

Guidelines Document

International Valve Design Challenge 2019



Fluid flow simulation apps for designers
<https://www.simulationhub.com>

Updated on Nov 15th, 2019

© Copyrights: 2015 - Current. All material in this document is, unless otherwise stated, the property of **simulationHub**. Copyright and other intellectual property laws protect these materials. Reproduction or retransmission of the materials, in whole or in part, in any manner, without the prior written consent of the copyright holder, is a violation of copyright law.

Copies of the document are made available for review. Individuals must preserve any copyright or other notices contained in or associated with them. Users may not distribute such copies to others, whether or not in electronic form, whether or not for a charge or other consideration, without prior written consent of the copyright holder of the materials. Contact information for requests for permission to reproduce or distribute materials available through this document is listed below:

simulationHub

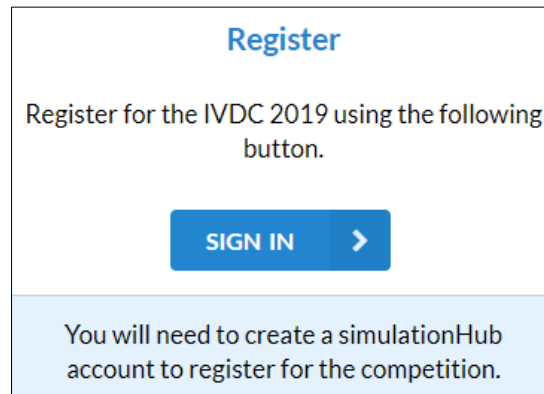
403, Pushpak Business Hub, Wakad
Pune - 411057, India

TABLE OF CONTENTS

1	HOW TO PARTICIPATE IN THE COMPETITION	3
1.1	REGISTRATION	3
1.2	CAD FILES AND TECHNICAL DOCUMENT	3
1.3	DESIGN MODIFICATIONS AND CFD SIMULATION	4
1.4	SUBMIT RESULTS	4
2	DESIGN PROBLEM DETAILS	6
2.1	WHAT IS THE PROBLEM TO BE SOLVED?	6
2.1.1	VALVE DETAILS	6
2.1.2	DESIGN OBJECTIVE	7
2.2	HOW TO PREDICT THE C_v VALUE FOR YOUR VALVE DESIGN?	7
2.3	RESULT EVALUATION	8
3	DESIGN CONSTRAINTS	9
3.1	DIMENSIONAL CONSTRAINTS	9
3.1.1	VALVE BODY	9
3.1.2	DISC	11
3.1.3	STEM AND PINS	12
3.1.4	SEAT RETAINER	13
3.2	REFERENCES	14

1 HOW TO PARTICIPATE IN THE COMPETITION

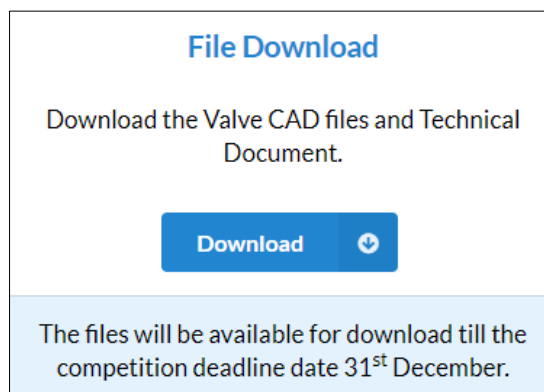
1.1 REGISTRATION



To participate in the International Valve Design Challenge (IVDC) 2019, you will first need to register for the competition. Here are the steps on how to register for the competition:

1. Go to the IVDC 2019 home page: [IVDC 2019](#)
2. Go to How to participate section:
 - a. You will need to create a simulationHub account to register for the competition. Click on Sign-In and create an account.
3. Once registration is complete, you will receive an email confirming your registration.

