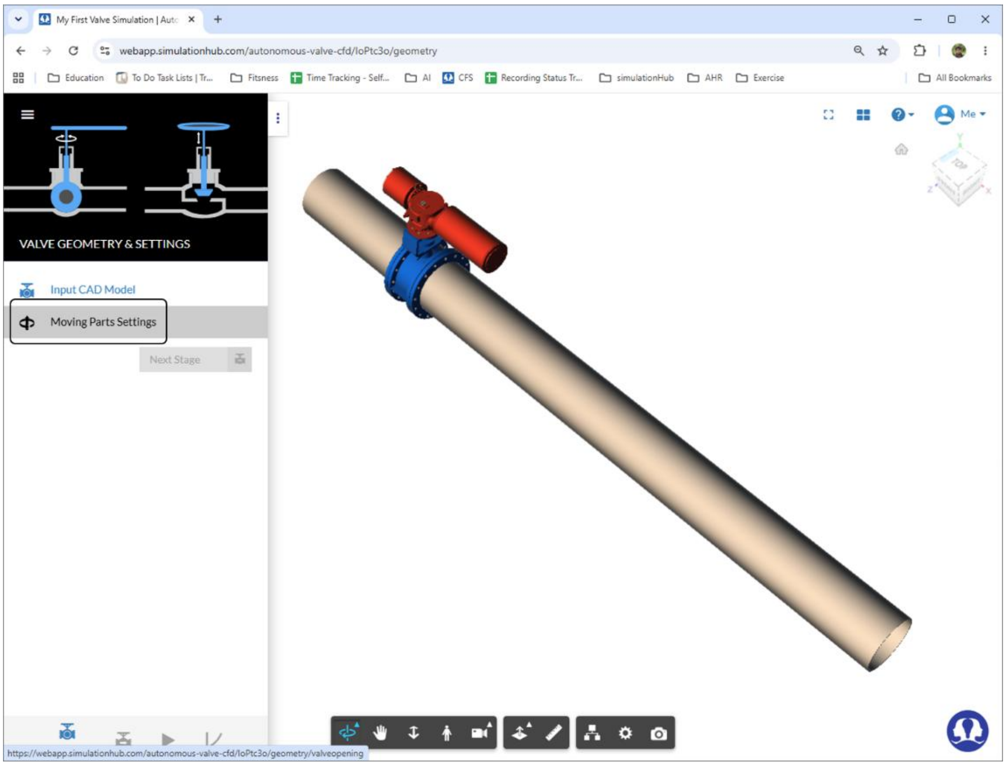


After entering all the required details, click "**Apply**" to save the valve specifications.



Step 3: Configure Moving Part Settings

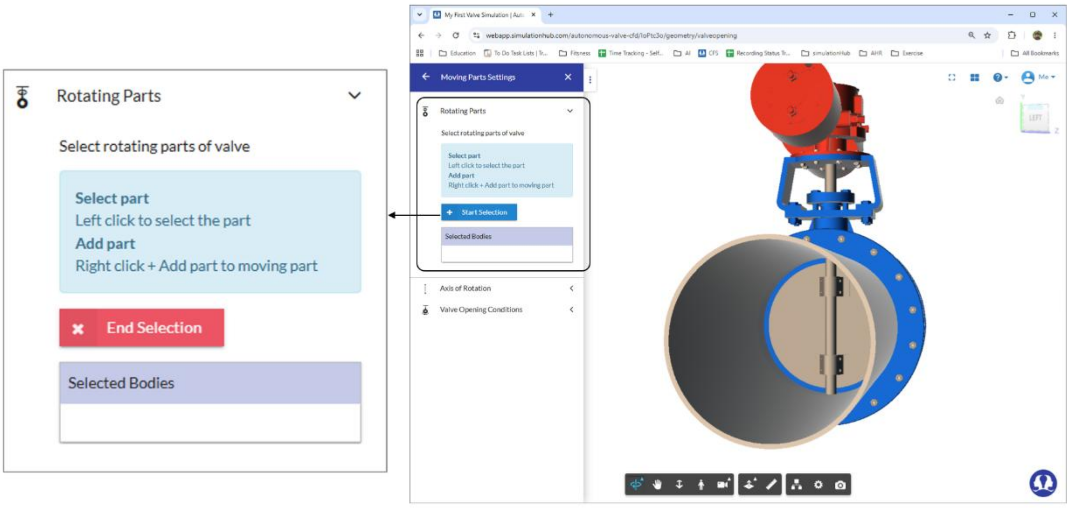
The next step in the Geometry stage is to define the valve trim motion. Select "**Moving Parts Settings**" under the Geometry stage.



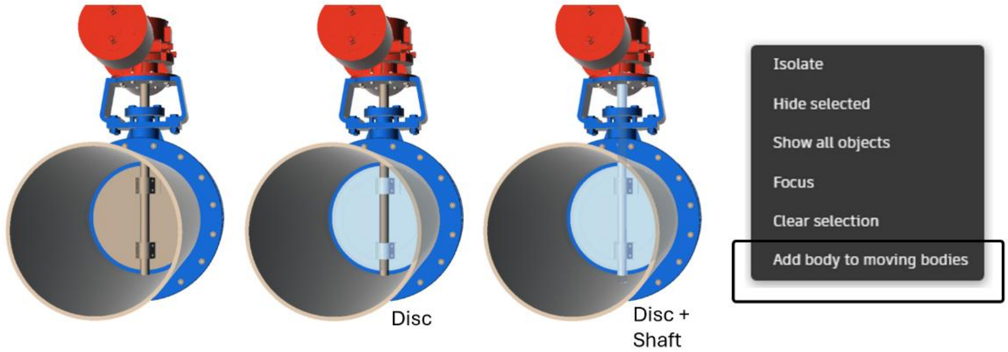
In Moving Parts Settings, you will define three key inputs related to valve motion.

