

BULETINUL INSTITUTULUI POLITEHNIC DIN IAȘI  
Publicat de  
Universitatea Tehnică „Gheorghe Asachi” din Iași  
Volumul 66 (70), Numărul 1, 2020  
Secția  
CONSTRUCȚII DE MAȘINI

## TURBULENCE ENHANCEMENT TECHNIQUES IN COLD PLATES

BY

GEORGE-GABRIEL CHIRIAC\* and CĂTĂLIN GABRIEL DUMITRAȘ

“Gheorghe Asachi” Technical University of Iași, Romania,  
Department of Machine Tools

Received: February 14, 2020

Accepted for publication: March 28, 2020

**Abstract.** Overview of the methods used to increase the turbulence of the fluid flow in cold plates. Different approaches in improving the thermal transfer between the cold plate or radiator and the fluid are presented. Introduction in thermal transfer paths from the heat source to the cooling fluid, fluid flow distribution inside the cold plate and multiple ways to enhance the turbulence inside the water flow. Each method is presented and evaluated to see the advantages and disadvantages of each one.

The method presented as water flow distribution inside the cold plate is to split the inside volume in multiple channels and use numerical simulation to evaluate the distribution and maximum temperature. Turbulence enhancement geometries presented are helix lamellas inserted in tubes, winglet pair and dimples and protrusions.

**Keywords:** Heat transfer; turbulence enhancement; cold plate; pins; fins.

### 1. Introduction

Due to automotive industry expansion and innovation, car manufacturers and suppliers compete against each other to offer innovative and

---

\*Corresponding author; *e-mail*: chiriageorgegabriel@yahoo.com

modern products. To stay competitive on the market, automotive suppliers and manufacturers have to continue upgrade their products or develop new technologies.

Most of the modern cars are equipped with electronic devices. Electrical cars are also equipped with additional batteries, charging and battery management systems. All the electronic devices, during functioning, produce heat due to Joule heating.

In order to keep all the electronic components functioning, cooling systems need to be added. One of the most efficient cooling methods used inside cars are cold plates fluid cooling.

Due to many constraints in designing and manufacturing the cold plates, as allocated space, price, manufacturing technology, total power that needs to be dissipated etc., cold plates need to be more efficient than ever.

The main purpose of a cold plate is to transfer the heat from the heat source to the fluid and then outside the cold plate due to fluid movement. This paper has as an objective to present the way in which the heat is transferred to the fluid and the methods used to improve and increase the heat quantity transferred from the heat source to the cooling fluid.

## 2. Synthetic Analysis of the Existing Solutions

In order to improve the quantity of heat transferred from the heat source to the fluid and outside the cold plate, the overall heat transfer coefficient between the heat source and fluid should be taken into consideration.

In cold plates the heat transferred from the heat source to the fluid encounters three heat resistance paths. The heat from the hot fluid, inside the component that needs cooling, is transferred to the metal wall, which is the cold plate body. The second thermal resistance is encountered inside the metal wall. This thermal resistance is highly influenced by the material used to manufacture the cold plate. The last thermal resistance can be found at the border between the cold plate metal body and the cooling fluid, which has a lower temperature than the hot fluid.

In Fig. 1 the heat path for a cold plate can be observed, where:

$L$  = Thickness of the metal wall,

$k$  = Thermal conductivity of the wall material,

$t_1$  = Temperature of the surface-1,

$t_2$  = Temperature of the surface-2,

$t_{hf}$  = Temperature of the hot fluid,

$t_{cf}$  = Temperature of the cold fluid,

$h_{hf}$  = Heat transfer coefficient from hot fluid to metal surface, and

$h_{cf}$  = Heat transfer coefficient from metal surface to cold fluid.

(The suffices hf and cf stand for hot fluid and cold fluid respectively.)

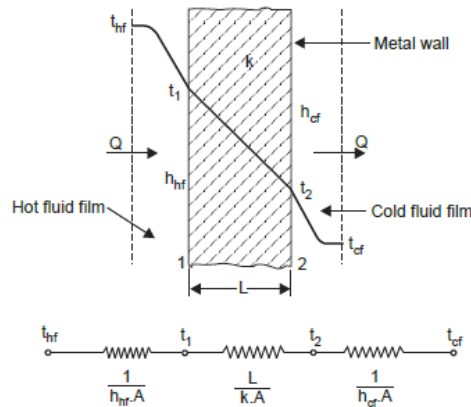


Fig. 1 – Heat transfer path through a plane wall (Rajput, 2007).

