

Flow Analysis and Linearization of Rectangular Butterfly Valve Flow Control Device for Liquid Cooling

Rajesh Kasukurthy, Pencala S Challa, Rishi Ruben Palanikumar, Barath Ragul Manimaran,
Dereje Agonafer
IEEE itherm 2018

The University of Texas at Arlington,
P.O. Box 19023

Arlington, Texas, USA, 76010

Email: rajesh.kasukurthy@mavs.uta.edu

Email: penchalasumanth.challa@mavs.uta.edu

Email: rishiruben.palanikumar@mavs.uta.edu

Email: barathragul.manimaran@mavs.uta.edu

Email: agonafer@uta.edu

ABSTRACT

One of the key areas in which electronic cooling research has been focusing on, is addressing the issue of non-uniform at package level. This challenge has incited the use of numerous temperature sensing mechanisms for dynamic cooling of electronic components. What dynamic liquid cooling effectively does is, use feedback from sensors as inputs for the pumps, supplying more amounts of fluid to parts of the electronics that is warmer while supplying minimal fluid to the parts of the electronics that are relatively cooler. A novel approach to address uneven heating in a liquid cooled system is the use of a temperature sensing flow control device that can control flow rate based on temperature. The necessity of numerous temperature and pressure sensors, a suitable control system and the maintenance and reliability issues that they present, can be significantly minimized with the use of a self sensing and controlling flow control device. This paper looks at the flow analysis of a self-regulating flow control device (FCD) designed for electronic module for data center application. An axially rotating butterfly valve is used to regulate the flow rate of FCD. Linearization of the flow with respect to damper angle is studied by modifying the dimensional ratios of the rectangular cross section of the FCD. Pressure drop, and flow rate characterization is done for the FCD.

KEY WORDS: Damper analysis, CFD, Mass flow rate linearization.

NOMENCLATURE

$r = L/B$

where, L is Length, mm

B is Breadth, mm

u Velocity, ms^{-1}

f Force, N

E Rate of Deformation

Greek symbols

ρ Mass density (kg/m^3)

μ Eddy Viscosity

Subscripts

a Ambient

i,j Directional vectors

b Body

INTRODUCTION

Data center technology has been improving rapidly with every passing year. The need for data storage and processing has increased tremendously which has triggered the need for energy efficient data centers. Trends of increasing microprocessor power densities has caused increased temperatures and non-uniform temperature distribution (hot-spots) at the module level. Reduction of the operating temperature is essential to minimize the static power consumption due to leakage and to reduce failure rates of microprocessors. Also, performance and reliability of device increases, and total power consumption decreases with the reduction of operating temperature. This has made creating effective cooling systems for high power devices an absolute necessity for reliable operation. Servers are most efficient when they operate near maximum utilization [1]. This concept governs the design ideology of currently existing conventional cooling solutions which aims to provide high power cooling and selective distribution of cooling. Unfortunately, this has often resulted in increased cost of cooling devices. Therefore, designing an energy efficient cooling system with selective distribution of coolant at the chip level is the challenge to be addressed.

Integration of control devices and schemes within a conventional cooling system to enable targeted delivery of coolant is one way to address the aforementioned challenges. Fernandes have suggested liquid cooling to be the most economical way to cool high power density packages [1]. The existing conventional liquid cooling solution uses a pump for each dynamic cold-plate (DCP) section [5]. Having a self-regulated FCD for each section will be more efficient and economical than using the pump for the targeted coolant delivery. A single pump can be used for the entire DCP with dynamic cooling, hence reducing the overall pumping power for cooling the rack. The flow control device is designed to react based on coolant temperature. This means that the perfect position for retrofitting the FCD would be as close to the section outlet as possible.

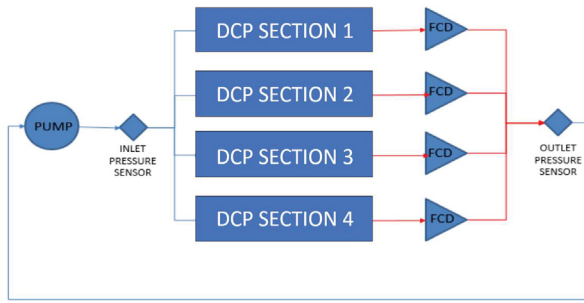


Fig. 1 Dynamic cold plate schematic

Design Considerations

In order to most accurately emulate a typical processor a 160W heat source is considered as the test module. A 160 W module is chosen just to show that the concept works. Experimentally testing a 160W module was much more feasible than designing a high-power density module. It is to be noted that higher power will only further improve the performance of the FCD. The heat dissipation calculations were done based on the assumption that the thermal resistance offered by the cold plate is negligible. We assumed a ΔT of 10 degrees across the module for convenience. A hydraulic diameter of 10 mm is considered, which provides optimum velocity for our flow control device and also is adaptable over a range of heat sources. This design choice also keeps pressure drop due to losses minimal at various locations.