Step 3A: Select Rotating or Moving Parts

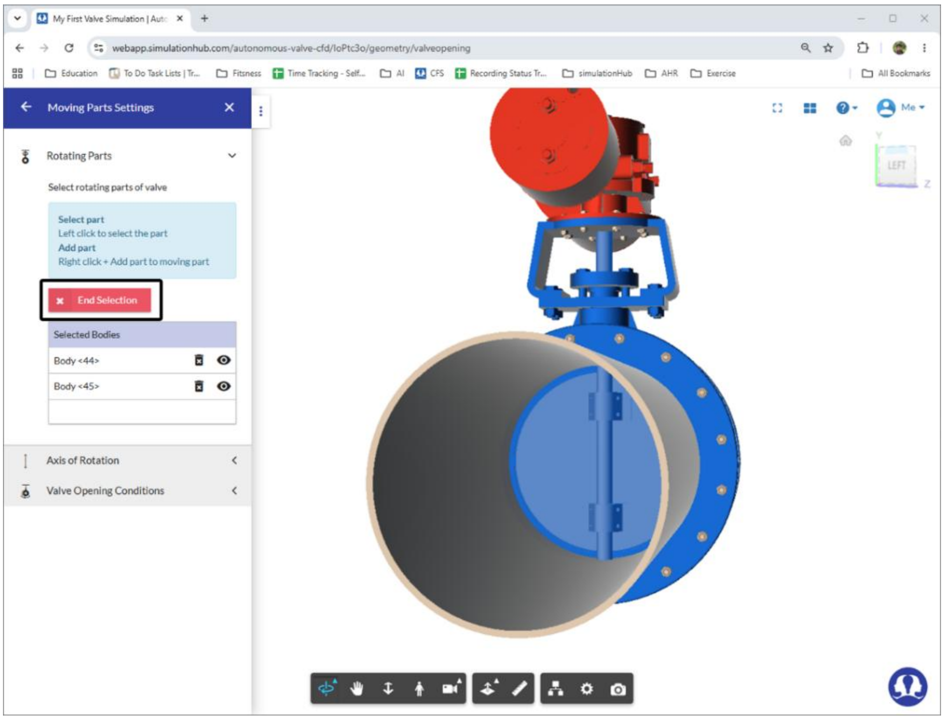
In the Rotating (or Moving) Parts section, click "**Start Selection**" to enter selection mode. You will notice the cursor icon change in the viewer when selection mode is active.



Adjust the model view so that the moving parts are easy to select. Then select the rotating parts, such as the disc, shaft, and any other applicable rotating components, using Ctrl + left click.

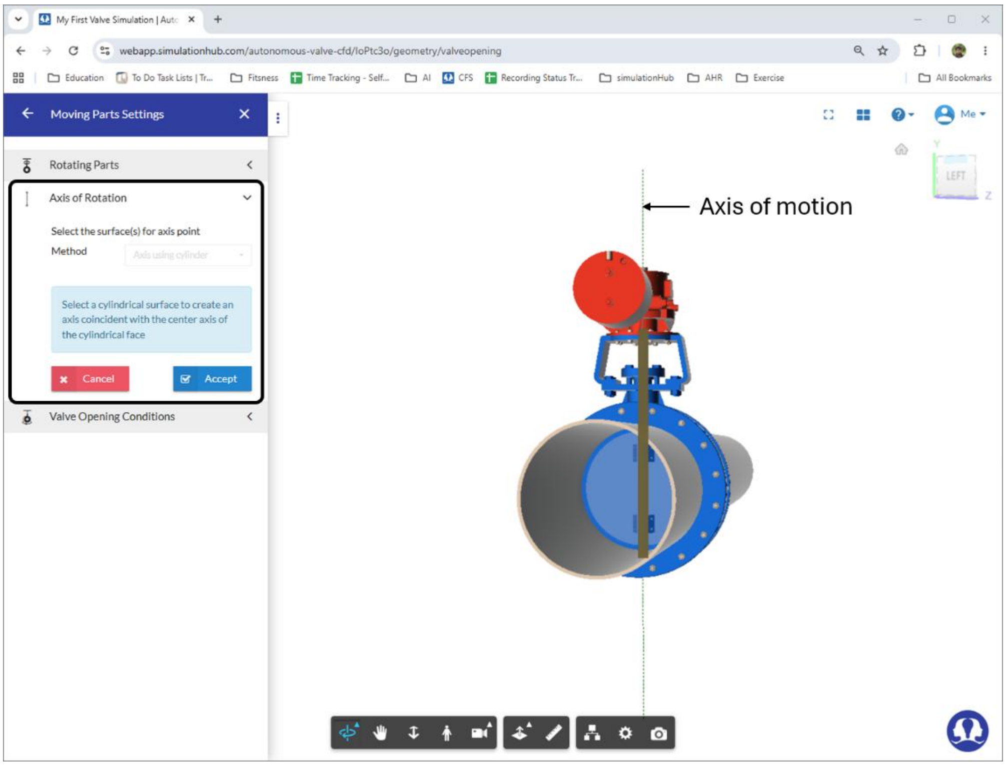


Right click anywhere in the viewer and select "**Add body to moving bodies**". You can verify the selected parts in the list shown on the left side of the window. Click "**End Selection**" to complete the moving-part selection.



