CZECH TECHNICAL UNIVERSITY IN PRAGUE

FACULTY OF MECHANICAL ENGINEERING

MASTER`S THESIS

2021

KAAN EGE TEMİZ

MASTER'S THESIS ASSIGNMENT

I. Personal and study details

Student's name:	**Temiz Kaan Ege**	Personal ID number:	**490972**
Faculty / Institute:	**Faculty of Mechanical Engineering**		
Department / Institute:	**Department of Process Engineering**		
Study program:	**Mechanical Engineering**		
Branch of study:	**Process Engineering**		

II. Master's thesis details

Master's thesis title in English:

Optimization of heat transfer between fluid stream and heat transfer surface

Master's thesis title in Czech:

Optimalizace přenosu tepla mezi proudem tekutiny a teplosměnnou plochou

Guidelines:

- Make a literature research concerning possible enhancement of heat transfer from a fluid stream towards the heat transfer surface.
- Create models in ANSYS Fluent describing two basic cases with flow perpendicular or parallel to the heat trasfer surface.
- Make comparisons of heat tranfer rates on modified heat transfer surfaces and assess their impact on the heat transfer and possibly other quantities.
- Summarize the results and propose possible improvements in future work.

Bibliography / sources:

According to the recommendation of Thesis supervisor

Name and workplace of master's thesis supervisor:

doc. Ing. Karel Petera, Ph.D., Department of Process Engineering, FME

Name and workplace of second master's thesis supervisor or consultant:

Date of master's thesis assignment: **21.04.2021** Deadline for master's thesis submission: **04.06.2021**

Assignment valid until: **19.09.2021**

doc. Ing. Karel Petera, Ph.D. Supervisor's signature	prof. Ing. Tomáš Jirout, Ph.D. Head of department's signature	prof. Ing. Michael Valášek, DrSc. Dean's signature

III. Assignment receipt

The student acknowledges that the master's thesis is an individual work. The student must produce his thesis without the assistance of others, with the exception of provided consultations. Within the master's thesis, the author must state the names of consultants and include a list of references.

Date of assignment receipt	Student's signature

I confirm that the diploma work was disposed by myself and independently, under leading of my thesis supervisor. I stated all sources of the documents and literature.

In Prague ……………………   ………………………………

Kaan Ege Temiz

Annotation sheet

Name: Kaan Ege

Surname: Temiz

Title Czech: Optimalizace přenosu tepla mezi proudem tekutiny a teplosměnnou plochou

Title English: Optimization of heat transfer between fluid stream and heat transfer surface

Scope of work: number of pages:55

number of figures:36

number of tables:11

number of appendices:0

Academic year: 2020/2021

Language: English

Department: Mechanical Engineering

Specialization: Process Engineering

Supervisor: doc. Ing. Petera Karel Ph.D

Reviewer:

Submitter: Kaan Ege Temiz

Annotation - English: The use of modified surfaces to improve heat transfer is quite common. When examined from a technological and economic point of view, modifying the heat transfer surface with pits is one of the most suitable techniques. After various simulations, it has been observed that the use of these surfaces in impinging jet flows negatively affects heat transfer. On the other hand, with the selection of the right geometries and parameters, it has been determined that there is an improvement in heat transfer thanks to these surfaces in wall jet simulations. However, in order to take into account the pressure drop variation, which is another important factor other than the heat transfer coefficient, the performance evaluation factor, the JF Factor, was calculated and used as a comparison parameter.

Keywords: Simulations, heat transfer coefficient, CFD, PEC, JF Factor, Dimpled Surface, Fluent

Table of Contents

List of Figures

List of Tables

Abstract

Studies conducted to date show that one of the most effective ways to increase the heat transfer rate is modifying the heat transfer surface. In this study, brief information about some heat transfer surface modification techniques has been given, and more detailed research results are shared about dimpled surface technologies. In addition, some basic simulations of the dimple surface were studied and the results of these studies were compared with a smooth surface. ANSYS Fluent Solver was used as a software for analyzing.

Two different types of flow are used in the simulation: impinging jet flow and wall jet flow. The simulations were made with stream flows over the heated surface with dimples. Environmental conditions throughout the flow are considered the same and constant for all simulations.

Performance evaluation criteria were used to evaluate the wall-jet simulation results more clearly. With the JF Factor calculations, which is the chosen PEC method, better efficiency results were obtained for the surfaces modified with dimples than the smooth surface. Furthermore, the simulations were repeated by changing different flow rates and pipe lengths and the results were examined.

1.Introduction

The advancement of technology has resulted in substantial energy consumption worldwide. For this reason, it has become very important to use energy more efficiently. In particular, techniques to increase heat transfer are vital in terms of energy saving and optimum efficiency from resource energies. In some areas like cooling microelectronic components, in the internal cooling passages of turbine blades, macro and micro scale heat exchangers, and even biomedical devices, the importance of heat transfer enhancement has gained greater significance.[2]

Although the modification of the heat transfer surface has not been a subject of much research before, heat transfer enhacement has been strongly related to surface modification techniques and improvements are always emerging in this field. Previous research shows that it is possible to increase the rate of heat transfer significantly by modifying the heat transfer surface. Especially for heat exchangers there are lots of research has been carried out. Researchs show that baffles/ribs/fins/dimples have been applied broadly in order to change flow pattern of the heat exchangers and enhance the heat transfer efficiency.[3] Over past few years, dimples have received much more attention to obtain higher heat transfer rates. This is because the previous studies show that with using dimples it is possible to enhance heat transfer with relatively smaller pressure loss compared to others.[4] In this study, basic information about some technologies used for the modification of the heat transfer surface is given but mainly focused on dimpled surface technology to enhance heat transfer.

In addition, various simulations have been studied to observe the enhancement of heat transfer by modification of the heat transfer surface. In these simulations, the heat transfer surface is modified with different shapes of dimples. Heat transfer behaviors and properties are observed under different type of flows. The results obtained are compared with the smooth surface under the same flow type and constant environmental conditions.

With additional simulations for the wall-jet, the effects of parameters such as the velocity of the fluid and the length of the pipe on the heat transfer on the modified surface were investigated. The heat transfer coefficient data calculated by Fluent as well as the heat transfer coefficient data calculated by the formulation were used for Nusselt correlations.

The data found after Nusselt number calculations are compared with the JF factor, which is a performance evaluation factor and assess the efficiency of the system using both, the heat transfer and pressure loss.

2. Surface Modifications Techniques

Today, rising energy costs and increases in material prices have led to the need to save on initial investment costs, to have a smaller volume of equipment, and to provide better heat transfer from the unit surface. Improving the heat transfer in the heat exchangers will not only reduce the size of the heat exchangers, but also reduce the investment costs. This will ensure that less material is used for the same amount of heat transfer. It is aimed to obtain detailed theoretical and technical information, which is felt in the literature and which is not given sufficient information by the companies producing in the field, through researches and to eliminate the lack of information on this subject, which is also needed by the industry.

It is possible to classify heat transfer enhancement techniques as active and passive methods. The method that provides an improvement in heat transfer by giving additional energy to the fluid or the environment to which the heat is transferred is called the active method, and the method that provides the improvement in heat transfer without additional energy is called the passive method. [14]

In order to increase heat transfer, methods such as rotating the surface, mixing the flow with mechanical parts, creating surface vibration, vibrating the fluid, creating electro-static fields in the flow medium can be given as examples of the active method. Active methods are generally more compact to design and utilize. This method usually requires an extra power input to improve heat transfer so they are not widely used today.

To increase heat transfer; enlargement of surface areas is routinely used in almost all heat exchangers. In these techniques, passive techniques are more considerable because active techniques are more complex from the design perspective.[13]

Increasing heat transfer has gained great importance in terms of heat exchangers today. In addition to the application of surface enhancement techniques in the refrigeration and automotive industry, the process industry is also attempting to use heat transfer enhancement methods in heat exchangers.

2.1 Passive Techniques

Special surface geometries or fluid additions are used for improvement in passive techniques. By processing the heat transfer surface; Covering the surface, changing the surface, creating protrusions separate from the roughness on the rough surfaces are the best examples of the passive method.

2.1.1 Surface Coating Technology

Due to the large number of materials that can be used and the ability to optimize these materials according to the desired result values, its area is very wide.

The surface contains metallic or non-metallic coating. Dry coating (Teflon) is used to provide condensation with drop formation. The well-scaled porous coating can be used to improve bubbling.

Many studies and experimental analyzes have been conducted on coating technology until today. Coating technology is a widely used method as it can prevent corrosion as well as increase the efficiency of energy. Especially nanocoated surfaces are common to design of heat exchangers. Some common materials used in coating and their thermal conductivity values are presented in Figure 1.

Foulant	Thermal conductivity (W/mK)
Alumina	0.42
Biofilm (effectively water)	0.6
Carbon	1.6
Calcium sulphate	0.74
Calcium carbonate	2.19
Magnesium carbonate	0.43
Titanium oxide	8.0
Wax	0.24

Figure 1. Thermal Conductivies for Given Foulants

Due to the large number of materials that can be used and the ability to optimize these materials according to the desired result values, its area is very wide.

2.1.2 Rough Surface Creation

It can be created by placing an element or adjacent to the surface (solid surface). The solid surface can be formed by machining or restructuring. In single-phase flow, instead of increasing the heat transfer surface area, it is more preferable to mix the boundary layer region near the surface. Two solid roughness examples are shown in Figure 2. Creating a rough surface with machining is not an economically viable approach.

Figure 2. Monolithic rough surface [22]

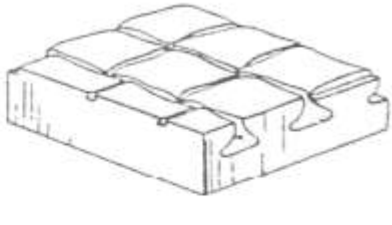


Figure 3. Improved surface for bubbly boiling [22]

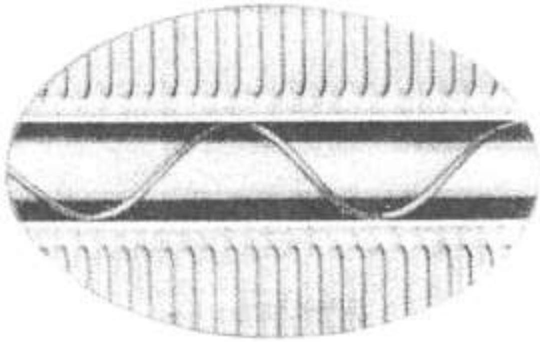


Figure 4. Surface with spiral helical spring addition [22]

Figure 3 shows the surface developed for bubble boiling. Artificial bubble zones are created in the surface structure. In this way, higher performance is obtained compared to the flat surface. Figure 4 shows the addition of a spiral helical spring that periodically stirs the boundary layer. Spiral helical spring application can be considered as an example of creating roughness with the element adjacent to the surface

2.1.3 Augmented Surfaces

It is routinely used in most heat exchangers. While the use of flat blades only increases the area, the use of a specially shaped increased surface can also increase the heat transfer coefficient.

Today's improvement efforts for gases are directed towards increased surfaces that provide a higher heat transfer coefficient than a flat fin. Figure 5 shows the various augmented surfaces used for gases.

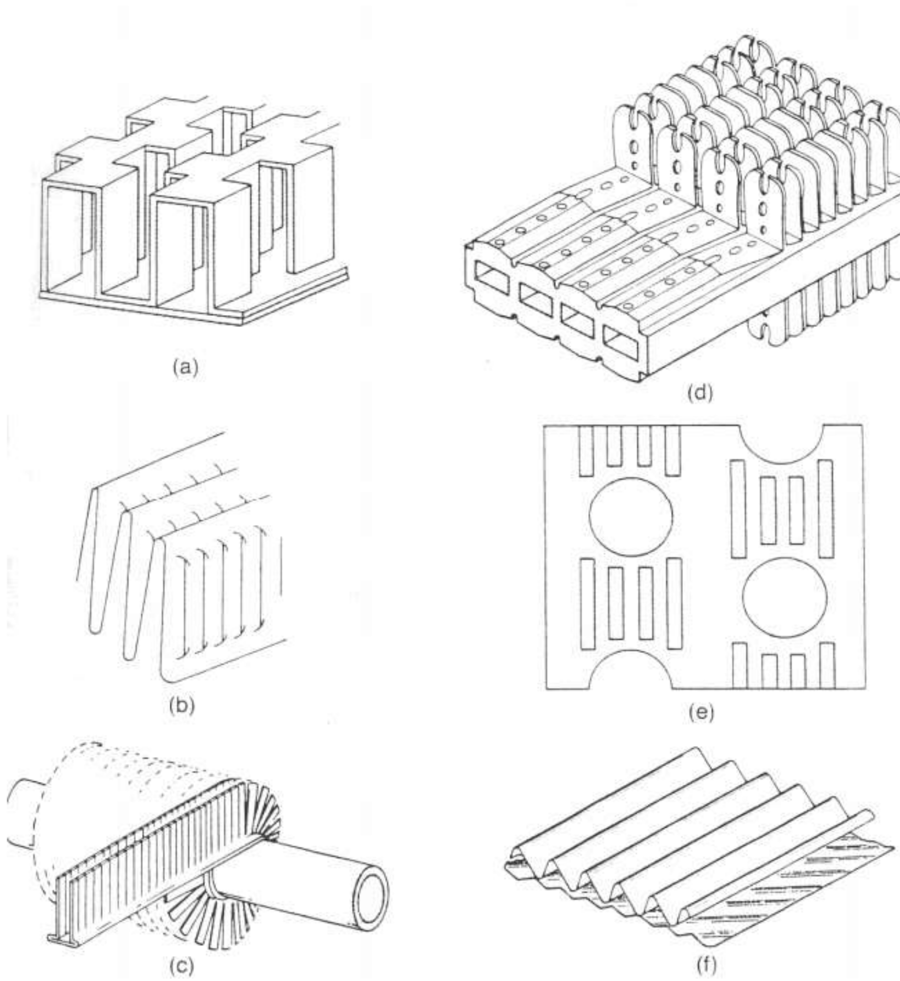


Figure 5. Surfaces enhanced by destruction [22]

Thin boundary layers are formed by repetitive shapes and surface destructions from Figure 5. Smaller blade lengths increasing surfaces are used more for liquids than for gases, because the heat transfer coefficients of liquids are higher than gases. Using long blades for liquids will result in low blade efficiency and use of excess material.