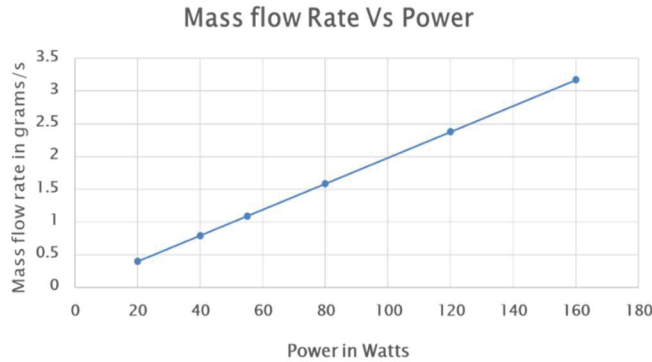


Fig. 2 Variation of mass flow rate in Kg/s with respect to power in watts.

Required hydraulic diameter is calculated for a fixed velocity and temperature difference for a range of power dissipations from 20W to 160W. It can be observed that there is an almost linear increase in hydraulic diameter from 3mm to 8mm with respect to power. Based on this deduction we designed a flow control device with butterfly valve to have variation in the flow rate for different power dissipations.

Valve design

Valve designing is done considering some factors that affect the overall system efficiency like pressure drop across the device, flow with respect to angle change and flow rate range. The traditional circular butterfly valve design will meet most of these requirement except for the flow rate variation with respect

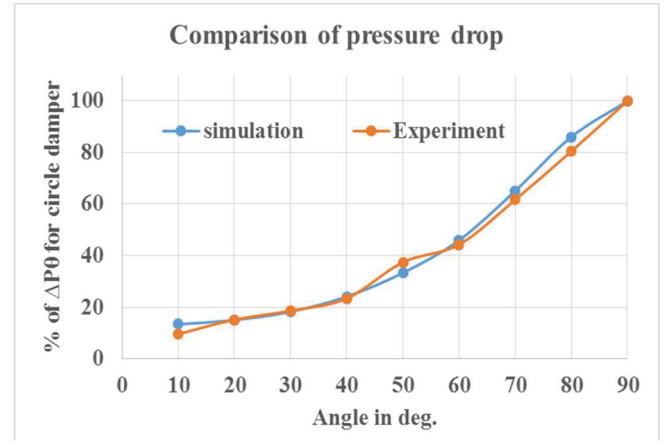


Fig. 3 Plot between pressure drop variation in percentage and damper angle, comparing experimental and simulated results.

to the damper angle, so to address this issue we changed the cross section of the damper from circular to rectangular. In rectangular cross section the length to breadth ratios are varied to find an optimized design.

A circular FCD is designed with the required hydraulic diameter to validate with available experimental data. As shown in the Fig. 2 below the percentage of pressure drop verses angle is compared between the simulation values and the experimental data taken from the thesis paper of Del Toro, A [2]. The maximum pressure value at 10 degrees is take as reference to calculate the percentage of pressure drop. These values are very close the maximum error is 3% at 20 degrees.