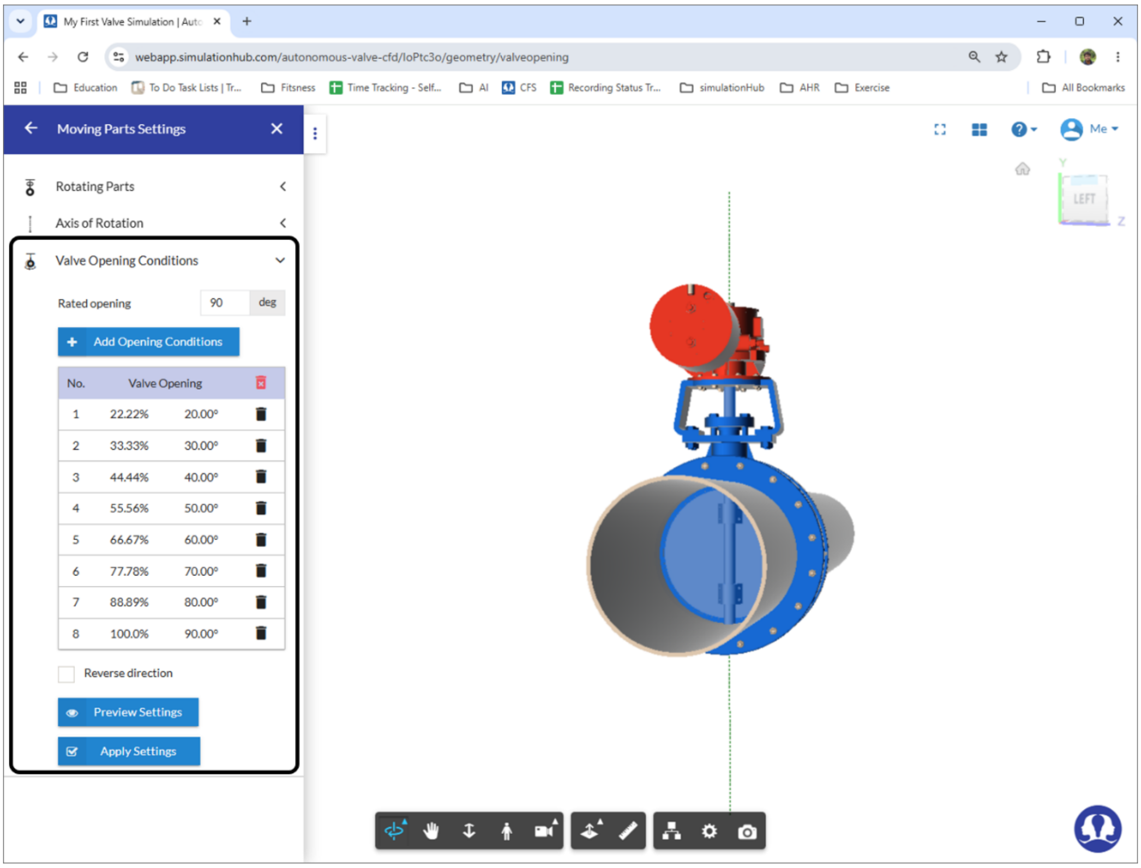
Step 3B: Define the Axis of Rotation

Next, define the axis of motion. For this butterfly valve, use the "**Axis using cylinder**" method. Select the cylindrical surface of the shaft, preview the axis shown as dotted lines, and click "**Accept**".



Step 3C: Define Valve Opening Conditions

In this step, define the valve openings for which simulations will be performed. By default, openings from 20° to 90° are added to the list. You can add or remove openings as needed. Each opening consumes one simulation credit; therefore, the default set of eight openings requires 8 simulation credits.