Examples of augmented surfaces used for liquids are shown in Figure 6. Figure 6a shows an external finned pipe and Figure 6b shows an internal finned pipe. In Figure 6c, multiple and concentric internal finned pipes are shown. Figure 6d includes a five-armed aluminum turbulator. The turbulator is tightly fitted to the surrounding pipe, ensuring good contact. The geometries shown in Figure 6 are also used in forced convection in evaporation and condensation.

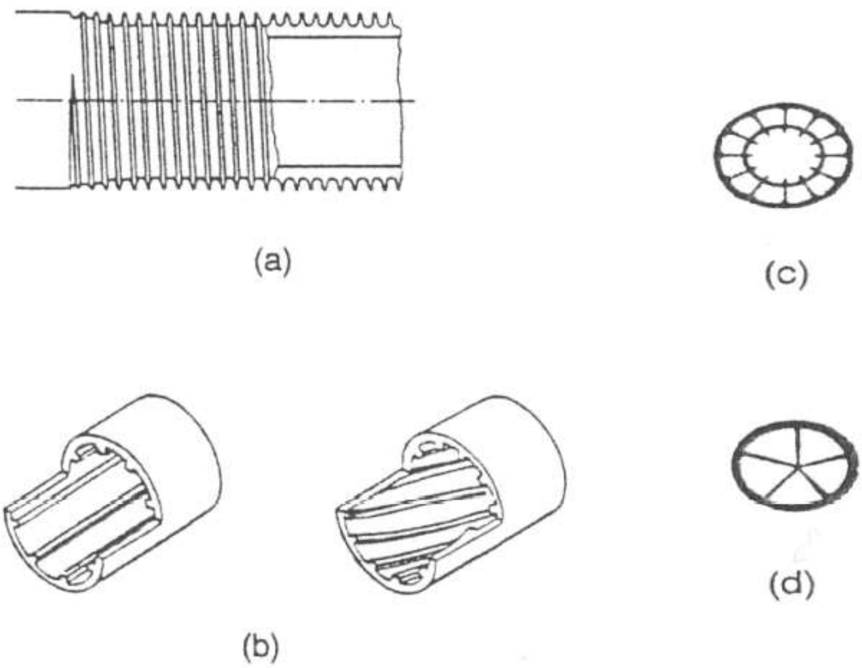


Figure 6. Examples of augmented surfaces used in liquids [22]

Fin geometries in rectangular, cylindrical or circular shapes are used to enlarge the heat transfer surface. Rectangular fins are the most preferred type of fins in terms of low production costs, easier production compared to other shapes and easy application. The Figure 7 shows the most common fin geometries used to improve heat transfer.[15]

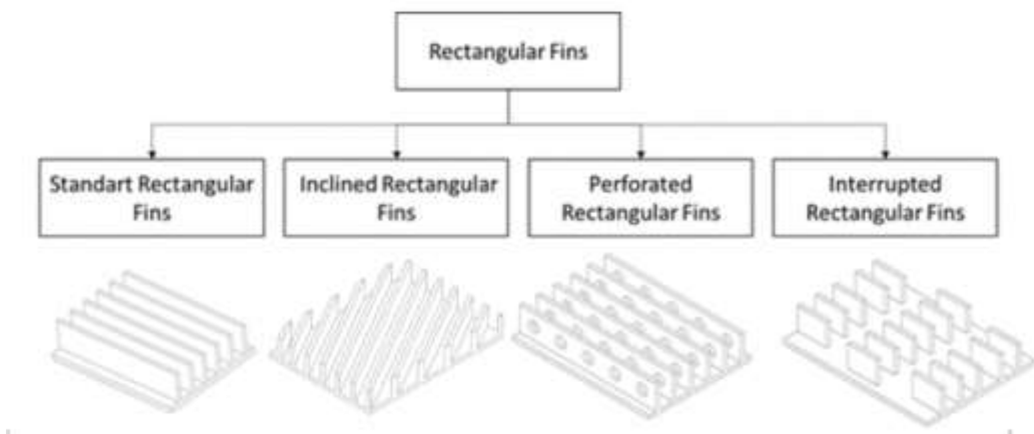


Figure 7. Rectangular Fins **[12]**

Installing fins on the heat transfer surface improves heat transfer by increasing the convective heat transfer coefficient or by expanding the heat transfer area. Since it is cheap and easy to manufacture, it is widely used in heat exchangers and cooling of electronic devices.

According to the amount of heat transfer demanded, the designer should optimize parameters such as blade geometry, number of blades, distance between blades. For this optimization, the experience of the designer is very important. Otherwise, the fins may block the incoming flow and reduce the heat transfer rate.

2.1.4 Thermoexcel Surface

Thermoexcel surfaces have a special micro structure. This surface promotes the boiling of liquids and the condensation of vapors. Thanks to the thermoexcel surface, a large amount of heat can be transferred despite the small temperature difference. Thermoexcel surfaces, especially used in heat exchanger designs, can significantly increase heat transfer.

It is a highly preferred application because it is easy to manufacture and can be applied to both the inner and outer surfaces of the pipes. On the other hand, the fact that the use of thermoexcel surfaces is very costly limits the usage area.

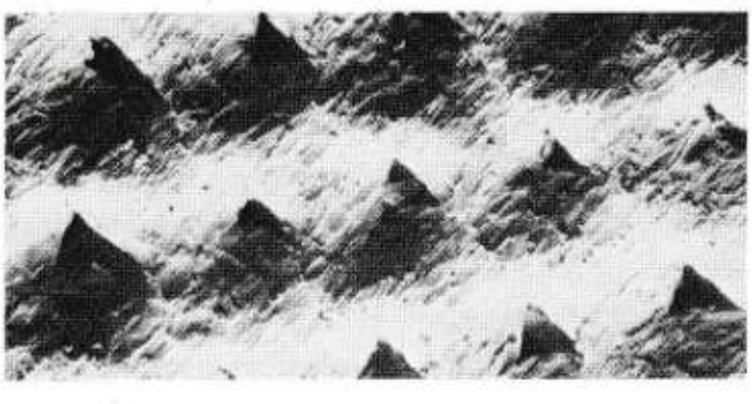


Figure 8. Thermoexcel Surface [23]

2.1.5 Wrapped Tubes:

Smaller sized heat exchangers can be provided with the help of wrapped tubes (Figure 9). Secondary flow in the wound tube produces a higher single-phase coefficient and improvements in most boiling regimes. However, very small coil sizes are required to achieve a reasonable improvement.

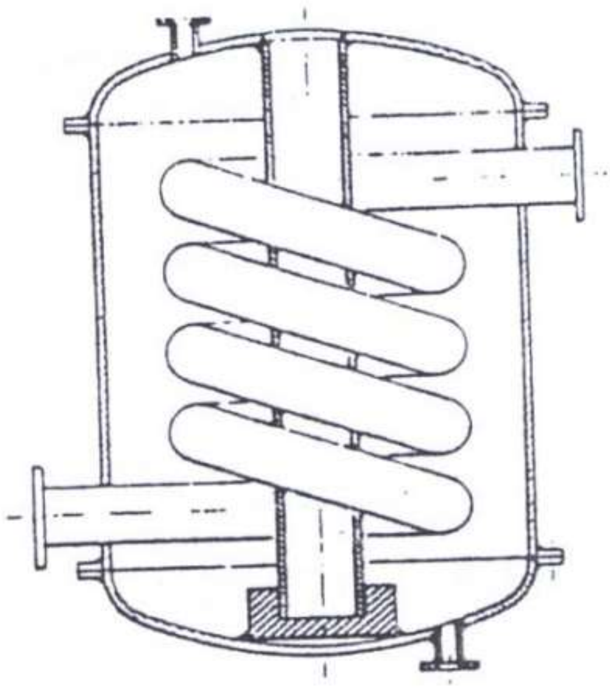


Figure 9. Wrapped tube heat exchanger [22]

2.1.6 Turbulators

In recent years, the use of turbulators has increased due to the difficulties of manufacturing finned surfaces and fixed deflector blades, the increase in the size of the heat exchanger and the difficulty of maintenance. The increase in heat transfer in turbulators is achieved by increasing the heat transfer coefficient rather than increasing the surface area. In order to increase the heat transfer coefficient, turbulence increasing techniques are tried in the heat exchanger.

In application to turbulators; it is encountered in systems such as smoke tube hot water and solid fuel boiler, heat exchangers, gas-cooled reactor fuel elements, ventilation equipment of electronic systems. Turbulators, conical ring surface, spiral and perforated pallet mixer, etc. they have many varieties, including.

Figure 10. Various Types of Turbulators

2.1.7 Dimple Technology

The surface having number of depressions on it is known as dimpled surface. In general, dimples are already well known from golf ball aerodynamics. [1] Dimples can be of various shapes and sizes. Today, dimpled surfaces are used for different purposes in many different sectors. Most commonly used dimple shape is concave spherical. [Figure 11]

Figure 11. Concave Dimples

Studies conducted to date show that the use of dimple surfaces provides a heat transfer improvement of 25% - 40% and at the same time the pressure change is almost negligible.[Figure 13] This is one of the biggest advantage of using dimple surfaces compared to other surface modification methods. In addition ,dimple jackets are also widely used for vessels at the present time.[Figure 12] Cost of dimple jackets applications are generally lower compared to other applications because pressure is lower therefore the thickness of vessel can have smaller value.[5]

Figure 12. Dimple Jackets [5]

Another advantage of using a concave dimple is about the weight of the material. For the concave dimple, the piece is removed from the material. But for the other applications like ribs,pins,fins; extra material added which increases weight and cost of the equipment.[7]

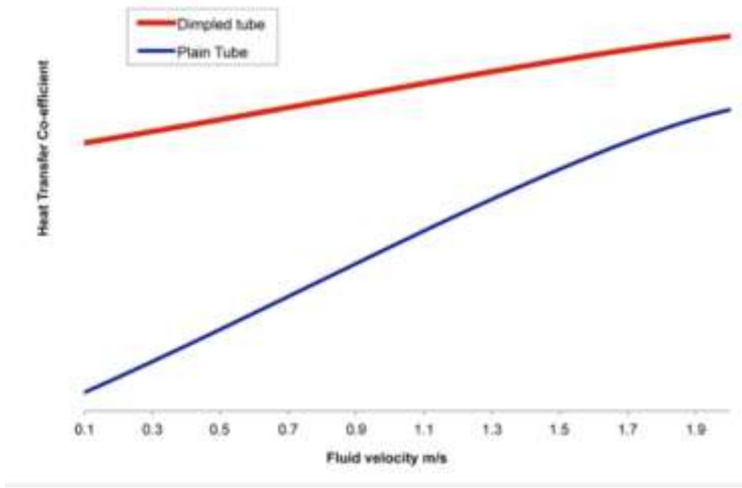


Figure 13. Comparison between plain tube and dimpled tube **[6]**

In most of the applications in industrial process technology, dimpled plates are preferred in heat exchangers because they are more advantageous technically and economically. The flexible design of each pillow plate allows the products to be tempered in other processing steps.

Welding contours in the desired geometry can be produced by manufacturing on CNC controlled laser welding equipment. This facilitates flexible shaping of the dimple plate. Thus, simple rectangular shape plates can be produced as easily as round plates or plates with complex geometric outer contours. Moreover, the flow regulation can be easily adapted to the flow rate of the heating or cooling medium in the plate, thus optimizing the heat transfer and pressure drop.[Figure 14]

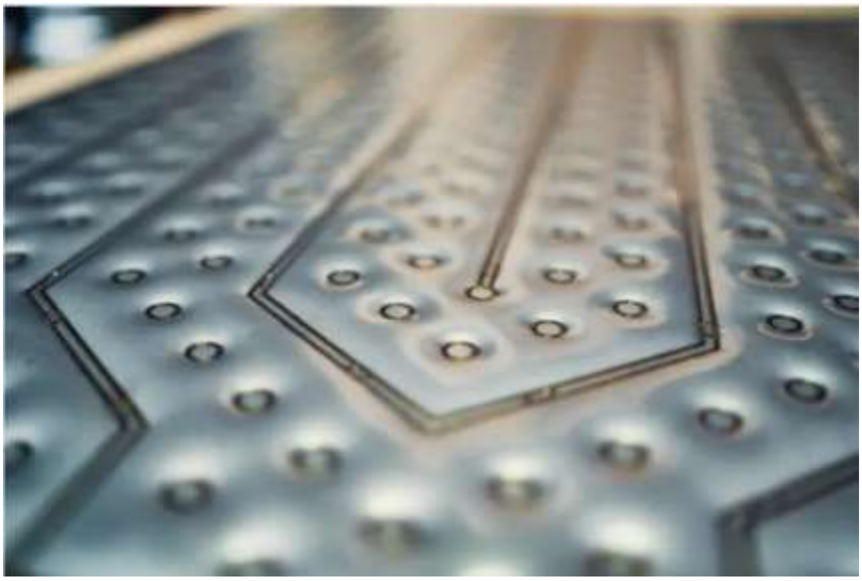


Figure 14. Optimal dimple plate welding contours thus optimizing heat transfer and pressure drop

Figure 15. Pillow plate design of a square process tank [24]

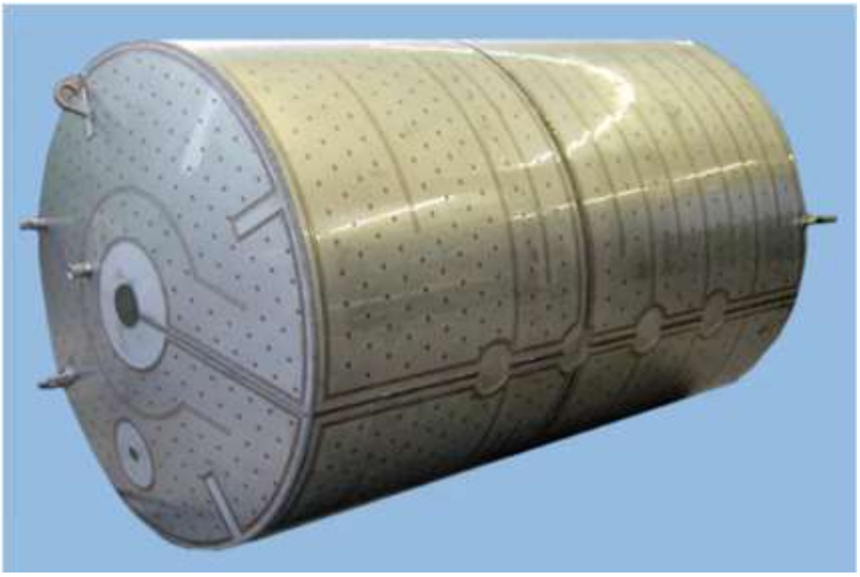


Figure 16. Pillow-Plate-Process Tank [25]

A lot of work has been done in the past on dimpled surface technology. Afanasyev et al. [16] studied the heat transfer and pressure drop in a modified flat plate with not very deep troughs for the turbulent flow regime. And he found that when pressure variation is neglected, there can be about 30-35% improvement in heat transfer.