1.2 CAD FILES AND TECHNICAL DOCUMENT



You will be provided with the following files:

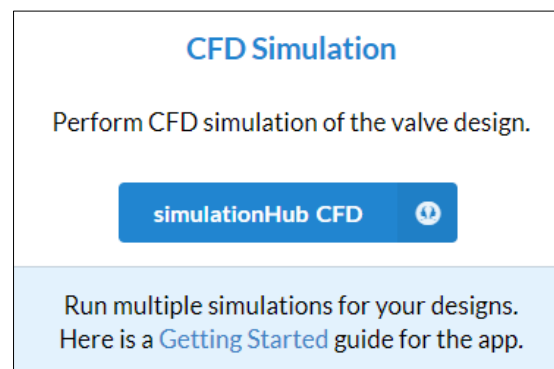
1. Valve CAD files (Assembly model(.iam); Multi-body part model(.step,.ipt))
2. Technical drawings of all parts
3. Competition guidelines document

4. Guidelines on how to prepare CAD model for CFD simulation
5. Demo video on how to perform a CFD simulation using Autonomous Valve CFD app

1.3 DESIGN MODIFICATIONS AND CFD SIMULATION

You can now modify the design using the CAD files and with the help of the technical guidelines document. You must make sure that the modified CAD design satisfies the mentioned [design constraints](#).

Once your design is ready you can evaluate valve Cv using the simulationHub Valve CFD app.



To use the app:

1. Click on the 'simulationHub CFD' under the How to participate section.
2. You will be directed to your [simulationHub Dashboard](#)

You can use the [Getting Started](#) guide to start using the app.

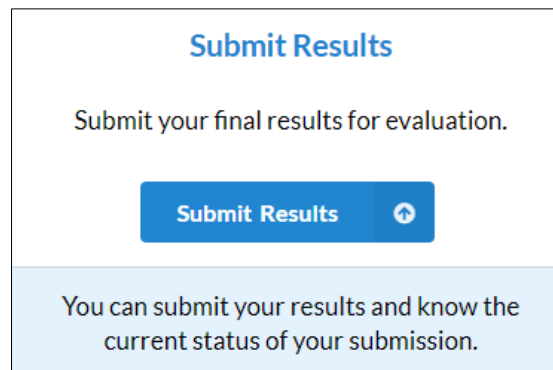
You can also look at the demo video provided with the downloads to understand the workflow of the app.

You will be provided with **60 simulation credits** in your account. You can use these credits to perform multiple CFD simulations on different valve designs.

1.4 SUBMIT RESULTS

After completing the design iterations and their CFD simulations you will have to submit these simulations for evaluation.

To do so, go to the Submit Results section on the IVDC 2019 home page.



You will be directed to your [Designer Profile](#) wherein you can submit your simulation through the following steps:

1. Select the project to be submitted from the 'Submit design for review' drop-down menu.
2. Click on the 'Submit Design' button.

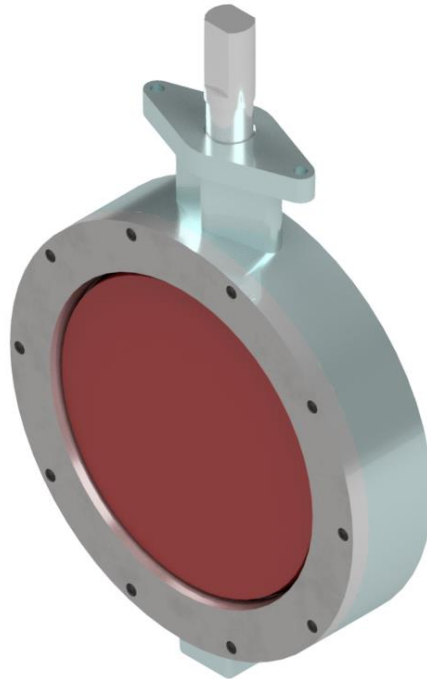
You will also be able to view the status of your previously submitted projects. Along with that, you can also see your current rank in the competition.

2 DESIGN PROBLEM DETAILS

2.1 WHAT IS THE PROBLEM TO BE SOLVED?

2.1.1 VALVE DETAILS

- Following are the details of the valve:



CAD Model of the DN250 Double Offset Butterfly Valve

NO.	PROPERTY	DESCRIPTION
1	Type	Butterfly Valve
2	Sub-type	Double Offset
3	Size	DN250 (10-inch)
4	Pressure Class	ASME Class 150
5	Valve Body Material	Carbon Steel
6	Disc Material	Stainless Steel
7	Stem Material	Stainless Steel
8	Seat Material	PTFE

- A DN250 (10-inch) Double Offset Butterfly Valve CAD model has been provided to all the participants.
- The CAD model is available in both Multi-part assembly files and Multi-body part files.
- Engineering drawings of every part have also been provided along with the CAD files.