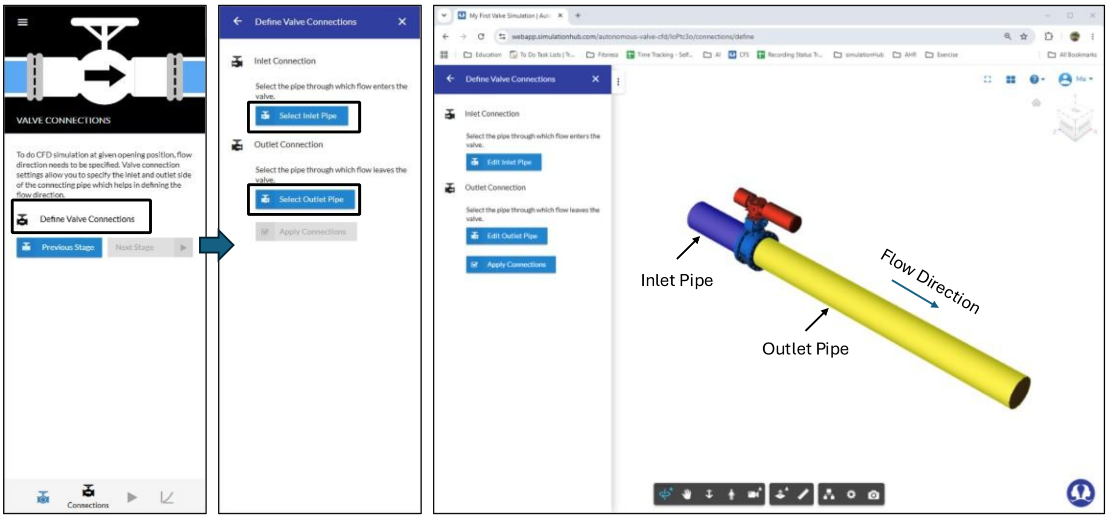


Use “Preview Settings” to preview the motion of the selected moving parts. This helps confirm that only the intended parts are moving and that the motion direction is correct. If required, select “Reverse direction” to change the direction of motion, such as clockwise/counterclockwise or upward/downward.

Click “**Apply Settings**” to complete the moving part configuration.

Step 4: Define Valve Connections

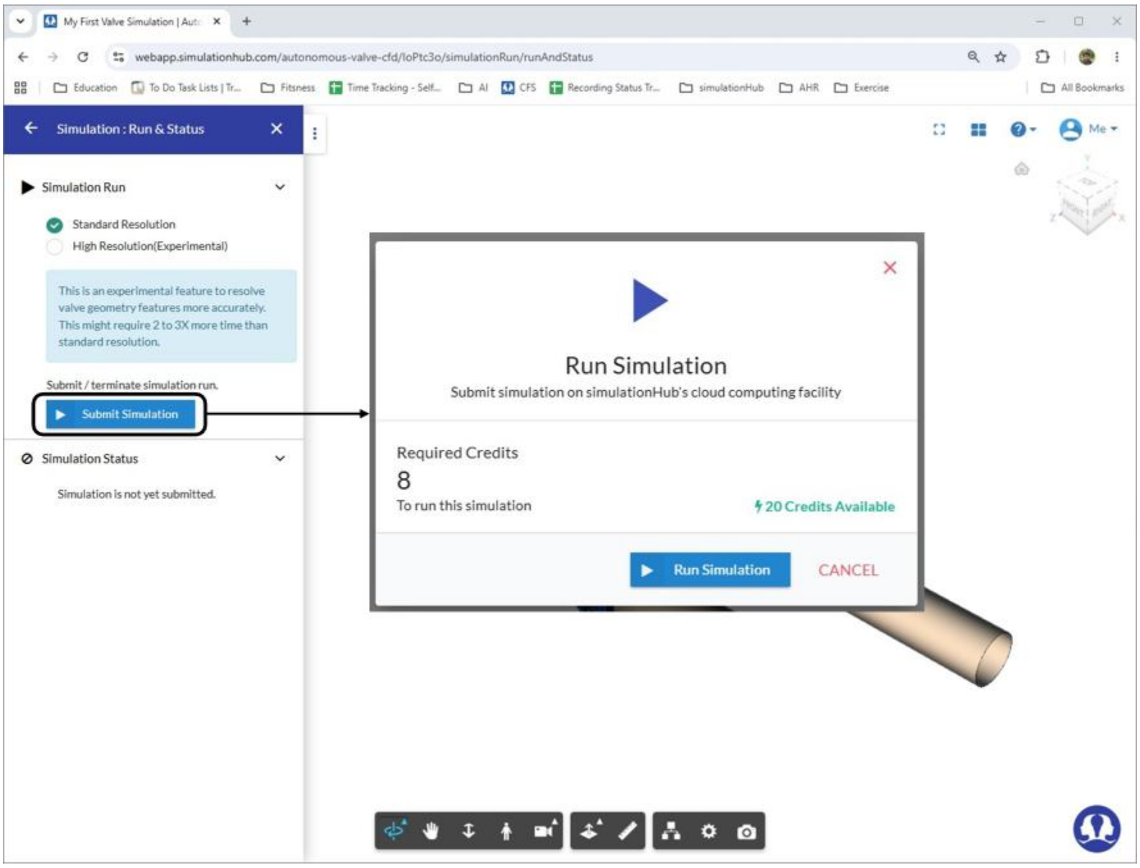
In the Connection stage, select “Define Valve Connections” to define the flow direction. Use “Select Inlet Pipe” and “Select Outlet Pipe” to select the inlet and outlet pipes, as shown in the image below. This sets the flow direction for the simulation.