Mahmood et al. [17] found the heat transfer distribution and local Nusselt number changes on the heat transfer surface by examining the structure of the flow on the dimpled surfaces by making visual and numerical analysis of the flow. Their results showed the existence of a low heat transfer region in the upstream half of the dimple cavity followed by a high heat transfer region in the downstream half. Additional regions of high heat transfer were identified at the downstream rim of the dimple and on the flat surface downstream of each dimple.

Ligrani et al. [18] investigated Nusselt numbers and flow oriantation on a modified channel with shallow dimples, including the effects of inlet turbulence level. They discover that the highest local Nusselt number ratios are presented within the downstream portions of dimples, as well as near dimple span-wise and downstream edges.

Also, It has been reported by Burgess et al. [19] that both the friction coefficient and the Nusselt number can increase with the increase of the depth of the dimple used in the surface modification.

Wang et al. [20] investigated a newly developed heat transfer tube with ellipsoidal dimples, and the calculated results revealed that the Nusselt number for ellipsoidal dimpled tube and spherical dimpled tube was 38.6–175.1% and 34.1–158% higher, respectively, than for straight tube. On the other hand, the friction factors of dimple pipes are 26.9-75% and 32.9-92% higher for ellipsoidal and spherical dimples, respectively, compared to straight pipe.

Chyu et al. 1997 [21], investigated the heat transfer coefficient distributions of the air flow over the duct covered with dimples in two different geometries. According to the results, it was determined that the local heat transfer coefficients were significantly higher than the values in the smooth surfaced channels.

From these investigations, the results show the superior performance of dimple surface. However, the influence of dimple arrangement performance on the heat transfer has no certain study or report on the literature.

2.2 Active Techniques

Active techniques, require the use of external force such as an electric or acoustic field and surface vibration.

Methods such as using mechanical auxiliary elements, rotating the surface, mixing the flow with mechanical parts, creating surface vibration, vibrating the fluid, creating electro-static fields in the flow medium to increase heat transfer can be given as examples of the active method.

Mechanical Tools: It consists of means that drive the flow or rotate the surface. Mechanical surface scrapers are widely used in gas pipe flows in the chemical process industry for viscous fluids. Equipment in rotary heat exchanger ducts is found in industrial applications.

Surface Vibration: It is primarily used to increase heat transfer in a single phase, both at low and high frequency.

Fluid Vibration: It is a more practical vibration technique due to the large mass of many heat exchangers. Single-phase fluids are primarily preferred.

Electrostatic Fields: Direct current (DC) or alternating current (AC) are used in insulating fluids in various ways. In general, electrostatic fields can be directed around the heat transfer surface to mix a larger volume of liquid.

Injection: It is used by supplying gas to a porous heat transfer surface in the liquid flow or by injecting the same liquid against the flow in the heat transfer area. The injected gas increases the two-phase flow. Degassing the liquid from the surface can have the same effect.

Absorption: It is applied in vapor transport, bubble or film boiling, liquid removal in two-phase flow from the porous heat transfer surface.

3. Procedure and Software Used

In this study, simulations were performed for 2 different flow types: impinging jet flow and wall-jet flow. The numerical experiments were done with stream flows over the heated surface with dimples for both types of flows. Environmental conditions throughout the flow are considered the same and constant for all simulations.

In the 2D simulations made for the impinging jet flow type, the surface where the heat transfer will take place has been modified with various dimples in different shapes and sizes. In these simulations, it is aimed to investigate the effects of different shaped dimples under the same conditions on heat transfer.

In the 3D simulations made for the impinging jet flow type, heat transfer surfaces modified with spherical dimples in different numbers and sizes were investigated and the results were evaluated. The purpose of these simulations is to examine the effect of the number and size of the dimples on the heat transfer.

All of the Wall-Jet Simulations, which are the other flow type, were studied on 2D geometries. But it was aimed to obtain results similar to pipe flow, therefore CFD simulations were applied as 2D asymmetric.

In repeated simulations for the wall jet, the heat transfer surfaces were first modified with dimples of different shapes. Then, different combinations of the selected spherical dimples were tried and the simulation results were compared with the geometry with smooth heat transfer surface.

From those simulations, two different geometries with the optimum results were selected and simulations were repeated with various velocities of flows to examine the effects of flow velocity on the heat transfer surface. In addition, some simple extra simulations were performed to investigate the effects of tube length on the modified heat transfer surface. In these simulations, variables other than pipe length were kept constant.

3.1 Impinging Jet Simulations

Flow orientation in the impinging jet takes place perpendicular to the surface. Jet impingement achieves locally high heat transfer on an interested surface. For these reasons, the impinging jet cooling technique has been widely used in many industrial systems such as gas turbine cooling, rocket launcher cooling and high-density electrical equipment cooling in order to remove a large amount of heat.

3.1.1 Geometry and Mesh

For the impinging jet simulations, first of all if we look at the simulations made with 2D geometries; four different geometries were studied. Three of these geometries have been modified with dimples of different shapes and one has a smooth heat transfer surface. Same analyzing conditions were applied for these geometries. Dimples shapes are chosen as concave (Figure 20), convex (Figure 18) and triangle (Figure 19). Simulated results will be compared with the geometry with smooth surface at the end (Figure 17). Also on the figure which represents smooth surface, we can see boundry conditions for the simulation.

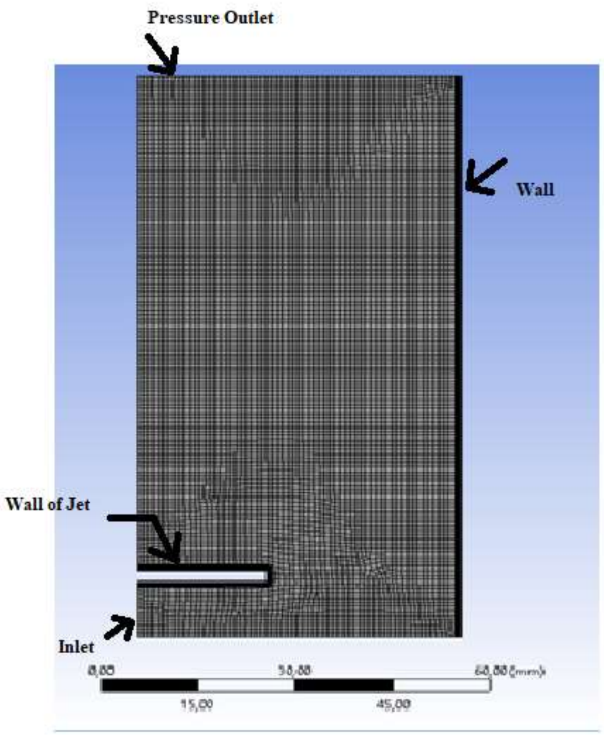


Figure 17. Smooth Surface Geometry

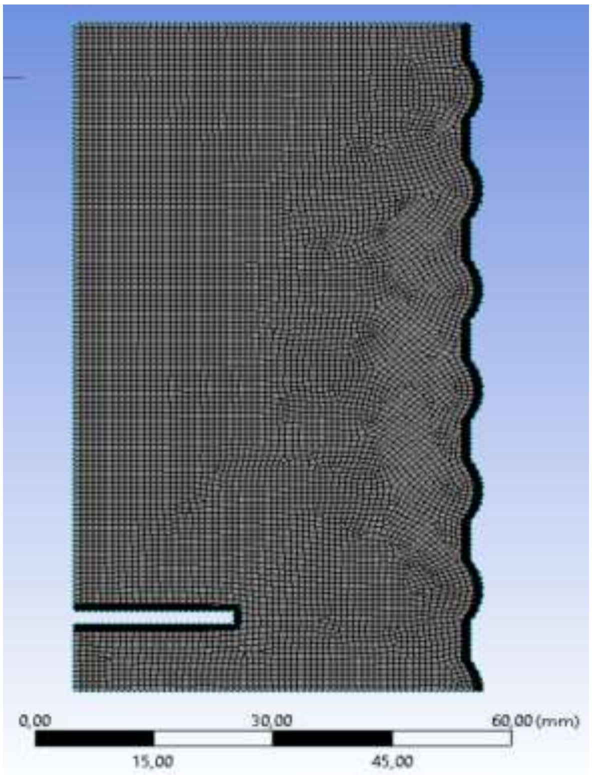


Figure 18. Convex Dimpled Surface Geometry

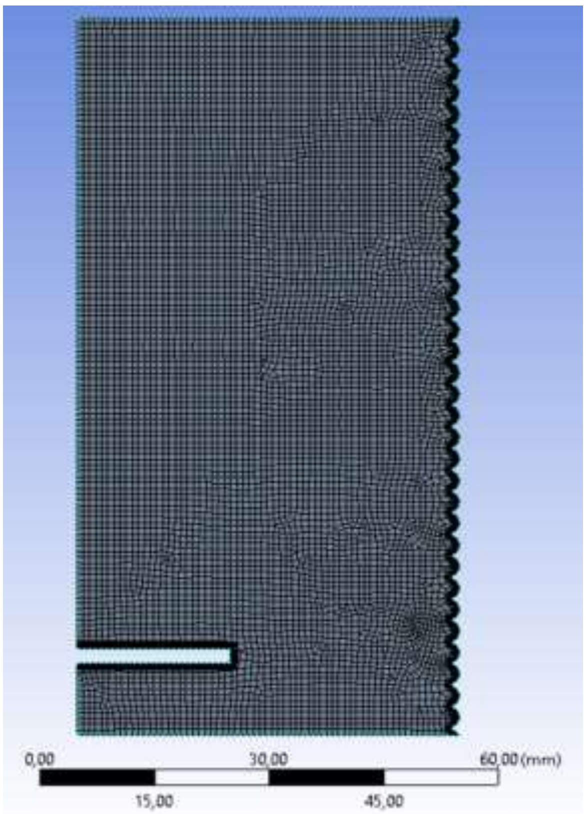


Figure 19. Triangle Dimpled Surface Geometry

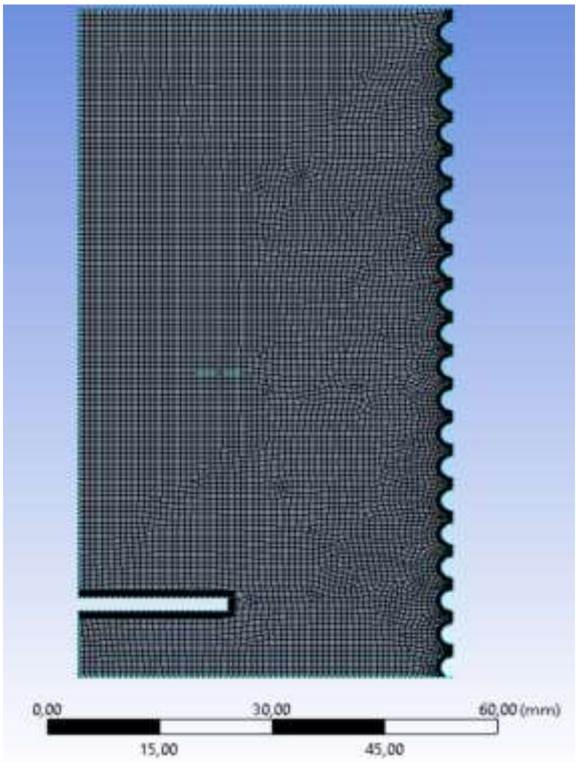


Figure 20. Concave Dimpled Surface Geometry

As we see from figures above, inflation layers were applied on two different edges. 10 inflation layers applied for the edge which represents wall of jet and 15 inflation layers added on the surface where heat transfer occurs. Furthermore, mesh parameters for four different geometries are given in Table 1.

	Element Size	Number Of Elements	Min Orthogonal Quality
Smooth Surface	0.8 mm	9886	0.374
Convex Dimple	0.8 mm	10321	0.5675
Concave Dimple	0.8 mm	12034	0.037
Triangle Dimple	0.8 mm	10288	0.2985

Table 1. Mesh Parameters for Given Geometries

3D simulations have been studied to examine the effects of dimples applied to the surface on heat transfer with more precise results. In repeated simulations, the average heat transfer coefficients of different values obtained by changing the number of dimples applied to the surface will be compared.

The geometry used for 3D impinging jet simulations is shown in the figure 21. This geometry was kept the same for all simulations, only the pattern of the concave dimple modification applied to the heat transfer surface area was changed. The heat transfer surface of this geometry was modified with 6 different dimples models and the simulations were repeated. In 6 different simulations, the dimples were applied convavely to the surface and the results will be compared by changing the parameters such as the number of dimples and the distances between the dimples.