In most of the automotive applications, the designer of the cold plate needs to evaluate many criterias before creating the most efficient product. The material used for the cold plate is most of the times chossed based on manufacturing technology, price or strangth requirements. The cooling fluid type, temperature and the volume flow rate of it is also a requirement from the car manufacturer or is the same for the entire car. In this case, not many variables can be changed in order to improve the total heat transfer between the heat source and cooling fluid. This paper is focusing on the last thermal resistance of the conduction path, at the conduction and phenomena inside the cold fild layer of the cooling fluid. Because the designer has to design the shape of the cold plate, this paper shows what shapes of the cold plate were used before in the industry and the impact of the shape on the cooling fluid flow.

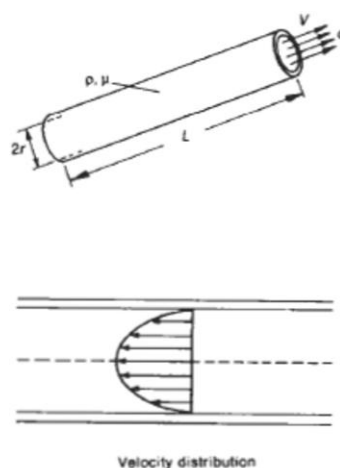


Fig. 2 – Velocity distribution throught a pipe (Marghitu, 2001).

When the fluid has a laminar flow inside a pipe or between two plates, the maximum speed of the fluid is in the center of the pipe or between the plates and the fluid speed next to the walls is decreasing towards zero velocity. The fluid velocity distribution can be seen in Fig. 2. Because the walls of the cold plate have a higher temperature than the cooling fluid, it means that the fluid with a high velocity has a smaller temperature compared with the fluid with a lower velocity.

If the temperature difference between the cold plate wall and the cooling fluid is increased ( $t_2$  and  $t_{cf}$  in Fig. 1), more heat is transferred into the cooling fluid. Considering this fact and the fluid velocity distribution mentioned above, the conclusion is that the fluid should be mixed inside the cold plate.

Mixing the fluid inside the cold plate means that the cooling fluid temperature is averaged if we consider one section of the cold plate. Changing the fluid flow from a laminar to a turbulent flow will reduce the layer of fluid with a small velocity and high temperature that can be found next to the cold plate walls. In this way, the heat transfer between the heat source and fluid is improved, only by changing the internal shape of the cold plate and fluid flow type.

The downside is, that with the increase of turbulences inside the fluid, the force needed to keep a steady and continuous flow increases. So the pressure drop is increasing with the increase of turbulence level inside the fluid flow.

Various types of turbulence enhancement techniques were used inside the cold plates or other components with fluid flows inside and many studies were made to see the effect that these geometries have on thermal transfer.

Chen *et al.* (2020) show in their paper called *Multi-parameter structure design of parallel mini-channel cold plate for battery thermal management* how important is the cold plate design in optimizing a good fluid distribution inside.

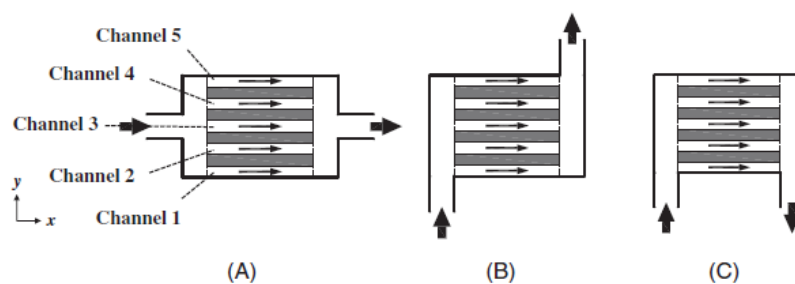


Fig. 3 – Inlet, outlet and channel distribution in a cold plate (Chen *et al.*, 2020).

Three geometries were used by the authors to start the investigation using numerical simulations. All three geometries have five channels inside for the water circulation, the main difference between them is the inlet and outlet pipe for the cooling fluid.

The authors showed that the numerical simulation methods are a very fast and reliable way to find the geometry with the best flow distribution and the best heat transfer. Their approach was also to change one parameter and observe the difference that occurs between each iteration. Even if the geometry is not made to enhance the turbulence inside, finding the best starting geometry for each use case using finite element analysis is a great approach in cold plate design.