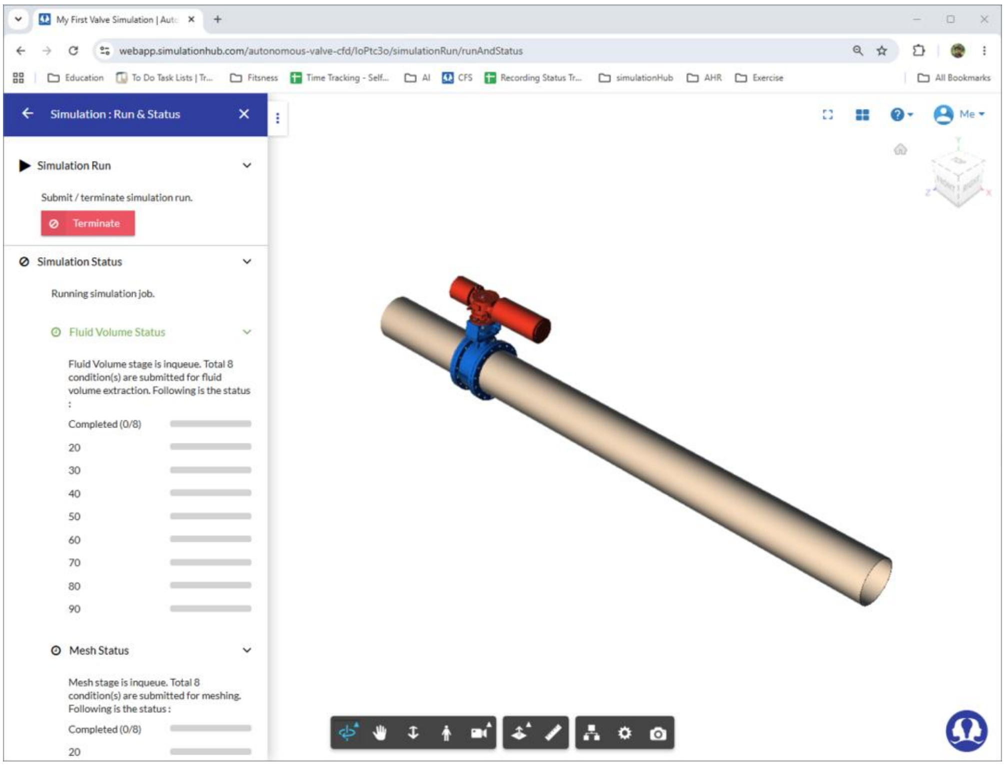
Click “**Apply Connections**” to save the connection settings and move to the next stage.

Step 5: Run the Simulation

In the Solve stage, select “Simulation Run & Status” to open the run options. Choose “**Standard Resolution**” and click “**Submit Simulation**” to submit the simulation request. The Run Simulation pop-up shows the total number of simulation credits required and the available credits in your account.

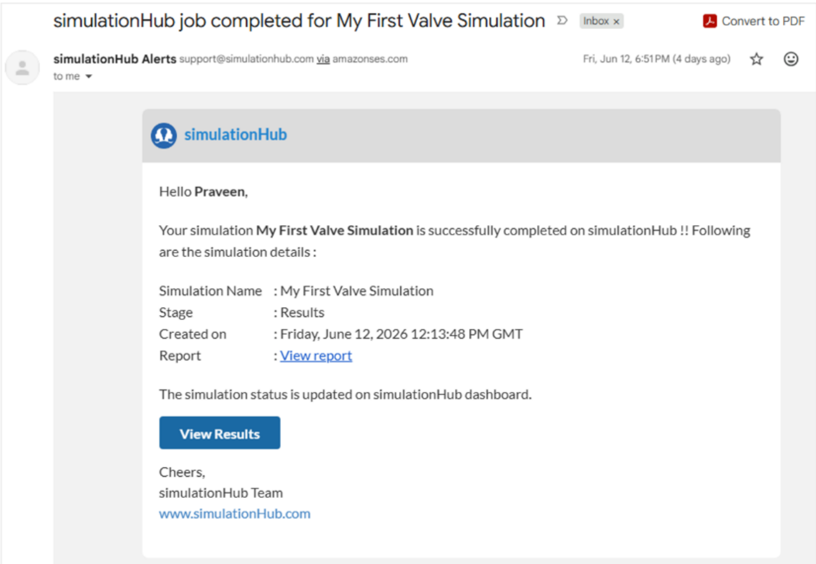


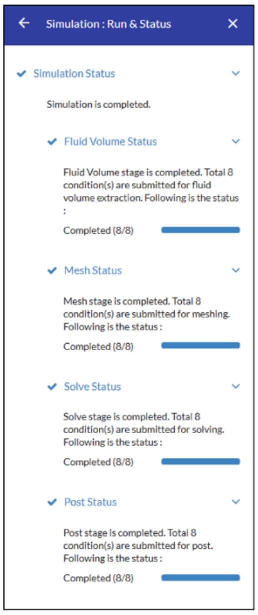
Click “**Run Simulation**” to start the CFD simulations on the cloud. You can track the progress of each simulation using the progress bars in the Simulation Status section.



The simulations may take an hour or two to complete, depending on the valve type, valve size, and geometry complexity. While the simulations run on the cloud, you can continue with other design tasks.