2.1.2 DESIGN OBJECTIVE

- The designer's objective is to modify the valve design optimally such that the [valve flow coefficient \(Cv\) is maximized and the weight of the disc is minimized](#).
- While doing so, care must be taken that the [modified design follows all the design constraints mentioned](#) in this guidelines document. The design constraints section illustrates all the allowable modifications in the valve design and dimensional constraints that must be followed.

2.2 HOW TO PREDICT THE Cv VALUE FOR YOUR VALVE DESIGN?

- The Cv value for the modified valve design must be predicted using the [simulationHub Autonomous Valve CFD](#) app.
- Details on how to use the simulationHub Valve CFD app are available here: [Getting Started guideline](#)
- You will be provided with 60 simulation credits at the beginning of the competition. You can perform multiple CFD simulations using these simulation credits, where 1 simulation credit will be consumed for each valve opening condition.
- While performing design iterations you can perform the simulation for a minimum of 2 opening conditions e.g. (80° and 90°)
- [Use the default roughness values:](#)
 - [Pipe roughness = 50 microns](#)
 - [Valve roughness = 50 microns](#)
- [While submitting the simulation project it is mandatory that the CFD simulation has been performed for 6 opening angles from 40° to 90° i.e. \(40°, 50°, 60°, 70°, 80°, 90°\).](#)
- You can reach out to our [Technical Support](#) team by raising a support request from the simulationHub dashboard.
- Once your CFD simulation is ready refer the [submit results](#) section for details about project submission.

2.3 RESULT EVALUATION

- The results will be evaluated based on the following criteria:
 - MAX C_v VALUE
 - DISC WEIGHT
 - DESIGN CONSTRAINTS CONFORMANCE
 - STRUCTURAL STRENGTH FOR CLASS 150
- Based on these criteria, the jury will determine the participant's score.
- The scores will determine participant's rank which will be displayed on the Designer Profile page and on the Leader board.

3 DESIGN CONSTRAINTS

3.1 DIMENSIONAL CONSTRAINTS

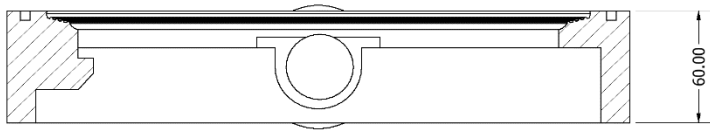
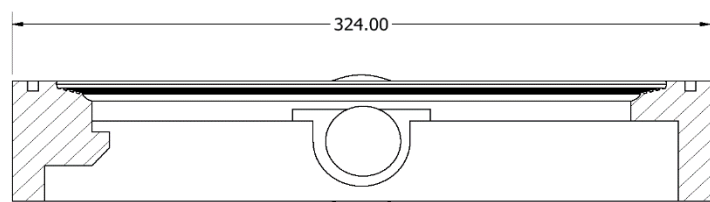
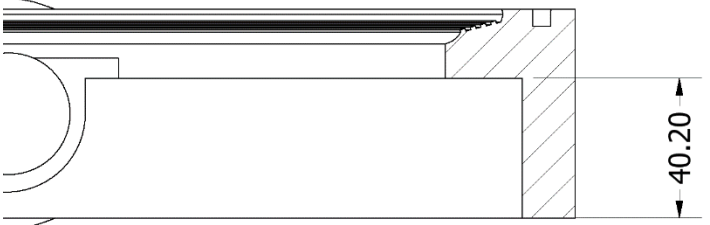
This section illustrates all the geometrical constraints to be followed while modifying the valve design. These constraints have been determined based on API 609, ASME B16.34 and AWWA C504 standards.