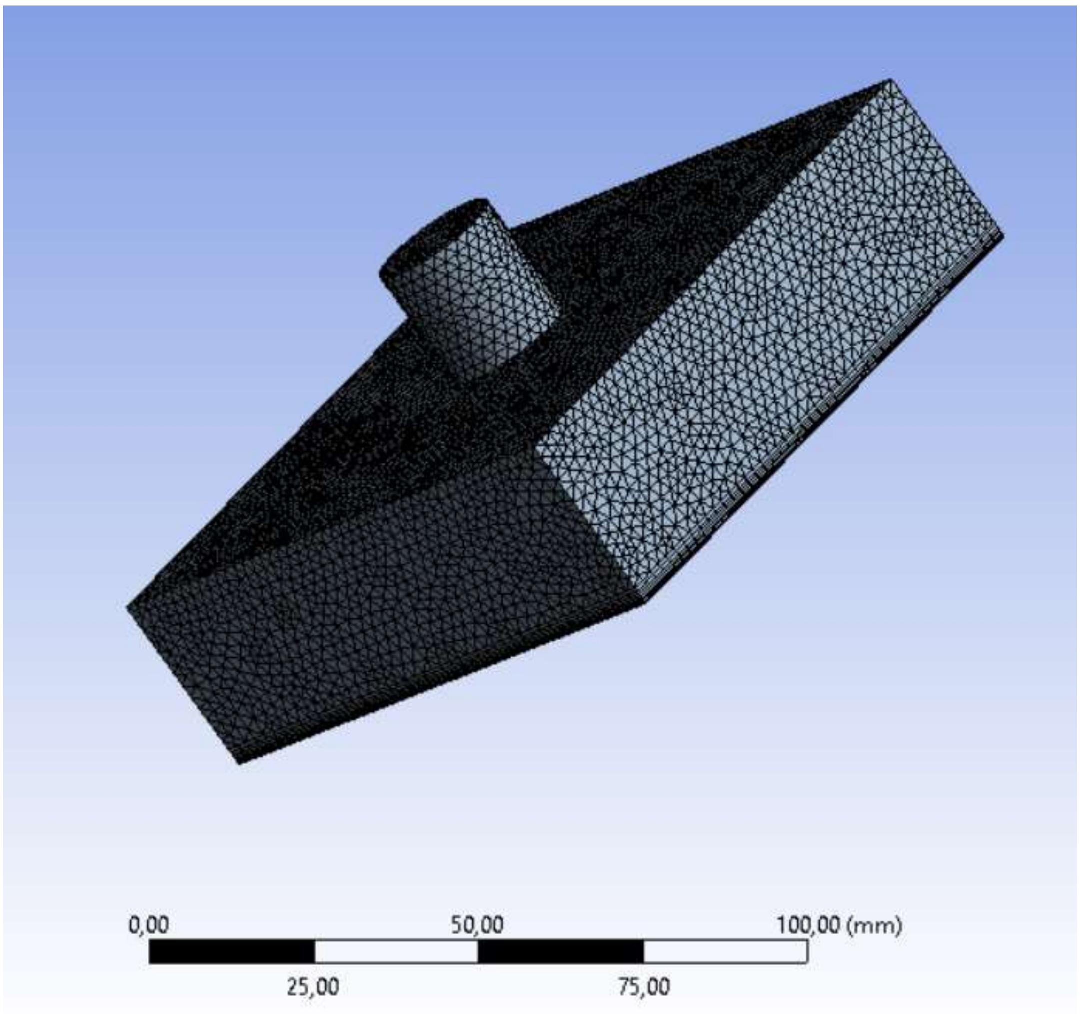


Figure 21. 3D impinging jet geometry

Geometries to be used for 3d simulations have 2,2 mm of element sizes and for all surfaces 10 inflation layers were applied.Number of elements for the mesh is around 40,000. Results will be given for each geometry in next step. All selected boundary conditions and setups for analysis are considered to be the same and constant for all simulations with 2 and 3 dimensions.

3.1.2 Numerical Setup for CFD Simulations

With simulations repeated for different geometries, the effects of various shapes of dimples applied to surfaces on heat transfer were investigated. The average heat transfer coefficient on the surface will be used as the evaluation criterion for heat transfer. These values were obtained using Ansys Fluent.

All selected boundary conditions and setups for analysis are considered to be the same and constant for all simulations with 2 and 3 dimensions. Material of fluid which flows through jet chosen as air for all simulations. The temperature of the air was accepted as 300 K at the inlet.

Simulations are performed in steady-state condition for all geometries. The velocity profile is assumed to be 5 m/s and constant. The calculated value for Reynolds number for all simulations is 6845.9, so the flow regime is turbulent and SST k-omega was used. The surface on which the average heat transfer coefficient will be calculated is chosen as stationary wall and has 1000 W/m^2 heat flux value for the boundry condition.

Applied parameters for residual monitors are applied same for all simulations and given in Figures 22. Moreover, hybrid initialization method was used for all simulations.

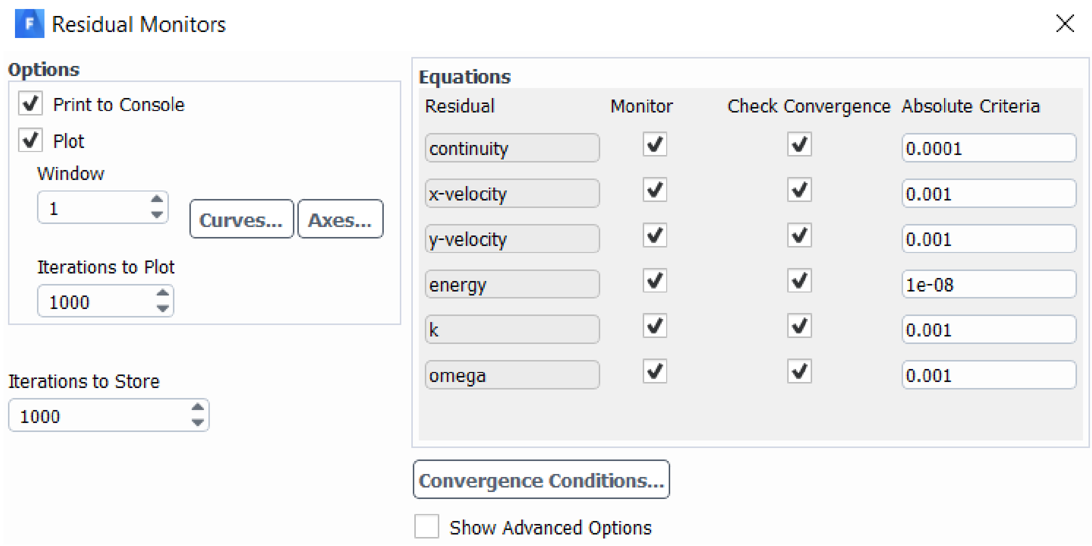


Figure 22. Applied Parameters for Residual Parameters

3.1.3 Results and Discussion

In the figures shown below, heat transfer coefficient data calculated on the heat transfer surface as a result of 2D simulations are given graphically.(Figures 23,24,25) The heat transfer coefficient data to be used for comparison was chosen as the area weighted average heat transfer coefficient and the data obtained through Fluent are given in the table 2 for four different modification types.

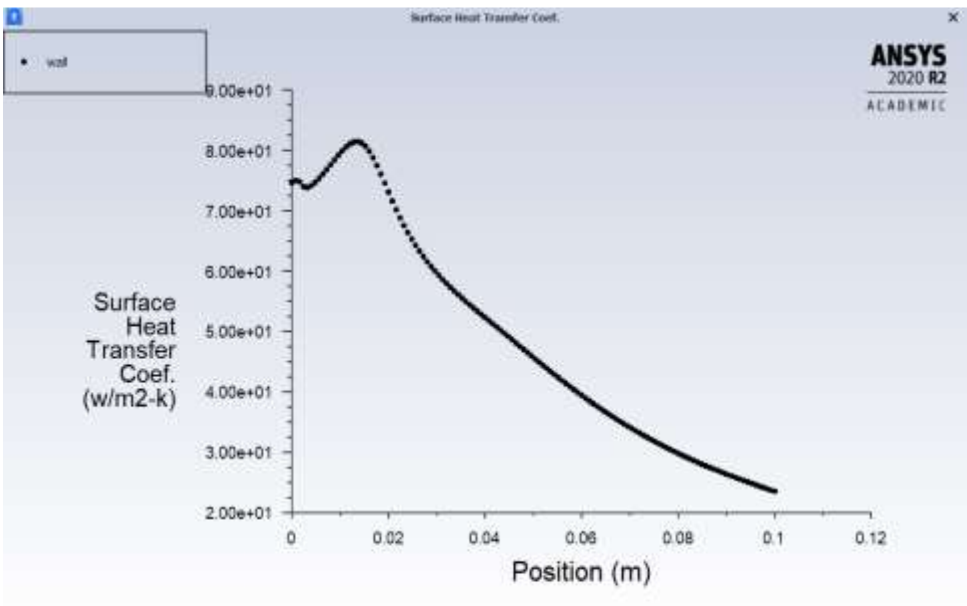


Figure 23. Heat Transfer Coefficients for Smooth Surface

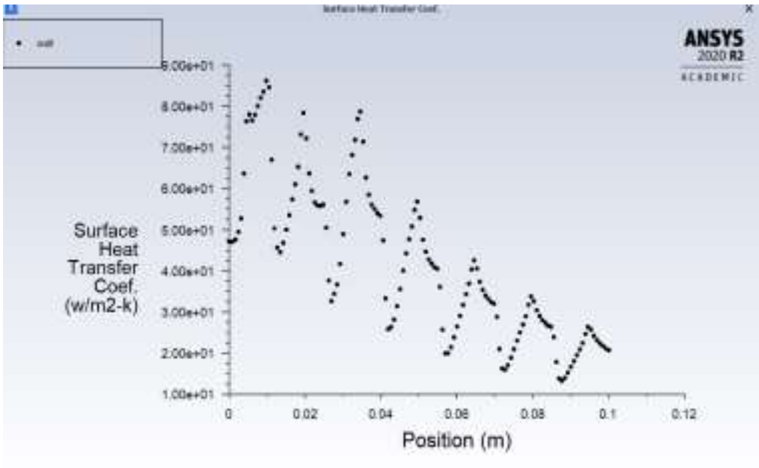


Figure 24. Heat Transfer Coefficients for Convex Dimpled

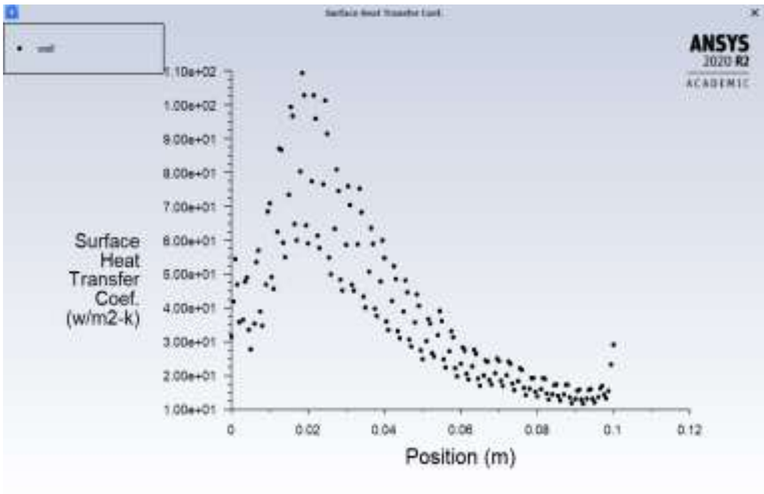


Figure 25. Heat Transfer Coefficients for Triangle Dimpled Surface

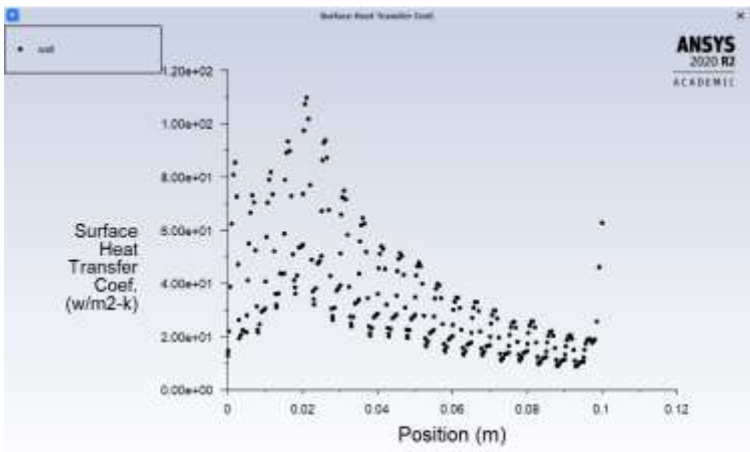


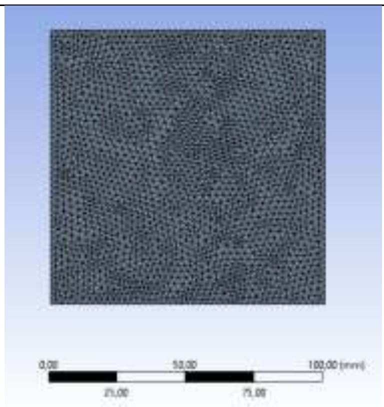
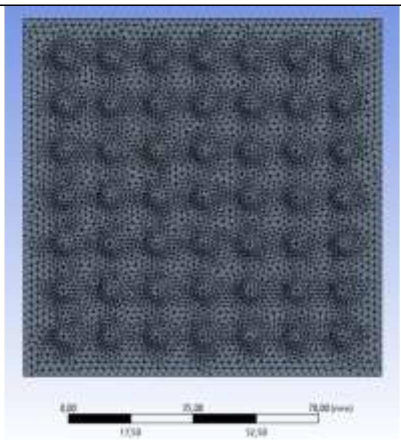
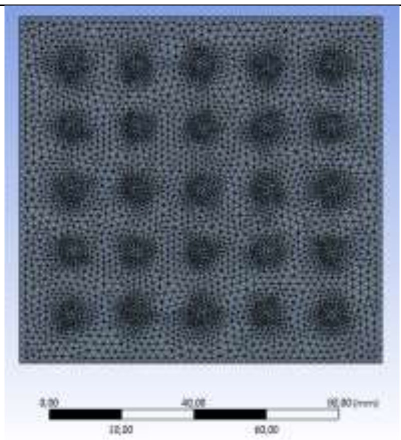
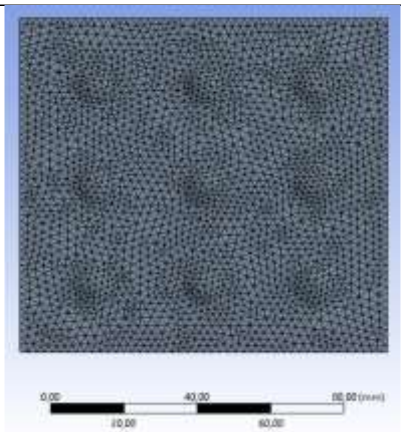
Figure 26. Heat Transfer Coefficients for Concave Dimpled Surface