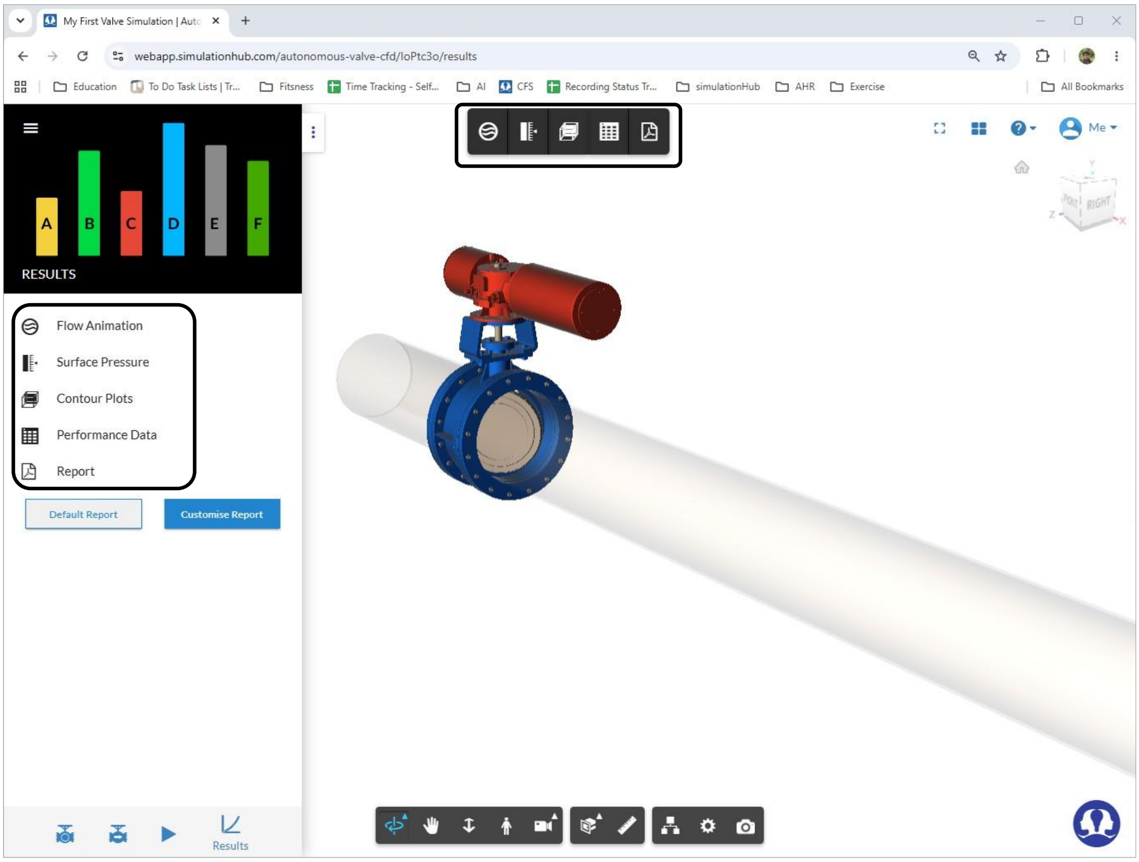
Once the simulations are complete, you will receive an email notification from support@simulationhub.com. The completed simulation status is shown in the image below.





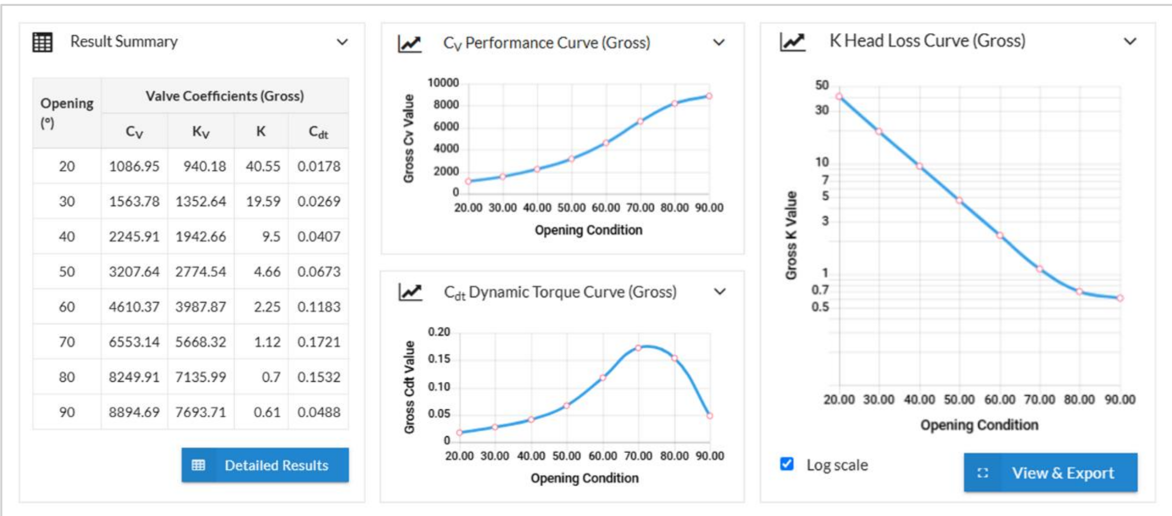
Step 6: Review the Results

The Results stage provides post-processing tools to view, interpret, and export the simulation results. The following sections explain the key result options available in AVC.