Sheikholeslami *et al.* (2020) concluded in their paper named *Nanomaterial thermal performance within a pipe in presence of turbulator* that the tube equipped with a turbulator creates a different boundary layer style.

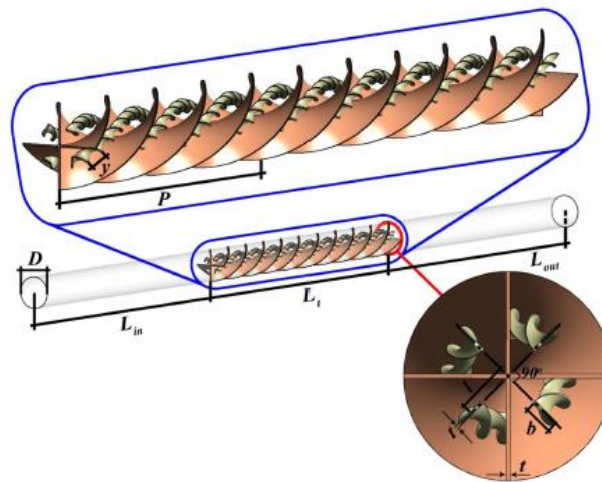


Fig. 4 – Style of turbulator (Sheikholeslami *et al.*, 2020).

In Fig. 4 the helix shaped turbulator used by the authors to investigate the enhancement of thermal transfer can be seen. The evaluation of turbulence and thermal transfer were evaluated and compared based on Reynolds and Nusselt numbers.

Based on the results of the numerical simulations, the authors saw that the inserted turbulator changed the velocity of the fluid near the wall. The velocity near the wall is enhanced and disrupts the boundary layer next to the cylinder wall. As a result, the fluid mixing is improved and the convective coefficient is increased, therefore the cooling is more efficient. In other words, increasing the turbulence level inside the system will improve the thermal transfer between the solid and fluid.

With the increase of turbulence inside the pipe, an higher pressure drop was observed in the numerical simulations. In order to find the best solution for the designer, both factors, thermal behaviour and pressure drop, should be taken

into consideration. Based on the product requirement, the best design can be then chosen.

This type of turbulence engancement geomtry was investigated only for a cylindrical tube. It's efficiency in cold plates, where the duct section may not be a circle, should be investigated. In the numerical simulations, the turbulator is not fixed inside, but in reality, the turbulator needs fixation inside the tube, to prevent movind. The fixation should be taken into consideration, because it may influence the fluid flow.

For very thin cold plates or small sized coolers, the helix geometry may not be as efficient as is the presented design. In thin fluid channels, the helix geomtry may significantly the volume left for the fluid to fill in, which may impact the transfer of heat outside the system. Numerical simulations is in this situations highly recommended the evaluate the improvement on the thermal transfer that this design is adding to the system.

For mass production, this design has some disadvantages too. The total cost of the cooling system is increased with the addition of another component and additional assembly steps need to be performed, to insert and fix the turbulator inside the fluid channel.

Esmaeilzadeh *et al.* (2017) studied the thermal behavior and fluid flow characteristics in a compact flat channel heat exchanger. Numerical simulation was used to evaluate the effect of using a trapezoidal winglet pair (TWP) and a curved trapezoidal winglet pair (CTWP), as can be seen in Fig. 5.

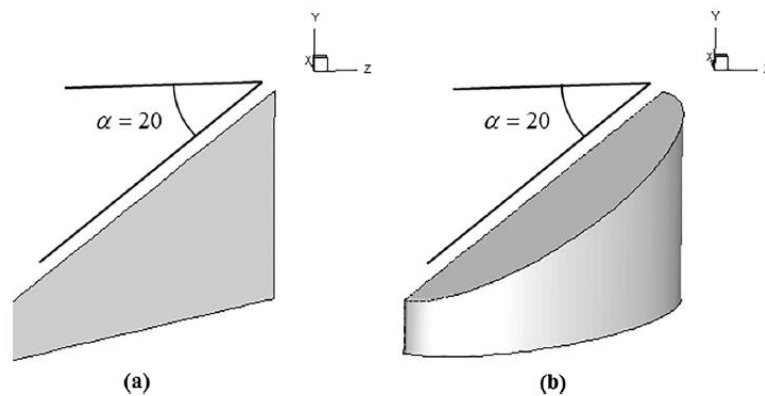


Fig. 5 – (a) trapezoidal winglet; (b) curved trapezoidal winglet (Esmaeilzadeh *et al.*, 2017).