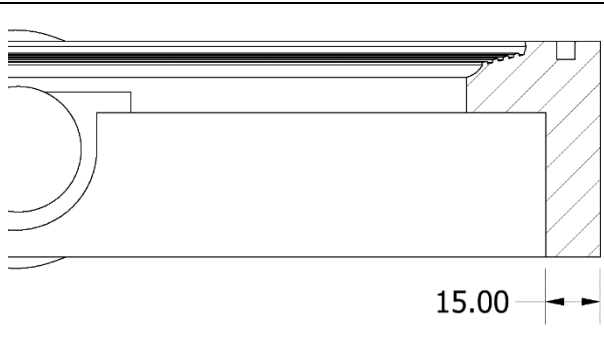
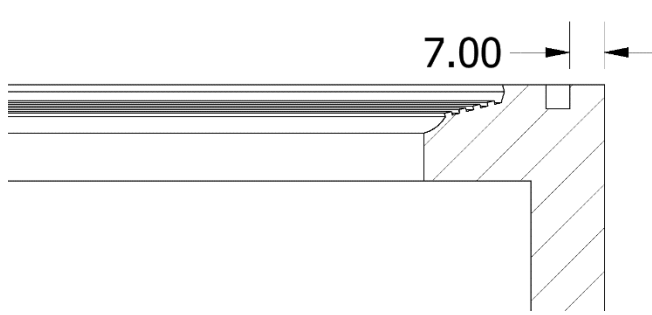
Note that, in the illustrations given below, the geometry regions highlighted in ● must not be modified. The modifiable regions are highlighted in ●. It is mandatory to make sure that while modifying the valve design, the geometrical constraints mentioned below must be maintained.

You may also refer to the technical drawings provided with the CAD files while making the design modifications.

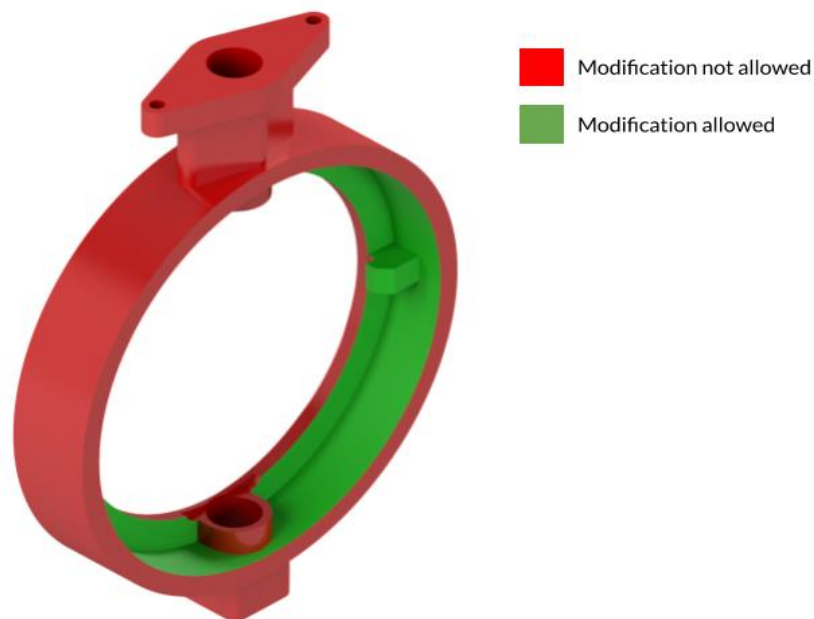
3.1.1 VALVE BODY

- GEOMETRICAL CONSTRAINTS

No.	Parameter	Illustration
1	Face-to-face length = 60 mm	
2	Outer diameter = 324 mm	
3	Internal face-to-face length = 40.2 mm	

4	Minimum wall thickness > 13.7 mm	
5	Hole to outer wall thickness = 7 mm	

- ALLOWABLE MODIFICATIONS

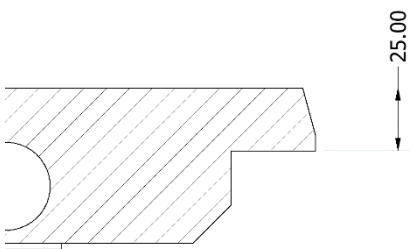
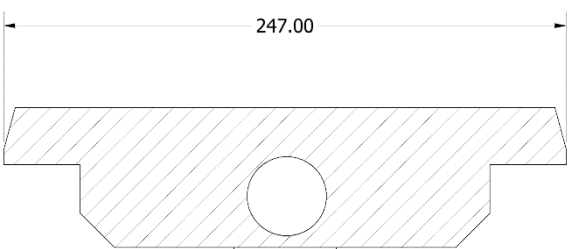
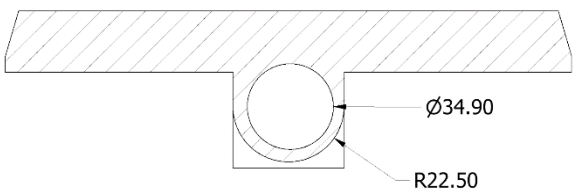