Circular vs Rectangular cross sections

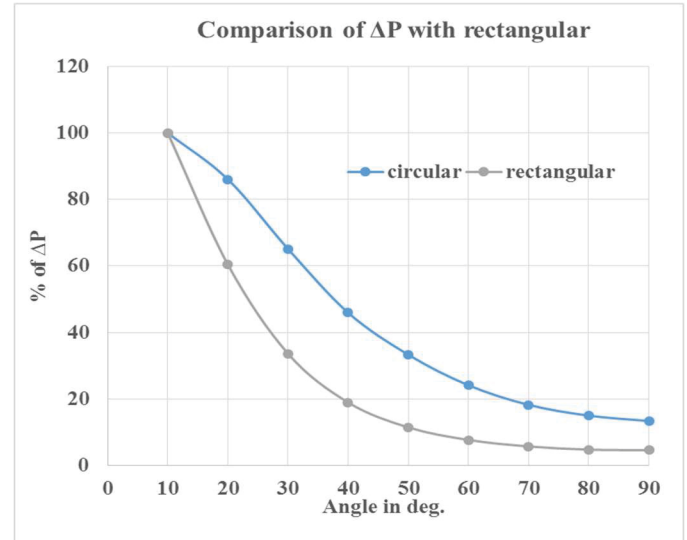


Fig. 4 Plot between pressure drop variation in percentage and damper angle, comparing circular and rectangular cross sections.

Now we compare the circular cross section with the rectangular one to compare the change in the pressure drop with respect to the angle. The Fig. 3 below shows the

pressure drop for a rectangular when compared to the circular cross section. We can observe that pressure losses are less in rectangular when compared with circular damper.

Another useful comparison to help illustrate the advantage of using a rectangular cross section is given below. We compare the mass flow rate of both cross section shapes for various damper angles. The rectangular design provides a wider range of flow rates, allowing much lower flow rates at angles nearing completely closed position. Hence the rectangular cross section allows us for use of the flow control device over a wider range of temperatures. Also the linear trend is much more consistent with the rectangular design especially in angles nearing closed position when compared to the circular design which tends to saturate at these angles.

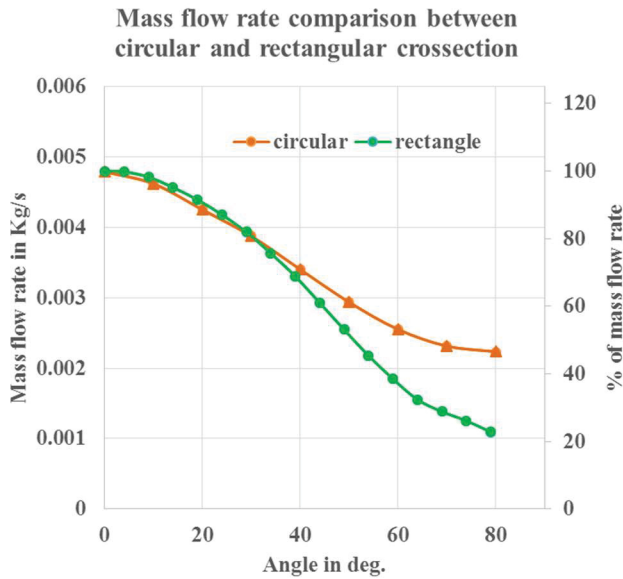


Fig. 5 Plot between the mass flow rate and the damper angle for circular and rectangular cross sections.

Design of flow control device

The flow control device consists of circular inlet and outlet. When we move closer towards the damper, the circular cross section tapers into a rectangular cross section. As discussed in the previous section a rectangular cross section at the damper allows us to have more control over the linearization of the flow, when compared to a circular cross section.

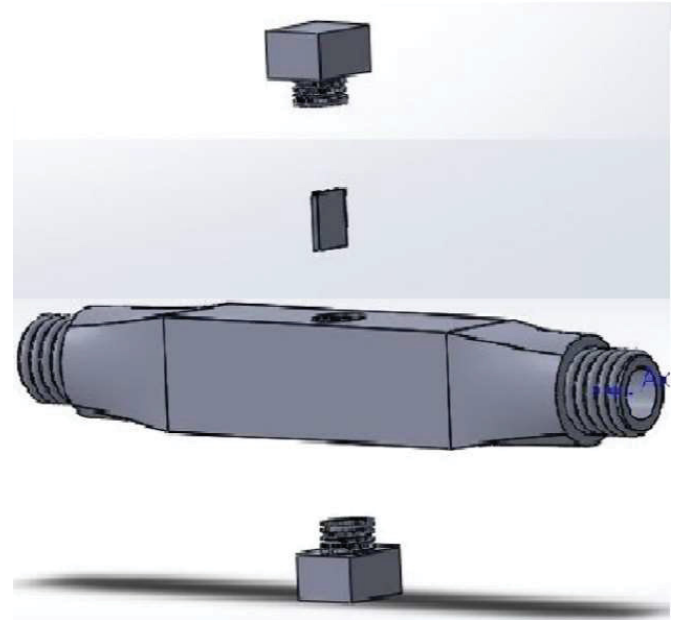


Fig. 6 Illustration of flow control device design.