In Fig. 6 can be observed the symmetrical position of the winglet pair inside the channel and the dimensions used for the computational domain. Because the model has a symmetry line, the numerical simulation was performed as a symmetrical model in order to reduce the computational time.

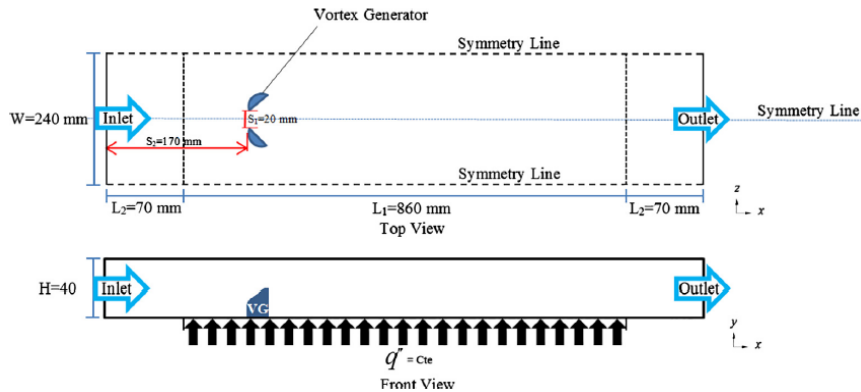


Fig. 6 – Schematic view of the channel (Esmailzadeh *et al.*, 2017).

For both TWP and CTWP improvements in turbulence enhancement and heat transfer is observed. After evaluating the heat transfer and pressure drop, the authors concluded that the CTWP has a better overall efficiency.

The thermal transfer was improved when the boundary layer thickness was reduced due to increase in turbulence and fluid flow, which created a higher temperature gradient between the fluid and cold plate walls.

In order to obtain the best thermal and hydrodynamic performance, the authors suggested to use numerical simulation to evaluate the efficiency of each shape and to find the best-case scenario for the desired application.

Xie *et al* (2015) used numerical simulations to investigate the fluid flow and heat transfer performance of teardrop dimple and protrusion with different eccentricities. The results were compared with the results obtain when using hemispherical dimple and protrusion and turbulent enhancement techniques.

The five structures used for the study can be observed in the Fig. 7. These structures include hemispherical dimple (a), hemispherical protrusion (b), teardrop dimple (c), teardrop protrusion with negative (d) and positive (e) eccentricity. Based on the former study, only the teardrop dimple with positive eccentricity was used in the present study, because it shows a better performance compared with the teardrop dimple with negative eccentricity.

In order to find the best shape of the teardrop protrusion, for both positive and negative eccentricity, eight shapes were studied using a ratio  $e/Dh = \pm 0.1, \pm 0.2, \pm 0.3, \pm 0.4$ .

In the computational domain, the structures are arranged in line arrays with a spacing between them equal with the width of the channel. When the flow is reaching the first structure, it is considered fully developed.

The authors concluded that the fluid flow covers the surface of the teardrop easily and hits the rear section of the teardrop dimple/protrusion with PE with higher energy compared with hemispherical dimple/protrusion.

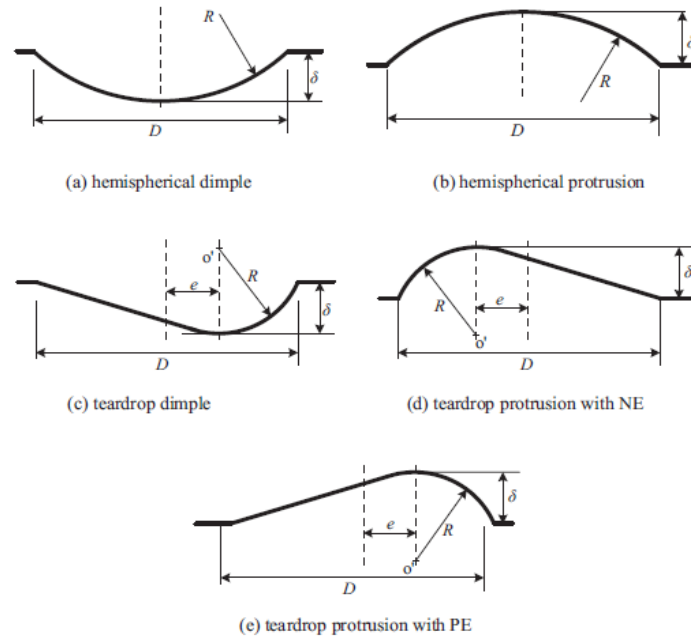


Fig. 7 – Geometric parameters of dimples and protrusion (flow direction from left to right) (Xie *et al.*, 2015).