CAD model of Valve Body showing allowable areas for modification

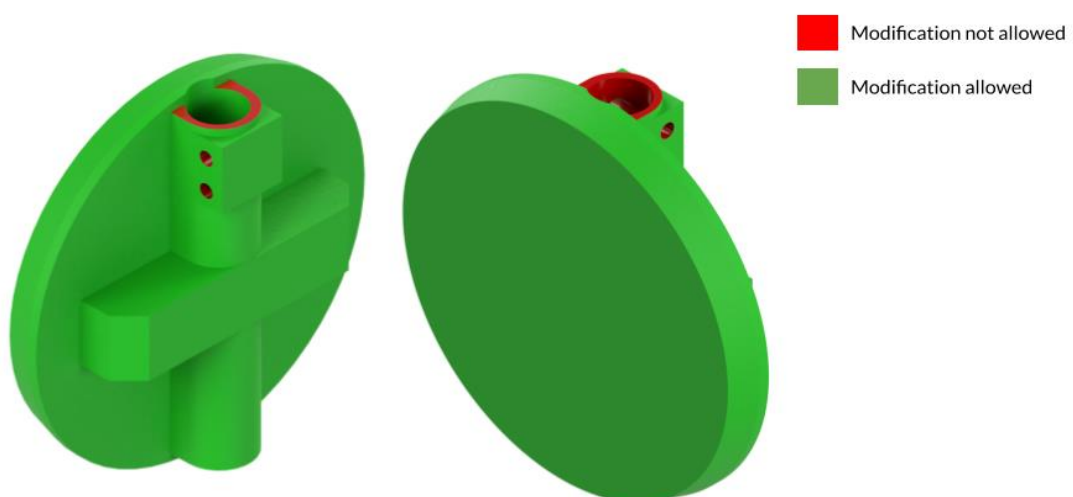
You can modify the valve design in the highlighted green region. Note that this modification must not conflict with the constraints mentioned above.

3.1.2 DISC

- GEOMETRICAL CONSTRAINTS

No.	Parameter	Illustration
1	Disc thickness = 25 mm	
2	Disc maximum diameter = 247 mm	
3	Stem diameter and Bearing diameter	

- ALLOWABLE MODIFICATIONS

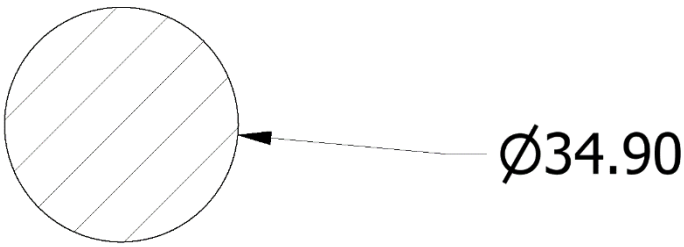
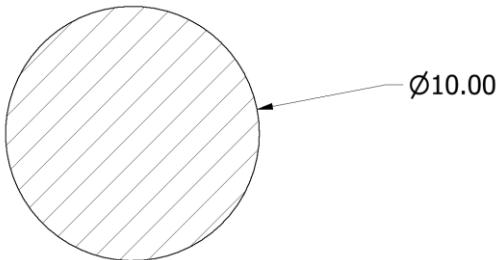