The damper is controlled by an active material that is sensitive to temperature. In order to fit the flow control device in the server, the overall dimension of the FCD are kept very small, with the cross sections in the order of 15mm and the entire length of the device around 80mm.

Meshing

To solve the Fluid model in CFD, the continuous fluid medium has to be partitioned into discrete volumetric cells vertices or nodes and elements. This process is called meshing. For any study using CFD solver, fluid properties like velocity, temperature, pressure etc. are calculated at every volumetric cell and are integrated and averaged to get a total value of all the properties for whole fluid [4]. Increase in the number of volumetric cells results in more refined results which imitate real life experimental results. In the present problem, CFX fine meshing is used with slow transition and a fine span angle center. The cell geometry features a minimum size of 2.5E-2 mm, max face size and max Tet size of 5mm. Inflation is used for fluid around the damper to capture highly refined flow features around the valve. Inflation is a meshing property which creates a fine and slowly growing mesh at a surface to capture more accurate fluid properties around that surface.

Mesh sensitivity analysis is done comparing the mass flow rate for different mesh size. We observed there is no significant change in the mass flow rate when mesh density goes above 50 elements per mm³. The chosen cell geometry of 2.5E-2 mm results in much higher element density than the above mentioned threshold and provides highly accurate results.

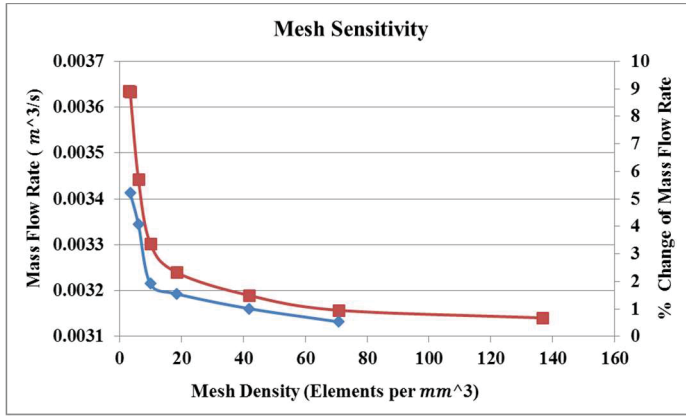


Fig. 7 Mass flow rate vs Mesh density and percentage change in mass flow rate vs Mesh Density.

Computational Fluid Dynamics

The equations that govern the motion of a Newtonian fluid are the continuity equation, the Navier-Stokes equations, the momentum equation and the energy equation. The set of equations listed below represents seven equations that are to be satisfied by seven unknowns [3]. Each of the continuity, energy, and momentum equations supplies one scalar equation, while the Navier-Stokes equations supply three scalar equations. The seven unknowns are the pressure, density, internal energy, temperature, and velocity components. The scope of our analysis predominantly lies in the laminar region of flow owing to low flow velocities and relatively simple geometries. The only instance where a k-Epsilon turbulence model is used is in the region between when the damper is completely closed and when it is open by an angle of 10 degrees. The equations used for solving in the laminar as well as turbulent regions are listed below.

$$\text{Continuity : } \frac{\partial \rho}{\partial t} + \vec{\nabla} \cdot (\rho \vec{u}) = 0$$

$$\text{Momentum : } \rho \frac{D\vec{u}}{Dt} = -\vec{\nabla} p + \nabla(\mu \nabla \vec{u}) + \vec{f}_b$$

$$\text{Energy : } \rho \frac{DE}{Dt} = -\vec{\nabla} \cdot (p\vec{u}) + \vec{\nabla} \cdot (k_t \cdot \vec{\nabla}(T)) + \Phi + S_E$$

$$\begin{aligned} \text{Turbulent kinetic energy : } & \frac{\partial(\rho k)}{\partial t} + \vec{\nabla} \cdot (\rho k \vec{u}) \\ &= \vec{\nabla} \cdot [\alpha_k(\mu + \mu_t) \cdot \vec{\nabla}(k)] + 2\mu_t E_{ij} \\ & \quad \cdot E_{ij} - \rho \varepsilon \end{aligned}$$