In this study, the authors showed that the thermal performance is changing based on the dimple or protrusion geometry for the given scenario. This turbulence enhancement method has the advantage that is build in the cold plate geometry. No additional components or assembly procedures are required the use the mentioned turbulence enhancement geometries. For die cast cold plates, the geometry can be designed with the dimples or protrusions, while for other manufacturing methods additional machining may be required.

While the additional material used for the protrusion will increase the mechanical strength of the cold plate, the dimples may decrease the mechanical performance of the cold plate, if the parts are exposed to high forces of high fluid pressure inside the fluid channels.

### 3. Conclusions

Turbulence enhancement geometries inserted or build in cold plates have an improvement on changing the fluid flow and increasing the Reynolds number.

A fluid flow with a higher turbulence level will decrease of break the boundary layer found next to the walls and a higher temperature gradient between the fluid and solid can be generated.



Numerical simulations can be used with success in finding the best geometry of the turbulence enhancement structures based on the application and the cooling system requirements, such as dimensions, manufacturing method, fluid flow parameters, dissipated power etc.

In order to find the geometry with the best performance, thermal transfer and pressure drop should be evaluated together, as the best thermal transfer may be found on the systems with the highest pressure

For the numerical simulations, grid independency simulations should be performed in order to reduce the results error caused by inappropriate mesh network dimensions.

**Acknowledgements.** This paper is the result of a study that will be included in the doctoral work developed at the “Gheorghe Asachi” Technical University of Iași.

#### REFERENCES

- Chen K., Chen Y., Song M., Wang S., *Multi-Parameter Structure Design of Parallel Mini-Channel Cold Plate for Battery Thermal Management*, International Journal of Energy Research, 44, 4321-4334 (2020).
- Esmailzadeh A., Amanifard N., Deylami H.M., *Comparison of Simple and Curved Trapezoidal Longitudinal Vortex Generators for Optimum Flow Characteristics and Heat Transfer Augmentation in a Heat Exchanger*, Applied Thermal Engineering, 125, 1414-1425 (2017).
- Marghitu D., *Mechanical Engineer's Handbook*, California: Academic Press, 2001.
- Rajput R.K., *Engineering Thermodynamics*, Third Edition, New Delhi: Laxmi Publications Ltd., 2007.
- Sheikholeslami M., Jafaryar M., Sheremet M.A., Shafee A., Babazadeh H., *Nanomaterial Thermal Performance within a Pipe in Presence of Turbulator*, Applied Nanoscience (2020).
- Xie Y., Qu H., Zhang D., *Numerical Investigation of Flow and Heat Transfer in Rectangular Channel with Teardrop Dimple/Protrusion*, International Journal of Heat and Mass Transfer, 84, 486-496 (2015).

#### TEHNICI DE ÎMBUNĂȚĂȚIRE A TURBULENȚELOR ÎN PLĂCILE DE RĂCIRE

(Rezumat)

În lucrarea de față se realizează o prezentare generală a metodelor utilizate pentru a crește turbulența fluxului de fluid în plăcile de răcire. Sunt analizate diferite abordări în îmbunătățirea transferului termic între placa rece sau radiator și fluid. Se face o introducere privind căile de transfer termic de la sursa de căldură la fluidul de

răcire, distribuția fluxului de fluid în interiorul plăcii reci și modalități multiple de a spori turbulența în fluxul de apă. Fiecare metodă este prezentată și evaluată pentru a vedea avantajele și dezavantajele fiecăreia.

Metoda prezentată ca distribuție a debitului de apă în interiorul plăcii reci este de a împărți volumul interior în mai multe canale și de a utiliza simularea numerică pentru a evalua distribuția și temperatura maximă. Geometriile de îmbunătățire a turbulenței prezentate sunt lamele elicoidale inserate în tuburi, pereche de aripi și gropițe și proeminențe.