	Average Heat Transfer Coef. $\frac{\boldsymbol{W}}{(\boldsymbol{m^2K})}$
Smooth Surface	38.3706
Convex Dimpled Surface 	32.967
Triangle Dimpled Surfaces 	27.749
Concave Dimpled Surface 	25.83

Table 2. Average Heat Transfer Coefficients for Impinging Jet Flow(2-D)

Looking at the data in the table above, the average heat transfer coefficients obtained from all modified heat transfer surfaces are lower than the smooth surface.(Table 2) This means that the effect of the modified surfaces on heat transfer is negative. Among the modified surfaces, the highest heat transfer coefficient was obtained for the convex dimple geometry. The concave dimpled surface, which has the lowest heat transfer coefficient, is 33 percent lower compared the smooth surface.

For 3D simulations, the average heat transfer coefficient values obtained from repeated simulations for a total of 7 different geometries, one of which is smooth surface, were found using Fluent Solver. In the table below, the heat transfer coefficient values evaluated for each geometry and the number of dimples are shared.(Table 3)

Dimples Pattern	Dimple Number	Average Heat Tras. Coef. $\frac{W}{(m^2\ K)}$
	Smooth	54.743
	49	50.699
	25	50.328
	9	52.643

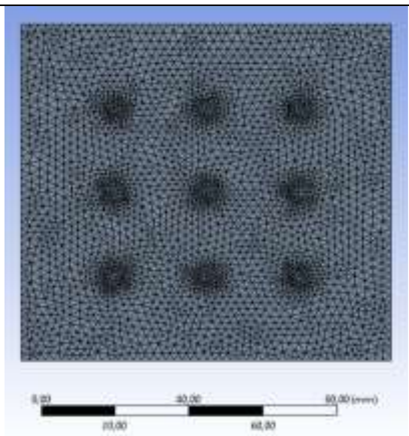
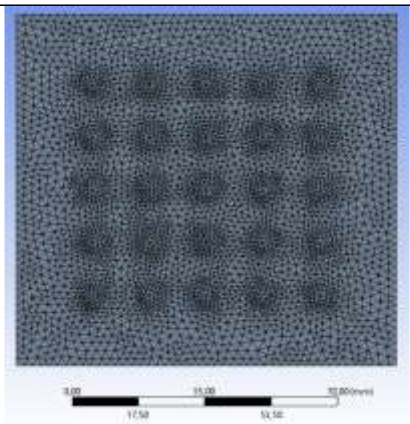
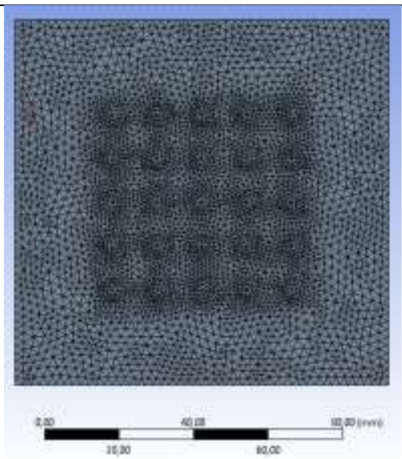
	9	53.624
	25	53.769
	25	52.872

Table 3. Average heat transfer coefficient values for impinging jet geometries

It may not be correct to draw a clear conclusion from the heat transfer coefficient values given in the table, but it is sufficient to have information about the effects of applied modifications on heat transfer. When we look at the results of 3-dimensional simulations, we can see results similar to 2-dimensional simulations.

We can say that all of the dimples applied in 6 different models affected the heat transfer negatively. However, it is not possible to say directly how the number of dimples and their positions affect the heat transfer coefficient. Optimum values can be reached with repeated simulations with more variables.

As a result, it is possible to say that the surfaces modified with various dimpls negatively affect the heat transfer for impinging jet flow

3.2 Wall Jet Simulations

The name “wall jet” is applied to a jet of fluid which spreads out over a surface, the fluid outside the jet being at rest. In this type of flow, orientation is parallel to the surface. It is a frequently used method in film cooling applications and boundary layer control. Also, the radial wall jet is a variation that is important in problems of heat and mass transfer, as in heating by a torch or drying by an impinging jet.

3.2.1 Geometry and Mesh

For wall jet simulations, firstly, simulations were made for 4 different surfaces, one of which was a smooth surface. These surfaces have been modified in different ways. Figure 27 represents the geometry with smooth surface. For other geometries, surfaces are dimpled surface. Figure 28 represents the geometry which has modified with convex spherical dimples. Also we can see from figure 29 that dimples have triangle shapes and Figure 30 shows the surface modified with concave dimples. Modified surface with concave dimples is one of the most widely used methods today. For this reason, the simulations focused on modified surfaces with these concave dimples.

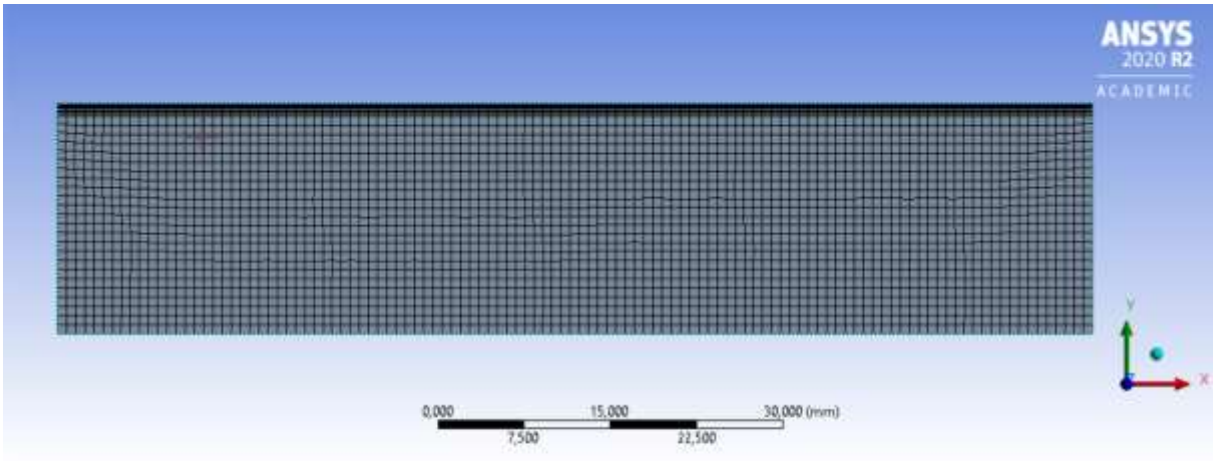


Figure 27. Smooth Surface

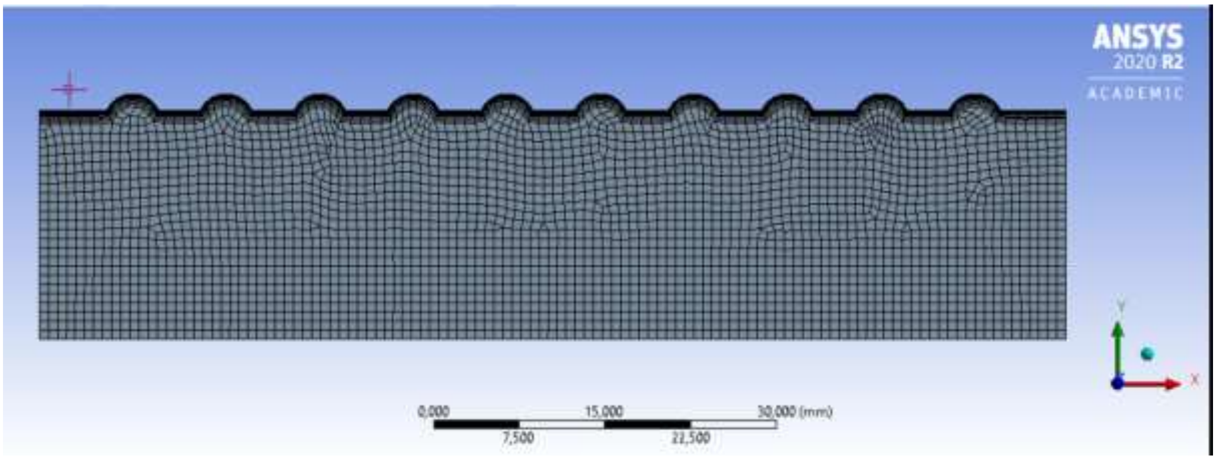


Figure 28. Surface with Convex Dimples

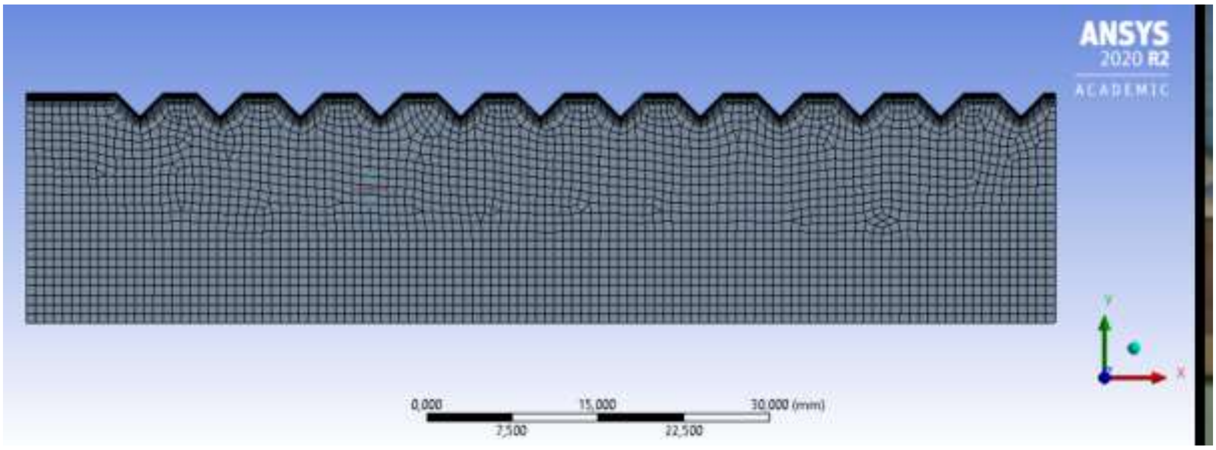


Figure 29. Surface with Triangle Dimples

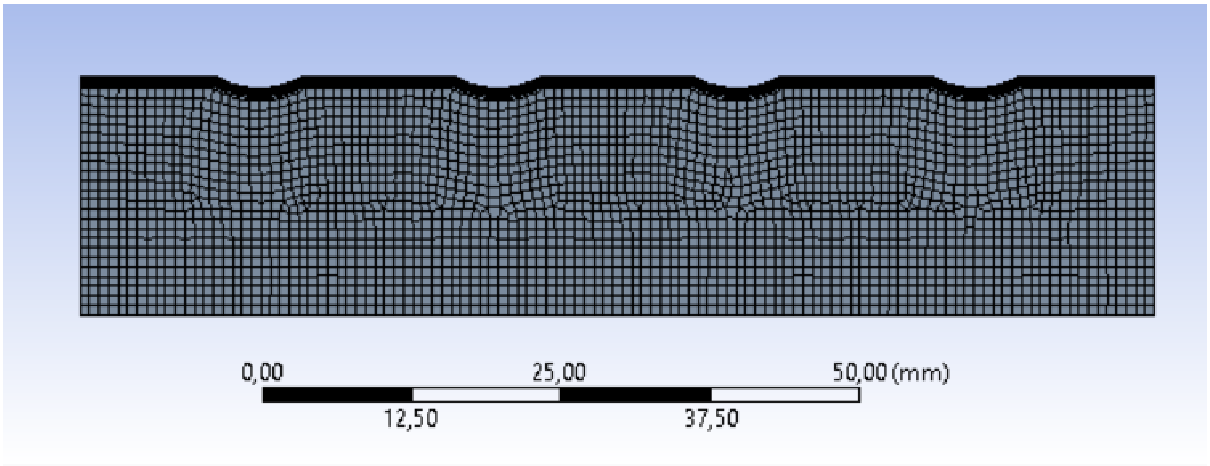


Figure 30. Surface with Concave Dimples

The 4 geometries given above are exactly the same except for the dimples applied to the surface. Since parallel flow in the pipe is represented with these geometries, a rectangle with 20 x 90 mm side lengths was chosen for the geometries.

For all wall jet geometries 10 inflation layers are applied to surface for more sensitive results. Besides mesh parameters for four different geometries are shown in table 4.