CAD model of the valve disc showing allowable areas for modification

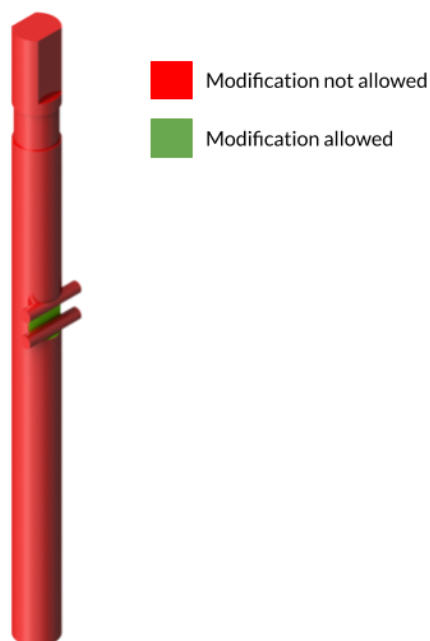
You can modify the valve design in the highlighted green region. Note that this modification must not conflict with the constraints mentioned above.

3.1.3 STEM AND PINS

- GEOMETRICAL CONSTRAINTS

No.	Parameter	Illustration
1	Stem diameter = 34.9 mm	
2	Pin diameter = 10 mm	

- ALLOWABLE MODIFICATIONS

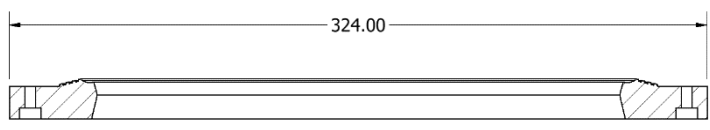
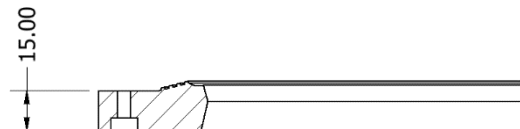


CAD model of the stem and pins showing allowable areas for modification

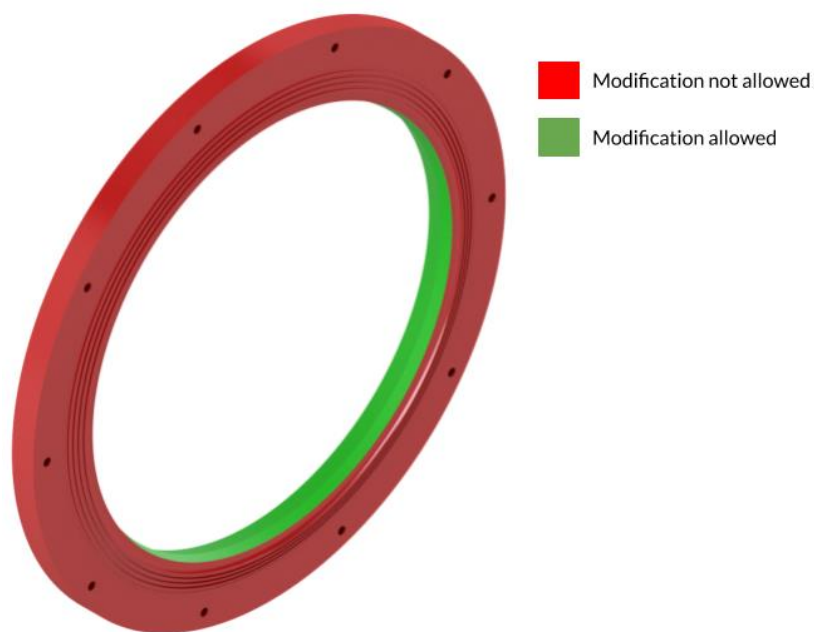
You can modify the valve design in the highlighted green region. Note that this modification must not conflict with the constraints mentioned above.

3.1.4 SEAT RETAINER

- GEOMETRICAL CONSTRAINTS

No.	Parameter	Illustration
1	Outer diameter = 324 mm	
2	Face-to-face length = 15 mm	

- ALLOWABLE MODIFICATIONS



CAD model of the seat retainer showing allowable areas for modification

You can modify the valve design in the highlighted green region. Note that this modification must not conflict with the constraints mentioned above.

3.2 REFERENCES

You may also refer to the following standards while modifying the valve design:

- ASME B16.34 (Valves-Flanged, Threaded, and Welding End)
- ANSI/AWWA C504 (Rubber-Seated Butterfly Valves)
- API 609 (Butterfly Valves: Double Flanged, Lug and Wafer Type)