$$\begin{aligned} \text{Turbulent dissipation : } & \frac{\partial(\rho \varepsilon)}{\partial t} + \vec{\nabla} \cdot (\rho \varepsilon \vec{u}) \\ &= \vec{\nabla} \cdot [\alpha_\varepsilon(\mu + \mu_t) \cdot \vec{\nabla}(\varepsilon)] + C_{1\varepsilon} \frac{\varepsilon}{k} 2\mu_t E_{ij} \\ & \quad \cdot E_{ij} - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \end{aligned}$$

where

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}, \quad C_\mu = 0.0845, \quad \alpha_k = \alpha_\varepsilon = 1.39,$$

$$C_{1\varepsilon} = 1.42, \quad C_{2\varepsilon} = 1.68$$

and

$$C_{1\varepsilon}^* = C_{1\varepsilon} - \frac{\eta(1 - \frac{\eta}{\eta_0})}{1 + \beta\eta^3}, \quad \eta = (2E_{ij} \cdot E_{ij})^{1/2} \frac{k}{\varepsilon},$$

$$\eta_0 = 4.377, \quad \beta = 0.012$$

Results

The damper ratios were varied parametrically on ANSYS fluent to obtain results for several other variables like pressure drop and mass flow rate. The height to width ratio of the damper cross section (damper ratio, r) is varied between 0.8 and 1.5 to determine the design which will provide linear change in flow with change in damper angle. Fig. 6 shows the plot between mass flow rate and damper angle for various damper ratios(r). The x axis shows the damper angle with 0 degrees indicating completely open position and 90 degrees indicating completely closed position; the y axis shows the mass flow rate in kg/s. We designed this flow control device to have a small non-zero flow rate even when the damper is completely closed.

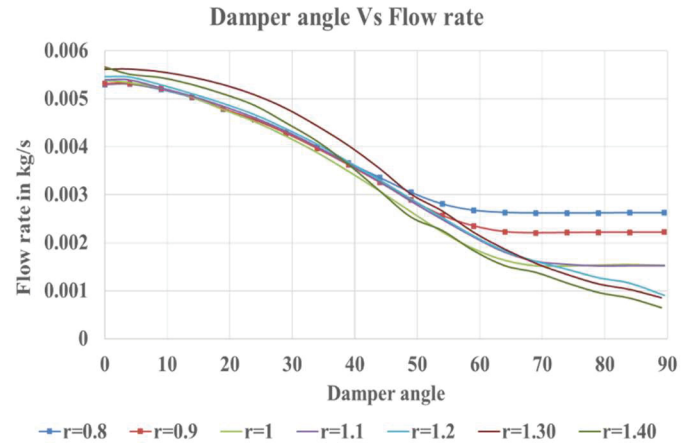


Fig. 8 Plot between the mass flow rate and the damper angle for various r ratios.

Based on the flow rate required at idle condition of the server, we are able to modify above mentioned minimal flow at closed damper position to meet that requirement. From the graph we can observe that for r=0.8 the range of flow rates is very limited. When we increase r from 0.8 through 1.5 we observe increment in the flow rate range, saturating at r=1.2 above which there is no pronounced increment in flow rate range. We choose 1.2 as the optimum value for r, as higher r values tend to produce greater torque on the damper.

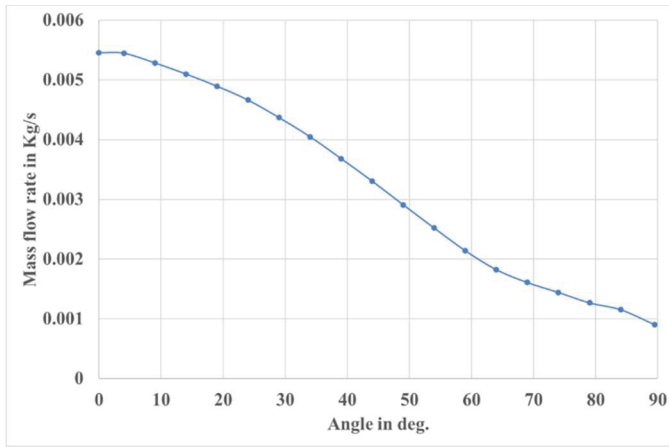


Fig. 9 Plot between mass flow rate and damper angle for $r=1.2$.