	Element Size	Number of Elements	Min Orthogonal Quality
Smooth Surface	0.8 mm	3842	0,979
Convex Spherical Dimple	0.8 mm	4048	0,579
Triangle Dimple	0.8 mm	4199	0,435
Concave Spherical Dimple	0.8 mm	4237	0,751

Table 4. Parameters for meshes

For these simulations, in addition to the above 4 different geometries, simulations were repeated for many different geometries modified with concave dimples. With these simulations, it was aimed to examine the effect of parameters such as the depth of the dimple applied to the surface, the radius of the dimples, the number of the dimples and the location of the area where the first dimple is applied, on the heat transfer. These geometries are shown in the next steps together with the heat transfer coefficient results.

3.2.2 Numerical Setup for CFD Simulations and Evaluation of Results

In repeated simulations for wall jet geometries, all data is set to be the same as for the impinging jet. The fluid in the jet was chosen as air and the inlet temperature of the air was accepted as 300 Kelvin. It is the same for all variations.

The velocity of the fluid was assumed to be 5 m/s and constant same as for the impinging jet, but for wall jet simulations; developed velocity profile applied the periodic boundary condition with the given mass flow rate corresponding to 5 m/s (0.0076969 kg/s).

The average heat transfer coefficient data obtained as a result of the simulations will be examined in comparison with the smooth surface.

The Reynolds Number is calculated as 6846, so SST k-omega turbulence model was used. The heat transfer surface is chosen as stationary wall and has constant 1000 W/m^2 heat flux value for the boundry condition.

Applied parameters for residual monitors are applied same for all simulations (Figures 31). Moreover, hybrid initialization method was used for all simulations.

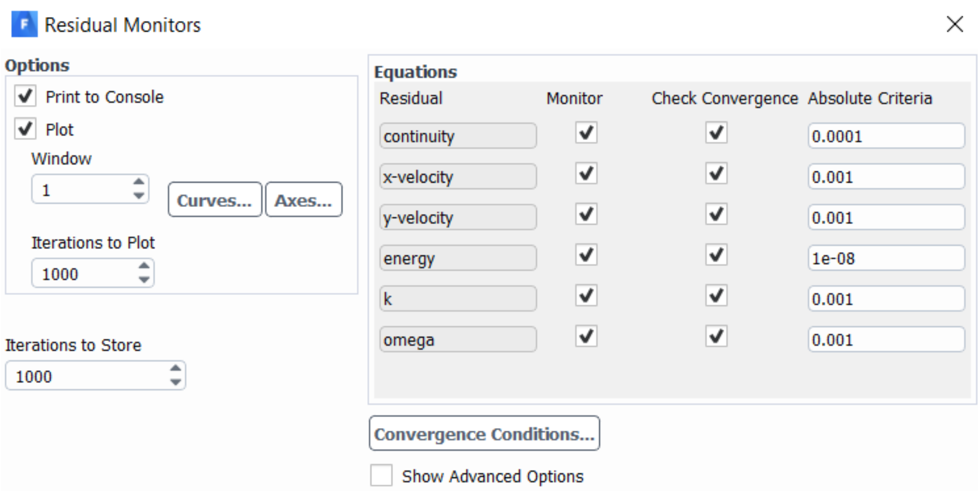


Figure 31. Applied Parameters for Residual Parameters

First, if we compare the modified surfaces with triangular, concave spherical and convex spherical dimples; we can see in the heat transfer coefficient values calculated by Ansys that all three dimple shapes improved heat transfer. In table 5, the average heat transfer coefficients were calculated for surfaces with three different geometries are given. We can see from the values that dimpled surfaces have higher average heat transfer values than smooth surface for wall jet flow.

Concave dimples give the most productive results from the modifications tried. Around 10% increase in heat transfer coefficient was obtained compared to the smooth surface. In addition, the average heat transfer coefficient obtained from the triangular and convex dimples was increased by about 8% compared to the smooth surface. So, we can say that enhancement of heat transfer is possible with using dimpled surfaces for wall jet flow. Also used shape and sizes of dimples can be modified for obtaining better heat transfer rates.

But triangular shaped dimples are more difficult to implement technologically. Especially in high velocity flows, triangular dimples increase the pressure drop. Therefore, it is not a widely used method in the industry.

	Average Heat Transfer Coef.. $\left[\frac{\mathbf{W}}{\mathbf{m^2.K}}\right]$
Smooth Surface	44.62
Convex Spherical Dimpled Surface 	47.314
Triangle Dimpled Surfaces 	47.612
Concave Spherical Dimpled Surface 	48.251

Table 5. Average Heat Transfer Coefficient Values for Wall Jet Flow

Additionally, the local heat transfer coefficient values from wall jet flow simulations for the heat transfer surface are shown graphically in the following visuals (Figures 32,33,34).

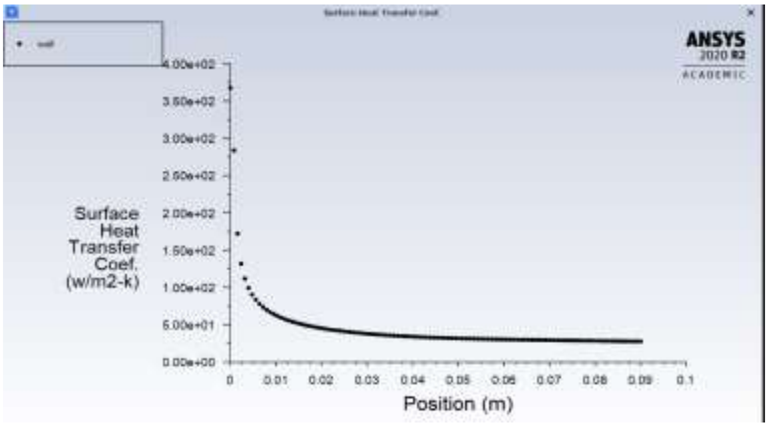


Figure 32. Local heat Transfer Coefficients for Smooth Surface

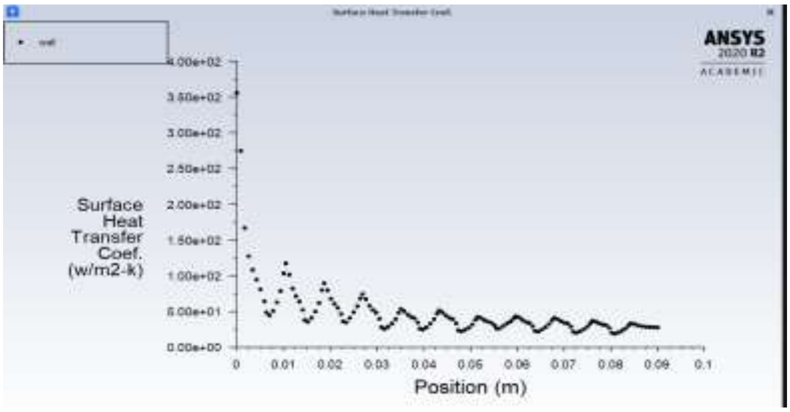


Figure 33. Local heat Transfer Coefficients for Convex Dimpled Surface

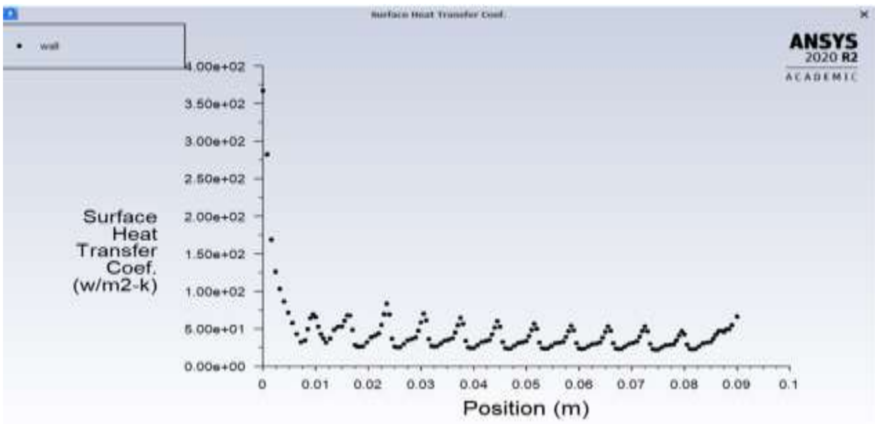
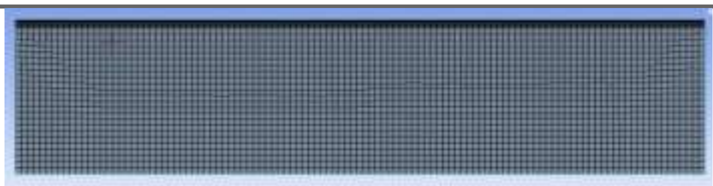
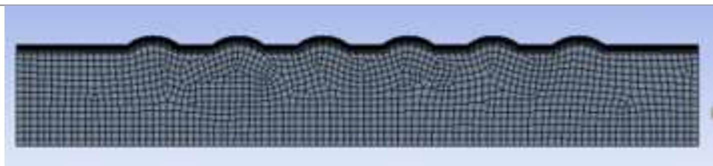
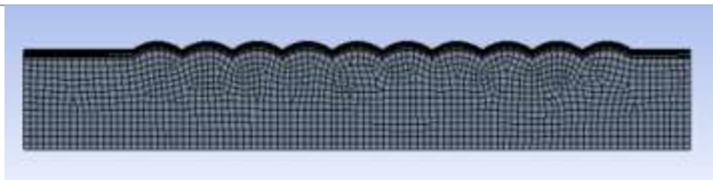
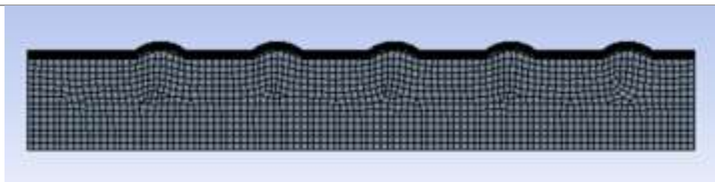
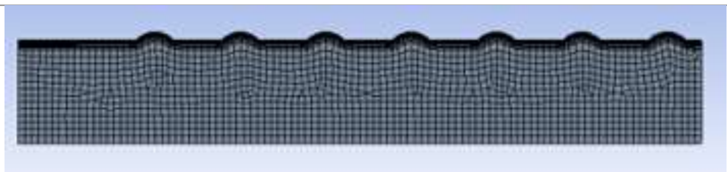


Figure 34. Local heat Transfer Coefficients for Triangle Dimpled Surface

In this study, one of the most detailed examinations was made for the 2-dimensional wall jet, and convex applied dimples were used as the modification technique. It is aimed to compare the effects of various convex dimples applied in different numbers, sizes and locations with the same geometry on heat transfer. In all geometry variations seen in Table 6, the jet radius is 20 mm and the jet length is 90 mm.

Geometry	**h** $\left[\frac{W}{m^2.K}\right]$
	44,620
	45.444
	43.425
	42.899
	41.478
	45.235
	39.588

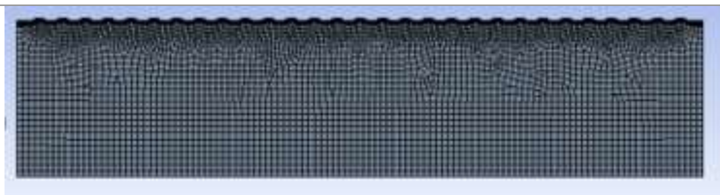
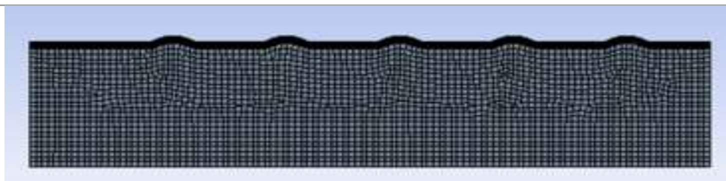
	42.879
	45.199
	47.599
	45.839

Table 6. Geometry variants and average heat transfer coefficient data

As we see from Table 6 above, although the modified shape is the same, the size, depth and number of each applied spherical dimple effects the heat transfer. Although the area-weighted average heat transfer coefficient values calculated by Fluent are close to each other, it should be noted that the geometries used in the simulations are small. For much larger geometries to be used in industry, dimples to be used in more optimum models and sizes can lead to a great improvement in heat transfer.

It is difficult to make general judgments from these simulations because there are so many variables . However, as we can see from Table 6, the heat transfer decreases as the number of dimples applied to the surface increases and the depth of the dimple increases. The lowest average heat transfer coefficient values were calculated on these surfaces. On the other hand, the position of the first dimple to be applied to the surface is an important variable. Dimples positioned too close or too far from the starting point of the flow can negatively affect heat transfer. The most optimum parameters were taken from large but few dimples with very little surface depth.

As a result, when we look at the modifications to the heat transfer surface in wall jet simulations; it is possible to improve heat transfer. However, more surface studies are required to find suitable dimple geometries and variants. According to the modification model applied to the surface, 10% improvement in heat transfer is possible, but up to 20% reduction in heat transfer can be observed if appropriate modification is not made.

4.PEC Criterion

A performance comparison of effectiveness for various types of enhanced surfaces may lead to selection criteria for designers and users. Several authors have proposed performance evaluation criteria (PEC) which define performance benefits of an structure having enhanced surfaces, relative to a standart structure with smooth subject to various design constraints.

In the previous stages of this study, the effects of different modification geometries on heat transfer were investigated. Now, among the previously studied geometries, 2 geometries with optimum average heat transfer coefficient data have been selected. These geometries are selected from wall-jet simulations and one has concave dimples and the other has convex dimpless. The geometries giving the optimum values are shown in figures 35 and 36.

With these selected geometries, the simulations were repeated at different flow rates. The heat transfer coefficient data obtained as a result of these various flow rates will be evaluated in comparison with the performance evaluation criteria calculations. In addition, some simple simulations were made and evaluated in order to investigate the effect of modified pipe length on heat transfer.