Performance Data

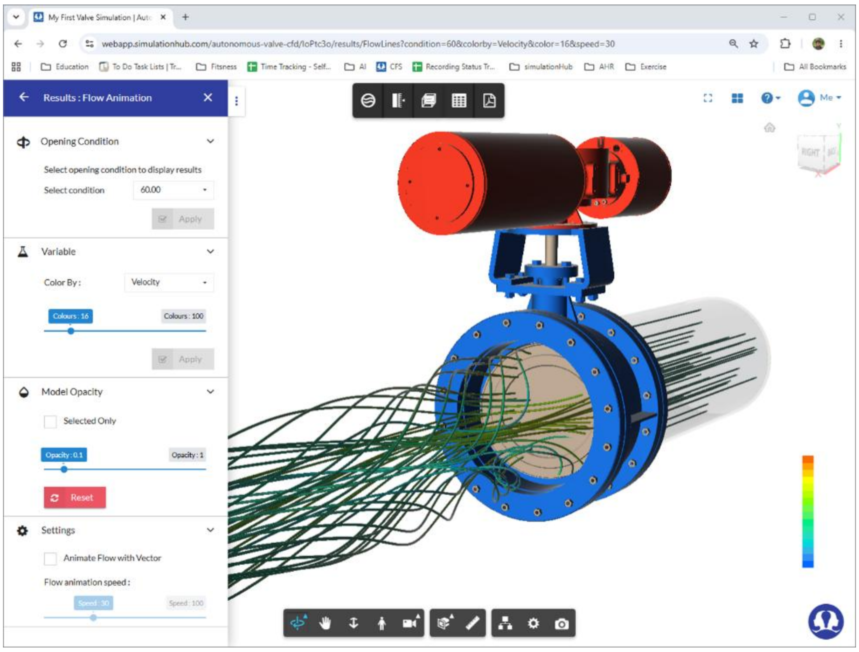
This section provides valve flow performance data from the CFD simulations, including valve flow coefficient (Cv), hydrodynamic torque coefficient (Cdt), head loss coefficient, flow rate, and pressure drop. The data is available in both tabular and graphical formats.



Opening (°)	Valve Coefficients (Gross)			
	C_V	K_V	K	C_{dt}
20	1086.95	940.18	40.55	0.0178
30	1563.78	1352.64	19.59	0.0269
40	2245.91	1942.66	9.5	0.0407
50	3207.64	2774.54	4.66	0.0673
60	4610.37	3987.87	2.25	0.1183
70	6553.14	5668.32	1.12	0.1721
80	8249.91	7135.99	0.7	0.1532
90	8894.69	7693.71	0.61	0.0488

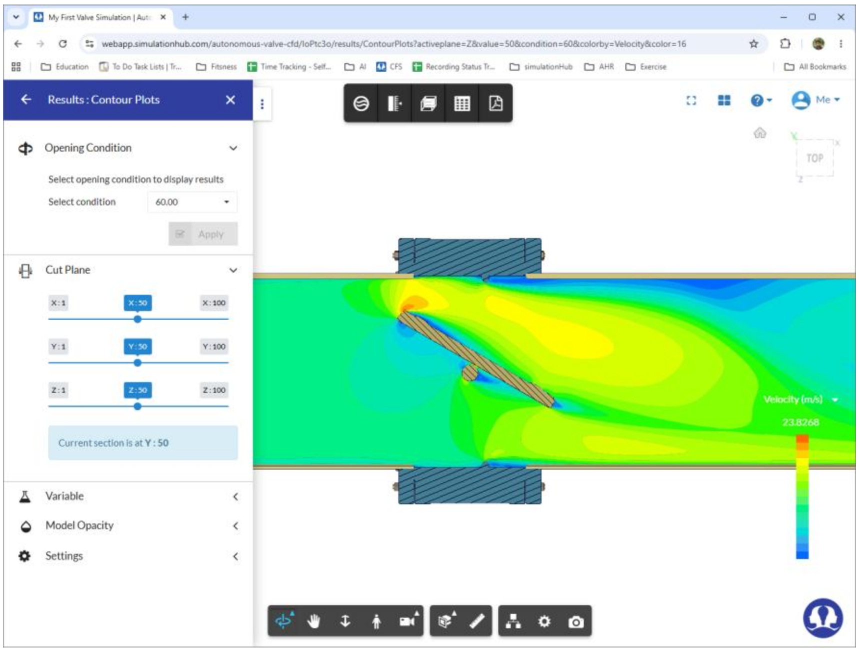
Flow Animation

Flow lines help visualize how flow particles move from the valve inlet, across the valve trim, and toward the outlet. This helps you identify flow separation, vortices, swirl regions, and other important flow features. AVC also provides options to control the flow-line visualization for better analysis.