Fig. 8 shows the plot between mass flow rate and damper angle for $r=1.2$. We observe that the flow rate varies most linearly with change in damper angle compared to other ratios. Fig. 9 indicates the change in Reynolds number with respect to change in damper angle.

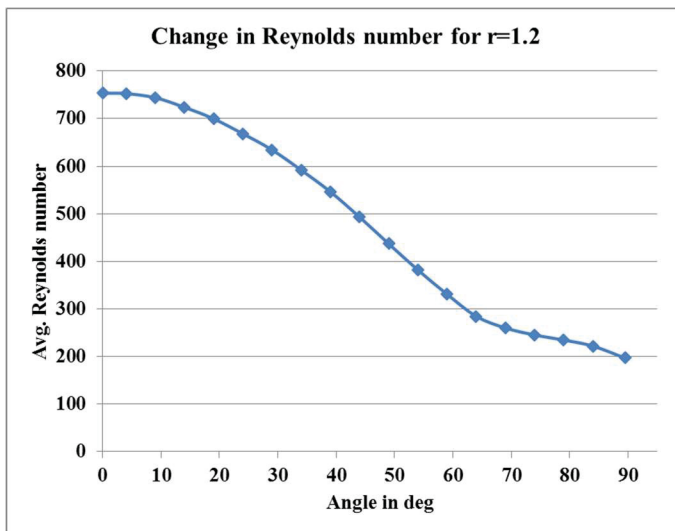


Fig. 10 Plot between Reynolds number and damper angle for $r=1.2$.

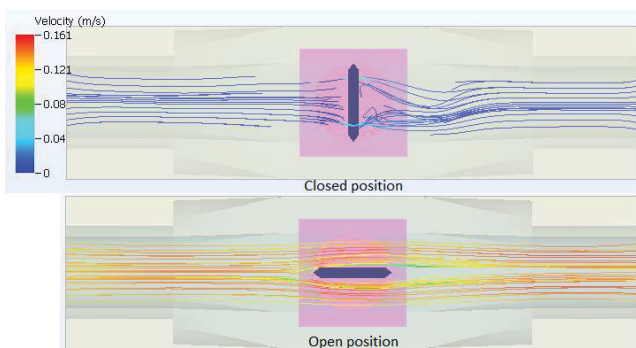


Fig. 11 Cross sectional view of the FCD at closed and open position

Conclusion

The dynamic cold plate works better with an array of smart flow control devices, which effectively does the job of the combination of a temperature sensor, control system and an actuator. In our flow control device, we use an active material which rotates the damper with respect to change in temperature. To have better control over flow rate, the rectangular butterfly valve design is better than the traditional circular butterfly valve because it varies flow rate more linearly with change of angle. Through CFD analysis we can conclude that a damper ratio of 1.2 is good in linearizing the flow. This can also possibly find applications in different fields apart from data center water cooling. Future work can be studying the hydrodynamic forces on the damper and validation.

Future work

Testing the FCD for its reliability and do failure analysis. Calculate the response time of the FCD. Experimental analysis of the dynamic cooling concept.

Acknowledgement

Immeasurable appreciation and deepest gratitude for the help and support are extended to everyone who have contributed in making this study possible.

Deserving of special mention are ANSYS without whose resources this project would not be possible.

This work is supported by NSF IUCRC Award No. IIP-1738811.

References

- [1]. Fernandes, J. E. (2015). Minimizing Power Consumption at Module, Server and Rack-Levels within a Data Center through Design and Energy-Efficient Operation of Dynamic Cooling Solutions.
- [2]. Del Toro, A. (2013). *Computational fluid dynamics analysis of butterfly valve performance factors*. Utah State University.
- [3]. Currie, I. G. (2012). *Fundamental mechanics of fluids*. CRC Press.
- [4] Manual, F. L. U. E. N. T. (2017). ANSYS/FLUENT Release Version 18 Ansys Inc
- [5] Chowdhury U, Sahini M, Siddarth A, Agonafer D, Branton S. Characterization of an Isolated Hybrid Cooled Server With Failure Scenarios Using Warm Water Cooling. ASME. International Electronic Packaging Technical Conference and Exhibition, ASME 2017 International Technical Conference and Exhibition on Packaging and Integration of Electronic and Photonic Microsystems (IPEP):V001T02A002. doi:10.1115/IPACK2017-74028.