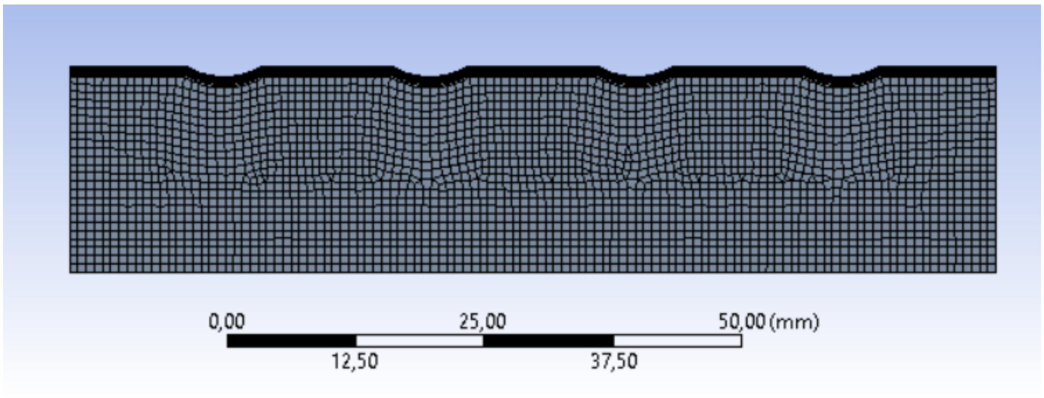


Figure 35. Optimum geometry with concave dimpled surface

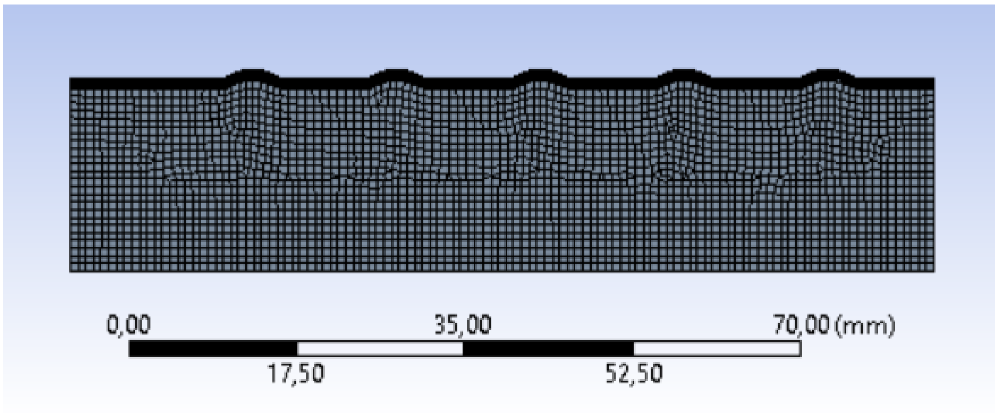


Figure 36. Optimum geometry with convex dimpled surface

4.1 Heat Transfer Coefficient

The heat transfer coefficient which will be used in Nusselt Number correlations on modified surfaces has been calculated both using Ansys Fluent and the equation of the experimental convective heat transfer rate (Eq.1) in order to obtain more precise results and comparison.

$$Q = m\, C_p \left(T_{f_{out}} - T_{f_{in}}\right) \qquad \text{Eq.1}$$

Where m, C_p , $T_{f_{out}}$and $T_{f_{in}}$ represent the mass flow rate, specific heat, inlet and outlet temperatures of the working fluid. In the simulations, the inlet temperature was set to 300 kelvin. Output temperature and mass flow rate values were obtained by Fluent.

After calculating heat transfer rate, the effective heat transfer coefficient was estimated from the ratio of the convective heat transfer rate to the total surface area and logarithmic mean temperature difference between the wall and the working fluid.(Eq.2). The formula used to calculate the logarithmic temperature change is given in the equation 3.

$$h = \frac{Q}{A\left(T_W - T_f\right)_{LMTD}} \qquad \text{Eq.2}$$

$$\left(T_w - T_f\right)_{LMTD} = \frac{\left(T_w - T_{f_{in}}\right) - \left(T_w - T_{f_{out}}\right)}{\log\left[\frac{T_w - T_{f_{in}}}{T_w - T_{f_{out}}}\right]} \qquad \text{Eq.3}$$

Wall and outlet temperatures used in logarithmic mean temperature calculations were obtained from Fluent. Where T_w is the area-weighted average wall temperature and $T_{f_{out}}$ is the mass weighted average outlet temperature.

Velocity (m/s)	Heat transfer Coefficients from Fluent $\left[\frac{\mathbf{W}}{\mathbf{m^2.K}}\right]$			Heat transfer Coefficients from Formula $\left[\frac{\mathrm{W}}{\mathrm{m^2.K}}\right]$		
	Smooth	Concave Dimples	Convex Dimples	Smooth	Concave Dimples	Convex Dimples
4	39.193	40.477	40.932	32.349	32.931	32.906
5	44.620	47.094	47.599	37.331	39.799	39.159
7	54.833	59.417	59.654	46.939	51.382	50.481
9	64.549	70.906	70.652	56.261	62.203	60.866
10	69.286	76.435	75.850	60.858	67.417	65.793
12	78.478	87.207	85.819	69.780	77.578	75.242
15	91.925	102.816	100.085	82.87	92.262	88.788
20	114.059	127.958	123.063	104.15	115.674	110.386

Table 7. Average Heat Transfer Coefficient Values for Various Velocities

The heat transfer coefficient values calculated for the flow rate of the selected fluid, as well as the heat transfer coefficient data found using Fluent are given in the table 7. As can be seen from the table, there are differences between the heat transfer values calculated by Fluent and the results of manual calculations using the formula. Ansys Fluent uses a reference Wall temperature when evaluating the heat transfer coefficient which cannot reflect the temperature change along the Wall. This is on one of the main reason of different values. But in general, the increase rate in the heat transfer coefficient compared to the flow rate is similar.

The obtained 2 different heat transfer coefficient data will be used separately in the calculations of nusselt correlations and performance evaluation criteria in the further step.

4.2 Nusselt Correlations

In Nusselt calculations, two separate correlations were used for smooth surface and modified surfaces. Gnielinski's correlationv (Eq. 4) generally preferred way to calculate Nusselt value for turbulent flow in tubes with smooth heat transfer surface. In addition, since factors such as the length and radius of the pipe are taken into account with the Gnielinski correlation, this correlation is used when calculating the Nusselt number of smooth surfaces.

$$Nu = \frac{\left(\frac{f}{8}\right)(Re-1000)\,Pr}{1+12{,}7\left(\frac{f}{8}\right)^{-\frac{1}{2}}\left(\mathrm{Pr}^{\frac{2}{3}}-1\right)}\left[1+\left(\frac{d}{L}\right)^{\frac{2}{3}}\right] \qquad \text{Eq.4}$$

In the equation given above, f is the Darcy friction factor that can either be obtained from the Moody chart or for smooth tubes from correlation developed by Petukhov (Eq.5)

$$f = (0{,}79\ln(Re) - 1{,}64)^{-2} \qquad \text{Eq.5}$$

The equation below (Eq.6) was used to calculate the Nusselt Number of the modified surfaces, where h is the convective heat transfer coefficient of the flow, L is the characteristic length, k is the thermal conductivity of the fluid.

$$Nu = \frac{h.L_{char}}{k} \qquad \text{Eq.6}$$

As seen in the Table 8 below, 2 different Nuselt Numbers were calculated for each variable. One of them was calculated using the heat transfer coefficient found by Fluent, and the other was calculated using the heat transfer coefficient calculated from the formula.

Velocity (m/s)	Nusselt Number from Fluent		Nusselt Number from Formula	
	Concave Dimples	Convex Dimples	Concave Dimples	Convex Dimples
4	33.452	33.828	27.216	27.195
5	38.921	39.338	32.892	32.363
7	49.105	49.301	42.465	41.719
9	58.600	58.390	51.408	50.303
10	63.169	62.686	55.717	54.374
12	72.072	70.925	64.115	62.184
15	84.972	82.715	75.250	73.378
20	105.751	101.705	95.599	91.228

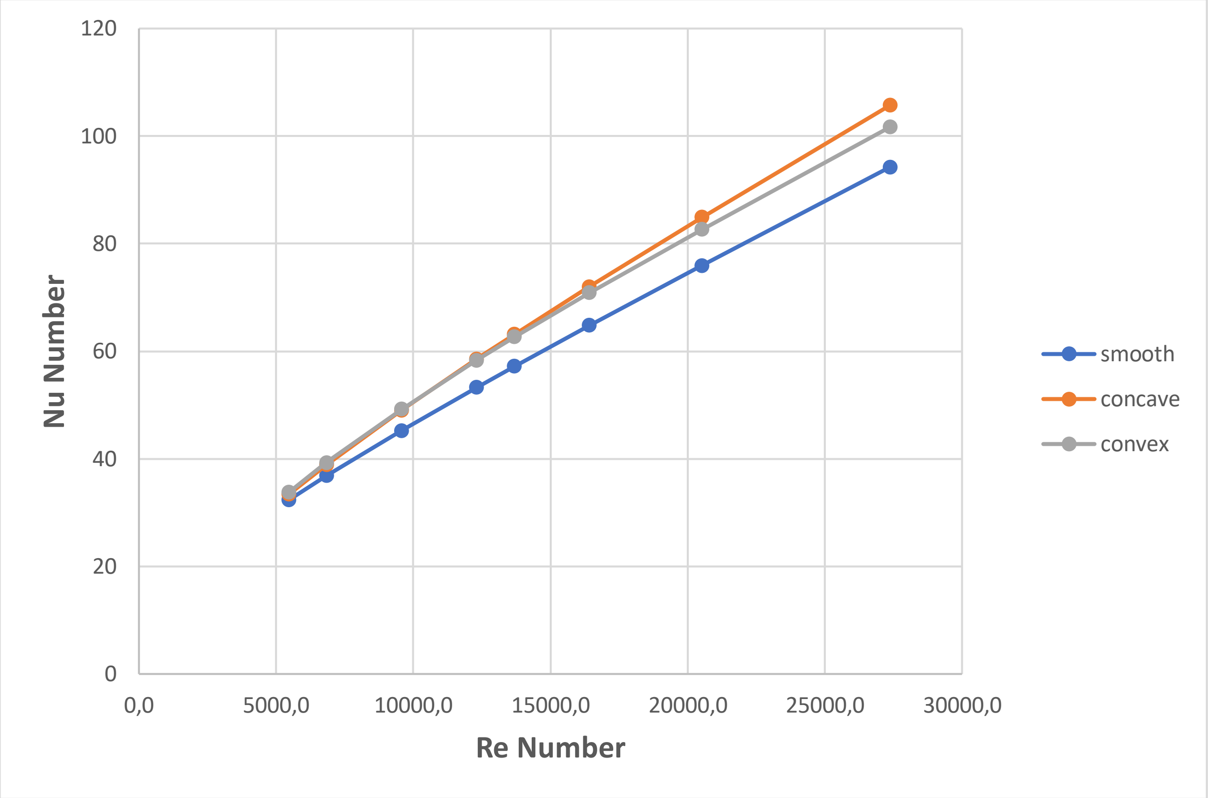
Table 8. Nusselt Numbers for Modified Surfaces

In addition, 2 different Nusselt numbers calculated for smooth surfaces are given in Table 9. One of them was calculated via Gnielinski correlation, which is calculated independently of the heat transfer coefficient, and the other was calculated using Eq 6, and the heat transfer coefficient data calculated via Fluent was used in this formula.

Velocity (m/s)	Gnielinski Correlation (Eq. 4)	Fluent
	Smooth Surface	
4	25.312	32.391
5	30.719	36.876
7	40.588	45.316
9	49.650	53.347
10	53.968	57.261
12	62.279	64.858
15	74.117	75.971
20	92.653	94.264

Table 9. Nusselt Numbers for Smooth Surface

In addition, Nusselt Numbers and Reynolds numbers obtained by fluent are given graphically below. By looking at this graph, we can easily say that; surfaces modified with concave and convex dimples have better heat transfer characteristics compared to smooth surfaces. In addition, if we compare concave and convex dimples, we can say that concave dimples provide slightly better heat transfer than convex ones. And this difference increases as the flow rate increases.



4.3 Evaluation of JF Factor

Heat transfer enhacement devices increase the rate of heat transfer , but they may also increase the pressure drop associated with the flow. Thus, the pressure drop constraint is a very important consideration for calculating the performance benefits of an enhanced surface in single-phase flow.

To evaluate the performance of the wall-jets and also select an optimum modified heat transfer surface, JF Factor performance evaluation criteria was employed.

The preferred evaluation method sets a performance objective (e.g., reduced surface area) and calculates the performance improvement relative to a reference design (e.g., smooth tubes) for a given set of operating conditions and design constraints (e.g., pressure drop).

A quantitative method is needed to evaluate the performance improvement provided by a particular development. Normally, the performance of an enhanced surface is compared to the performance of a corresponding flat (smooth) surface.

Performance evaluation criterion is consideration a comparative criterion, namely thermal hydraulic performance factor (JFi). Similar to the other evaluation criterions, this criterion is a 'the larger the better' parameter, and a high value of this criterion indicates a superior thermal hydraulic performance.

Definitions of the JF factor describing the performance advantages modified by dimples compared to the smooth surface are presented in Eq.7.

$$JF_i = \left(\frac{\frac{j_i}{j_{smooth}}}{\left(\frac{f_i}{f_{smooth}}\right)^{\frac{1}{3}}}\right) \qquad \text{Eq.7}$$

Where the subscripts 'i' and 'smooth' mean a type of the dimpled surface and smooth one as the reference channel or baseline, respectively.

