

ME 24-688 – Week 14

Problem Set Assignment



In this problem set, you will be using a cloud-based CFD software called Ciespace CFD. The software runs on the Amazon cloud – you don't need to install any software on your computer. Access all the CFD functionality through its web-browser-based GUI. Use either Chrome or Firefox browser.

You will simulate the internal water flow within a valve, shown in Figure 1. Figure 2 shows the fluid volume extracted from the valve assembly. See further details of your tasks in the PDF tutorial document posted on the class web. Figure 3 shows the simulation results.

Step 1: Watch a 4-minute video clip that explains the overview of the cloud-based CFD software on the web at: <http://www.ciespace.com>. The video clip can be found at the bottom of the web page.

Step 2: Logon to the Ciespace website <https://na1.ciespace.com/> using the user name and password provided by the instructor.

Step 3: Follow the instruction in the tutorial document posted on the class web and perform a CFD analysis of the internal water flow in the valve. Do NOT delete your work after you finish the tutorial.

Step 4: Answer each of the following three questions with more than 40 words and less than 100 words and submit them on a web form – the link is listed on the class web.

- What is your overall impression about the cloud-based CFD software?
- What did you like about the cloud-based CFD software?
- What are your thoughts on how the cloud-based CFD software can be improved?

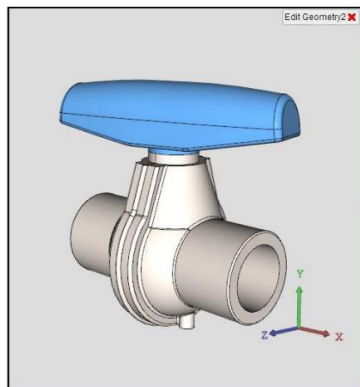


Figure 1: A valve assembly

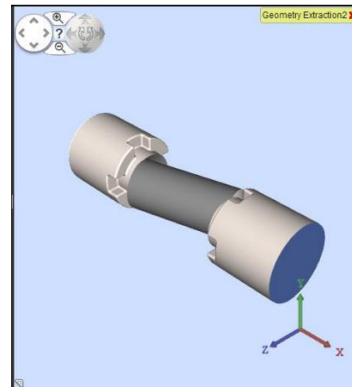


Figure 2: The internal flow volume

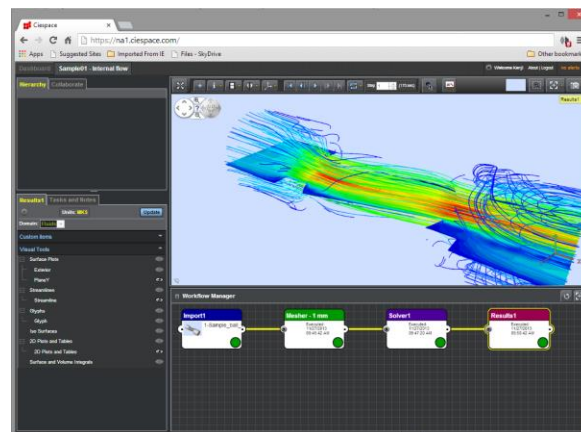


Figure 3: Cloud-based CFD Software