stabilizing cooling flow.

PREVENT CAVITATION WITH INSERTS

CONTACT

+31 (0)88 - 115 20 20
demcon.com/multiphysics

Goal

A liquid cooling system is often used when a high heat load is present on a key component of a system, for a long period. To prevent damage due to the heat, the coolant must continuously negate the full heat load. A small issue in the coolant supply can lead to significant damage. It is therefore important to take measures against possible problems that limit the flow and disrupt the cooling process.

The observed system is an abstract representation of a confidential client application. In this system the coolant flows around or through key component 'X', indicated with the grey rectangle, at high velocities. The main risk is cavitation, which is the spontaneous transition of a liquid to its gaseous phase due to a drop of pressure below the vapor pressure. Downstream of X, cavitation is a significant risk, which can lead to damage and limitations in flow rate, known as choked flow.

An insert will be added below the risk area to prevent cavitation at any time, and secure the cooling flow.

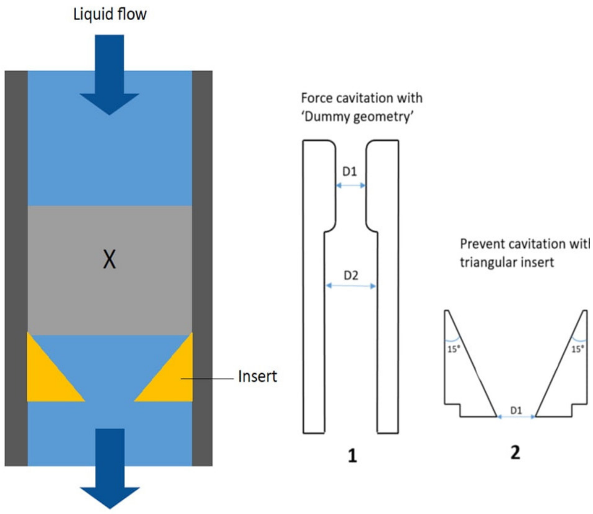

Figure 1 An abstract representation of the observed system

Approach

The design of such an insert is not trivial as it has to meet the following requirements: It must prevent cavitation near region X, it must not cause cavitation itself, it must allow the cooling flow to continue and it must be (relatively) cost effective to manufacture.

Preventing cavitation can be achieved by increasing the downstream pressure of the system. A well designed converging insert increases the pressure upstream of the insert, removing the risk of cavitation near the important parts of the system. To prevent cavitation and choked flow caused by the insert itself, the design should theoretically not have pressure decreases near a wall, but instead in the fluid layer. This will allow for a circulation of the cooling liquid, preventing a local pressure decrease. The red circles in the figure mark the risk area for a poorly designed insert, in which the point with the lowest static pressure is close to the wall. If this point is located in the liquid behind the nozzle, the low pressure can be prevented through circulation of the liquid.

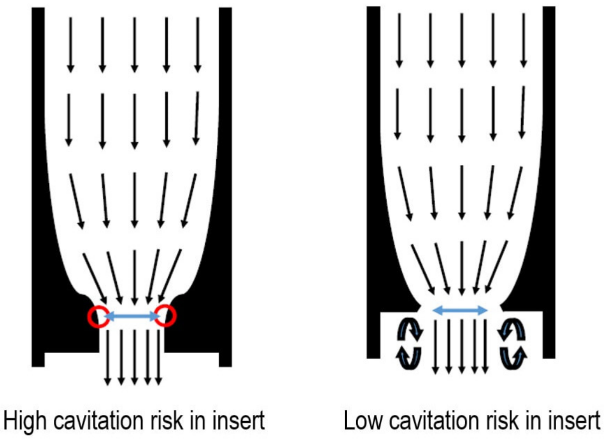

Figure 2 Different designs showing potential cavitation zones

To validate this concept, two insert designs are taken and compared numerically as well as experimentally.

The first insert design causes cavitation quickly and can act as a dummy geometry (Left). The second design prevents cavitation as described (Right). For both designs, a CFD simulation was setup and compared to an representative experiment.

The numerical results show cavitation in the liquid flow for a pressure difference of 3 bar when only insert 1 is used. No cavitation occurred when insert 2 was added below insert 1 using the same pressure difference. The colors show the volume fraction of the gaseous phase, with the red region being only gas and the blue region only coolant.

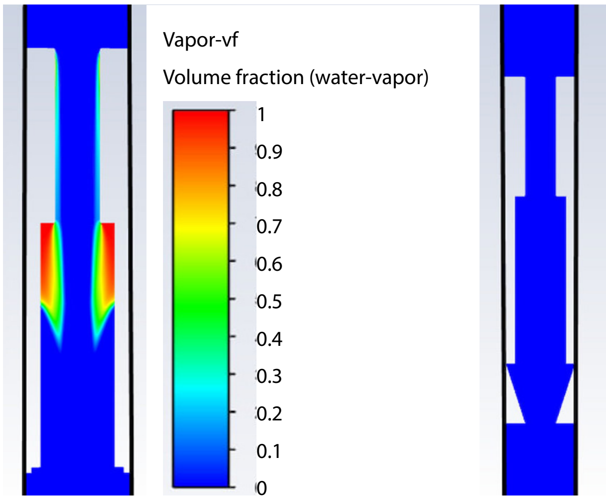

Figure 3 CFD simulation of two insert designs

Results

The moment cavitation occurs, the flow rate becomes independent of the downstream pressure. So further lowering the downstream pressure results in a constant flow rate.

The graph shows the results for the experiment as well as the numerical model results. Here, insert 1 shows cavitation at low , and insert 2 prevents cavitation when it is added below insert 1, up to a very high pressure difference. The difference in maximum flow rate between the experiment and simulations is explained by the experimental imperfections not included in the simulation.

In a system where the downstream pressure is limited by other requirements, adding a converging inserts can prove to be useful in preventing cavitation. Choked flow seems unavoidable at extreme pressure differences, even with such a design. It is therefore important to choose appropriate up and downstream pressures.

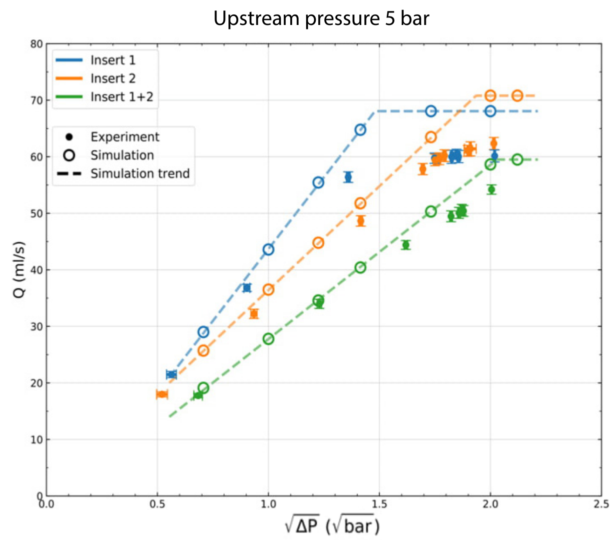

Figure 4 Results for the experiment as well as the numerical model