The heat transfer characteristics can be illustrated in a non-dimensional form, namely Colburn factor.(Eq.8)

$$j = Nu/(Re\ \mathrm{Pr}^{\frac{1}{3}}) \qquad \text{Eq.8}$$

Where Nu is the Nusselt number, Re is Reynolds number, and Pr represents the Prandtl number.

Pressure drop characteristics examined in terms of the Fanning friction factor ang given in Eq.9.

$$f = \frac{2.\rho.D_h.\Delta P}{L.V^2} \qquad \text{Eq.9}$$

Where ρ is the working fluid density, ΔP is the pressure drop, and V is the mass velocity. The pressure drop value here represents the difference between the inlet and outlet values of the fluid in the pipe, and these values have been calculated by Fluent. Both pressure values are are-weighted average values.

The JF factor values calculated in the following Table 10 are given. Values are calculated separately for each flow rate. In addition, 2 different JF factor values were obtained according to 2 different Nusselt values calculated in the previous stage. Fanning friction factor and Colburn factor values are given for all variations in Table 10.

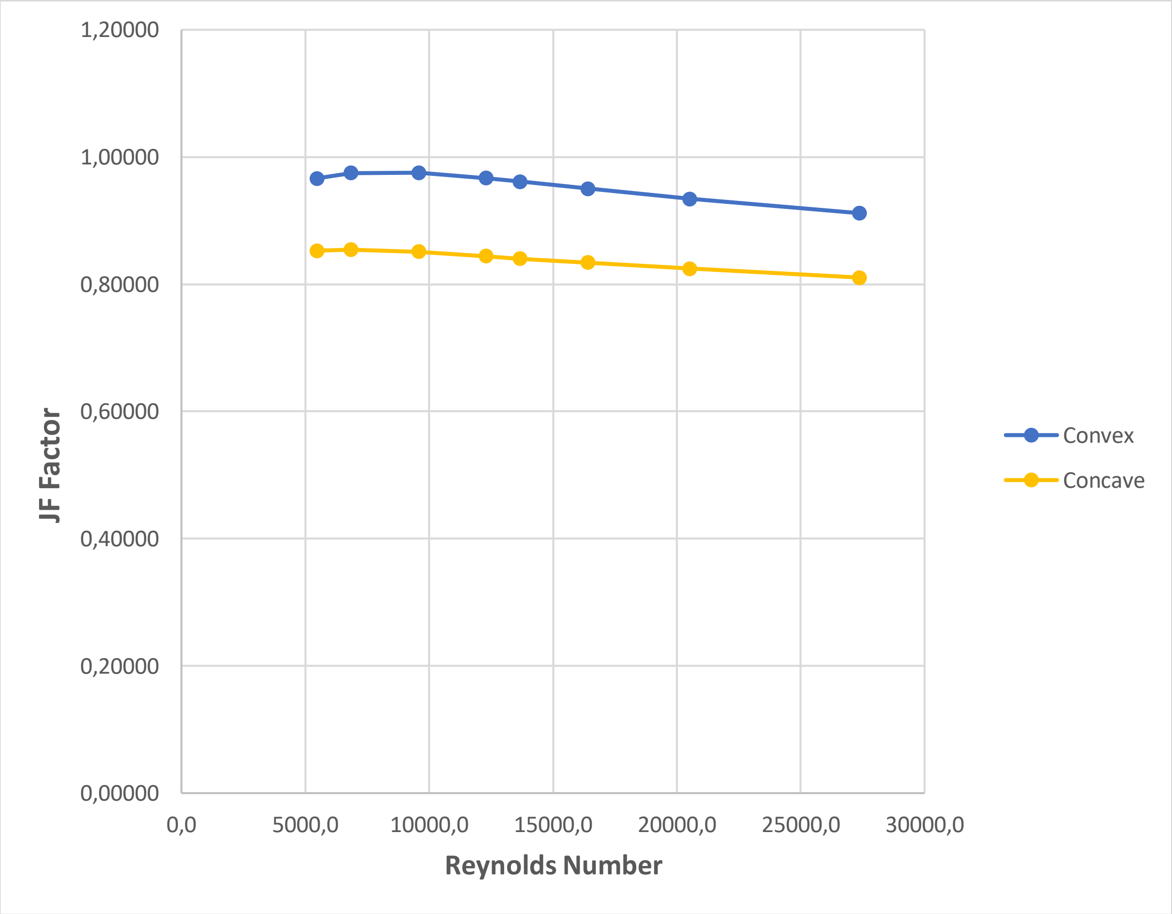
Velocity (m/s)	JF Factor (Fluent)		JF Factor (Formula)	
	Concave Dimples	Convex Dimples	Concave Dimples	Convex Dimples
4	0.85238	0.96627	0.84020	0.94112
5	0.85423	0.97498	0.86285	0.95869
7	0.85085	0.97531	0.85953	0.96411
9	0.84398	0.96685	0.84946	0.95563
10	0.84022	0.96129	0.84372	0.94931
12	0.83381	0.95040	0.83420	0.93712
15	0.82452	0.93414	0.82070	0.91921
20	0.81045	0.91164	0.80234	0.89551

Table 10. JF Factor Values for Modified Surfaces

If we consider the criterion that the larger value is better when interpreting the JF factor, it is clearly seen from the data in the table above that the convex dimples applied to the heat transfer surface affect the heat transfer more positively compared to the concave dimples.

The most important reason for this is that if we compare two different types of dimples, the heat transfer characteristics are similar, but the Fanning friction coefficient values of the concave dimples are much lower than the convex surfaces. Pressure drop values are calculated 40-50 percent higher for convex surfaces. In addition, if we compare the heat transfer improvements at different flow rates, optimum values are obtained at 5m/s for concave dimpled surface and 6m/s for convex dimpled surface. As the flow rate increases, the pressure drop value increases proportionally.

In addition, although the 2 different JF Factor values given for each flow rate in the table are not the same, they are very close to each other and do not affect the result while deciding the optimum result.



In the graph given above, the calculated JF values for both dimples types are given in proportion to the Reynolds number. Once again, we see that convex dimples give better results than concave dimples. Also, as seen in the graph, we cannot say that JF factor values increase in proportion to the Reynolds number. Optimum JF values for two different dimple types were calculated when Reynolds Number was 7500. After reaching this optimum value, we can say that the JF factor value decreases inversely as the Reynolds number increases.

Finally, as mentioned before, some simple simulations were tried by changing the pipe length. The aim is to investigate how the increase in pipe length affects the improvement in heat transfer. The JF Factor was recalculated for this comparison.

	Concave			Convex		
Velocity	Length=90 mm	Length=120 mm	Length=150 mm	Length=90 mm	Length=120 mm	Length=150 mm
5	0.8542	0.72618	0.80475	0.9749	0.91431	0.91470
10	0.8402	0.67520	0.76178	0.9612	0.87820	0.87852
20	0.8245	0.63638	0.72694	0.9341	0.84612	0.84725

Table 11. JF Factor Values for Various Pipe Lengths

When we look at the data given in Table 11, the effects of increasing the pipe length on heat transfer give interesting results. Considering the concave dimpled surface, although the 120 mm long surface gives worse results than the 90 mm, the 150 mm long surface gives better results than 120 mm. On the other hand, when we look at convex dimples, the effect of pipe length on heat transfer is very small. The calculated JF factor values for 120 mm and 150 mm are almost the same.

5. Conclusion

In many studies up to now, the effects of surface modification on heat transfer have been investigated, and as a result of these studies, surface modification techniques have been found to be very important in terms of improving heat transfer. As a result of the analyzes made in this study, firstly the effects of various shapes of dimples applied to the heat transfer surfaces on heat transfer were investigated for wall jet flow and impinging jet flow.

These examinations were made on 2D geometries for wall jet simulations and on both 2D and 3D geometries for impinging jet simulations. According to the average heat transfer coefficients calculated as a result of the impinging jet simulations, all modified surfaces reduced the heat transfer compared to the smooth surface. Wall jet simulations were examined in more detail in this study. First of all, heat transfer surfaces modified with dimples of different geometries were investigated. As a result of these examinations, it was found useful to make modifications to improve heat transfer on wall jet surfaces. Especially the effect of concave and convex spherical dimples on heat transfer is visible.

Additional simulations were then made for surfaces modified with convex and concave dimples. In these simulations, it was aimed to examine the heat transfer behavior dependency on the flow rate with dimpled surfaces. As a result of all simulations, the average heat transfer coefficient on the heat transfer surface was calculated by means of Fluent. For comparison, JF Factor calculations, which is a performance evaluation criterion, were made.

The JF factor was used to examine the overall efficiency of the system, as it also takes into account the energy loss due to the pressure drop in the system. JF factor values for convex dimpled geometries seem to be better compared to concave ones. However, in terms of heat transfer characteristics, the result is the opposite that concave dimpled surface provides better heat transfer rates compared to convex one but also larger pressure loses.

As a result, it can be said that spherical dimples applied convex or concave to the surface can significantly effect heat transfer, but correct geometry and optimum values are very important for this. Many variables such as the depth, size, number, location of the dimple applied to the surface, and the distances between the dimples can affect the heat transfer positively or negatively. For this reason, it is possible to reach the most optimum parameters by trying many simulations.

References

[1] Dafedar, M.A, Mujtabalayeeq, I., Mohemmed, Taher M., & Shahid, Mohammad Idressurf (2013). *Heat Transfer Enhancement Through Different Circular Diametrical Dimple Surface Under Forced Convection –An Experimental Approach.* IJRET: International Journal of Research in Engineering and Technology e ISSN: 2319-1163 | ISSN: 2321-7308, Volume: 02 Issue: 07.

[2] Iftikarahamad, H. Patel, Dr. Sachin, L. Borse, (2012). *Experimental Investigation Of Heat Transfer Enhancement Over The Dimpled Surface*. International Journal of Engineering Science and Technology (IJEST), ISSN: 0975-5462 Vol. 4 No.08

[3] Turnow, J., Kornev, N., Isaev, S., Hassel, E., 2010. *Vortex mechanism of heat transferenhancement in a channel with spherical and oval dimples*. Heat Mass Transfer47, 301–313 .

[4] Ligrani, P.M., Oliveira, M.M., & Blaskovich, T., (2003). *Comparison of heat transfer augmentation techniques*. 41(3):337–372.

[5] https://www.hollowayamerica.com/products/dimple-overall-heat-transfer-surface/

[6] https://teralba.com/dimple-technology/

[7] Akthar, Faheem, Kaladgi, Abdul Razak R., & Dafedar, Mohammed Samee A. *Heat Transfer Enhancement Using Dimple Surfaces Under Natural Convection-An Experimental Study.* ISSN 2278 Vol. 4, No. 1. 2015

[8] Liu, S., Sakr, M., A "*Comprehensive Review on Passive Heat Transfer Enhancements in Pipe Exchangers*", Renewable and Sustainable Energy Reviews, 19, 64-81, 2013.

[9] Nagarani, N., Mayilsamy, K., Urugesan, A., Kumar, S.G., "*Review of Utilization of Extended Surfaces in Heat ransfer Problems*", Renewable and Sustainable Energy Reviews, 29, 604-613, 2014.

[10] Kraus, A., Azız, Ab., Welty, J., "*Extended Surface Heat Transfer*'', John Wıley &Sons, INC., 2001, USA.

[11] Wang, F., Zhang, J., Wang, S., "*Investigation on Flow and Heat Transfer Characteristics in Rectangular Channel with Drop-Shaped Pin Fins*", Propulsion and Power Research, 1, 64-70, 2012.

[12] Aylı ,Ece, İnce , Ender "*Review of Enhancement of Heat Transfer From Rectangular Fin Arrays*" , December 2018

[13] Kumar, S., Nandhakumar, A. J. D., & Prabhu, S. (2013). *Analysis of natural convective heat transfer of nano-coated aluminium fins by using Taguchi method.* Journal of Heat and Mass Transfer, vol. 49, no. 1, pg. 55-64.

[14] Gee, D.L.; Webb, R.L. *Forced Convection Heat Transfer in Helically Rib-Roughened Tubes*. Int. J. Heat Mass Transf. 1980, 23, 1127–1136.

[15] KELEŞOĞLU, Bülent *Yön Değiştiren Kanal Akışlarında Isı Transferinin İncelenmesi*, 2010.

[16] Afanasyev, V.N. , Chudnovsky, Y.P. , Leontiev, A.I. Roganov, P.S. *Turbulent flow friction and heat transfer characteristics for spherical cavities on a flat plate*, Exp Therm Fluid Sci, 7 (1) (1993), pp. 1-8

[17] Mahmood, G.I. , Hill, M.L. , Nelson, D.L. , Ligrani, P.M. , Moon, H.K. , Glezer, B. *Local heat transfer and flow structure on and above a dimpled surface in a channel*, J Turbomach, 123 (1) (2001), pg. 115-123

18] Ligrani, P.M. , Harrison, J.L. , Mahmmod, G.I. , Hill, M.L. *Flow structure due to dimple depressions on a channel surface*, Phys Fluids, 13 (11) (2001), pg. 3442-3451

[19] Burgess, N.K. , Ligrani, P.M. *Effects of dimple depth on channel Nusselt numbers and friction factors*, J Heat Transfer, 127 (1) (2005), pg. 839-847

[20] Wang, Y. , He, Y.L. , Lei, Y.G. , Zhang, J. *Heat transfer and hydrodynamics analysis of a novel dimpled tube*, Exp Therm Fluid Sci, 34 (8) (2010), pp. 1273-1281

[21] Chyu MK, Yu Y, Ding H, Downs JP, Soechting FO. *Concavity enhanced heat transfer in an internal cooling passage*. ASME Paper 97-GT-437; 1997

[22] *Principles of Enhanced Heat Transfer*, Webb, Ralph L. , Kim, Nae-Hyun 2nd edition

[23] O'Neill P. S. , Gottzman, C. F. and Terbot, J. W. *Adv. Cryogenic Eng*.,17 (1972).

[24] https://www.refrigeration-ag.com/en

[25] https://www.htt-ag.com/en/dimple-plate/storage-and-tempering-of-liquids