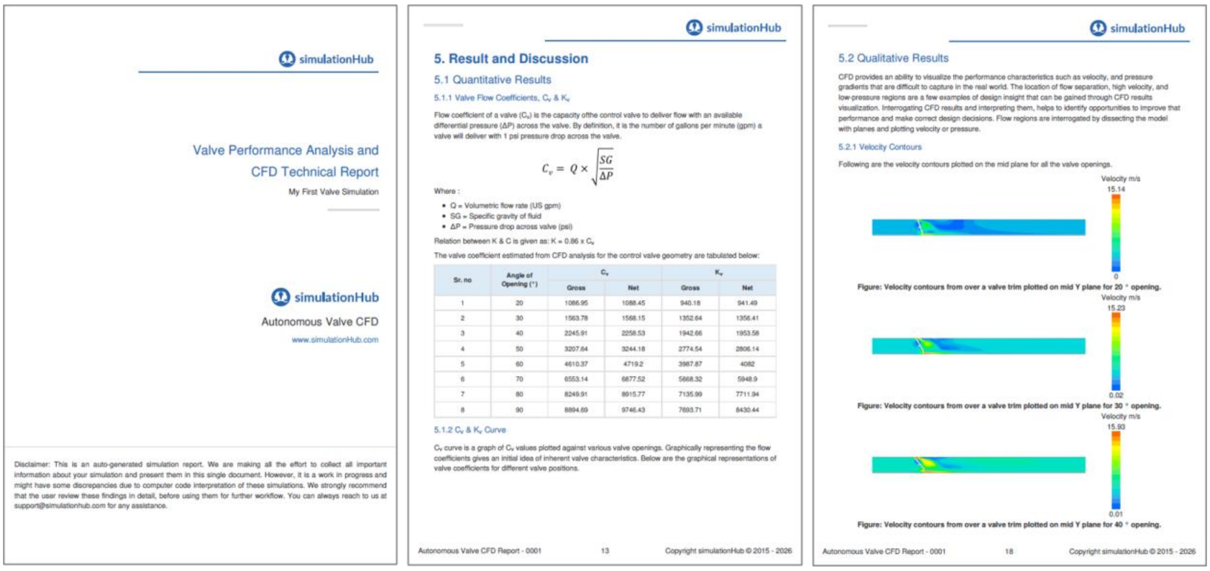
Contour Plots

CFD contour plots help visualize scalar flow variables, such as pressure and velocity, across a 2D cut section of the valve. Velocity contours help identify boundary layers, separation zones, and wakes. Pressure contours highlight high- and low-pressure regions, which are important for understanding pressure recovery and cavitation risk.



Report

AVC automatically generates a detailed CFD report that summarizes the valve performance data along with velocity and pressure contours. The report also includes relevant CFD process details. You can download the report in PDF format for documentation, review, or sharing with your team.



simulationHub

Valve Performance Analysis and CFD Technical Report

My First Valve Simulation

simulationHub

Autonomous Valve CFD

www.simulationHub.com

Disclaimer: This is an auto-generated simulation report. We are making all the effort to collect all important information about your simulation and present them in this single document. However, it is a work in progress and might have some discrepancies due to computer code interpretation of these simulations. We strongly recommend that the user review these findings in detail, before using them for further workflow. You can always reach to us at support@simulationhub.com for any assistance.

simulationHub

5. Result and Discussion

5.1 Quantitative Results

5.1.1 Valve Flow Coefficients, C_v & K_v

Flow coefficient of a valve (C_v) is the capacity ofthe control valve to deliver flow with an available differential pressure (ΔP) across the valve. By definition, it is the number of gallons per minute (gpm) a valve will deliver with 1 psi pressure drop across the valve.

$$C_v = Q \times \sqrt{\frac{SG}{\Delta P}}$$

Where :

- Q = Volumetric flow rate (US gpm)
- SG = Specific gravity of fluid
- ΔP = Pressure drop across valve (psi)

Relation between K & C is given as: $K = 0.86 \times C_v$

The valve coefficient estimated from CFD analysis for the control valve geometry are tabulated below:

Sr. no	Angle of Opening (°)	C_v		K_v	
		Gross	Net	Gross	Net
1	20	1086.95	1088.45	940.18	941.49
2	30	1563.78	1568.15	1352.64	1356.41
3	40	2245.91	2258.53	1942.66	1953.58
4	50	3207.64	3244.18	2774.54	2806.14
5	60	4610.37	4719.2	3987.87	4082
6	70	6553.14	6877.52	5668.32	5948.9
7	80	8249.91	8915.77	7135.99	7711.94
8	90	8894.69	9746.43	7693.71	8430.44

5.1.2 C_v & K_v Curve

C_v curve is a graph of C_v values plotted against various valve openings. Graphically representing the flow coefficients gives an initial idea of inherent valve characteristics. Below are the graphical representations of valve coefficients for different valve positions.

Autonomous Valve CFD Report - 0001   13   Copyright simulationHub © 2015 - 2026

simulationHub

5.2 Qualitative Results

CFD provides an ability to visualize the performance characteristics such as velocity, and pressure gradients that are difficult to capture in the real world. The location of flow separation, high velocity, and low-pressure regions are a few examples of design insight that can be gained through CFD results visualization. Interrogating CFD results and interpreting them, helps to identify opportunities to improve that performance and make correct design decisions. Flow regions are interrogated by dissecting the model with planes and plotting velocity or pressure.

5.2.1 Velocity Contours

Following are the velocity contours plotted on the mid plane for all the valve openings.

Figure: Velocity contours from over a valve trim plotted on mid Y plane for 20 ° opening.

Figure: Velocity contours from over a valve trim plotted on mid Y plane for 30 ° opening.

Figure: Velocity contours from over a valve trim plotted on mid Y plane for 40 ° opening.

Autonomous Valve CFD Report - 0001   18   Copyright simulationHub © 2015 - 2026

Conclusion: From First Simulation to Design Insight

You have now completed your first valve flow simulation using Autonomous Valve CFD. Through this workflow, you created a new simulation, uploaded the valve CAD model, defined the moving parts and axis of rotation, assigned inlet and outlet connections, submitted cloud-based simulations, and reviewed the resulting performance data and flow visualizations.

The objective of this tutorial is not only to help you run one example, but also to make the complete AVC workflow familiar and repeatable. For your next project, you can apply the same steps to your own valve design, compare multiple opening conditions, evaluate Cv and Cdt trends, study pressure and velocity contours, and generate a ready-to-share CFD report.

With AVC, valve designers can move from CAD geometry to actionable flow-performance insight without setting up conventional CFD manually. This makes it easier to validate designs earlier, explore design changes faster, and support product decisions with simulation-backed evidence.