International Refrigeration and Air Conditioning Conference

School of Mechanical Engineering

2022

Additive Manufacturing For Thermal Management Applications: From Experimental Results To Numerical Modeling

Giacomo Favero

Massimiliano Bonesso

Pietro Rebesan

Razvan Dima

Adriano Pepato

See next page for additional authors

Favero, Giacomo; Bonesso, Massimiliano; Rebesan, Pietro; Dima, Razvan; Pepato, Adriano; and Mancin, Simone, "Additive Manufacturing For Thermal Management Applications: From Experimental Results To Numerical Modeling" (2022). *International Refrigeration and Air Conditioning Conference.* Paper 2462. https://docs.lib.purdue.edu/iracc/2462

Authors

Giacomo Favero, Massimiliano Bonesso, Pietro Rebesan, Razvan Dima, Adriano Pepato, and Simone Mancin

Additive Manufacturing for Thermal Management Applications: from Experimental Results to Numerical Modelling

Giacomo FAVERO[1,2,]*, Massimiliano BONESSO[2,3], Pietro REBESAN[2,4], Razvan DIMA[2], Adriano PEPATO[2], Simone MANCIN[1,2]

[1]Dept. of Management and Engineering, University of Padova, Str.lla S. Nicola 3, 36100 Vicenza, Italy

[2]I.N.F.N. - Sezione di Padova, Via Marzolo 8, 35131, Padova, ITALY

[3]Dept. of Industrial Engineering, University of Padova, Via Marzolo 9, 35131, Padova, Italy

[4]Dept. of Mechanical Engineering, Politecnico di Milano, Via La Masa, 1, 20156 Milan, Italy

* Corresponding Author

ABSTRACT

Additive Manufacturing (AM) of copper and copper alloys has opened new frontiers in heat transfer applications, going beyond the capabilities of conventional technologies. Despite the great design freedom offered by AM, when dealing with metal powders, a few issues should be considered to exploit the great capabilities of this manufacturing technology. In fact, the surface roughness of the components is expected to affect the performance of the devices, which can be remarkably different from the ones simulated with software. This paper presents a critical analysis of the accuracy of the numerical tools to simulate the fluid flow behaviour inside cooling channels obtained via AM. The work shows the major limitations of the standard approaches to accurately predict the pressure drops in straight and complex channels. Three different copper channels of growing complexity were built via LPBF (Laser Powder Bed Fusion) and then they were experimentally tested at different water flow rates to evaluate the predictive abilities of the numerical model. The results revealed that the surface roughness deeply affects the fluid flow behaviour, thus the numerical models need to be calibrated to become a reliable design tool. The proposed procedure can be considered the first attempt in this direction and allows for a proper integration of the AM with the numerical simulation tools, to boost the design capabilities of LPBF technology.

1. INTRODUCTION

Additive Manufacturing is a technology process, which intrinsically demonstrates many advantages and a few limitations. It has found wide use in fast prototyping for many years due to its versatility and limited costs per unit. More recently, this technology has involved the manufacturability of new components, using metallic materials, for example in aerospace and automotive applications. Wrobel *et al*. (2020) designed and characterized the AM aluminium alloy metal to build a heat exchanger for high-altitude aircraft. The compactness and lightness are required properties and AM technology is able to satisfy these requirements. The process uses a method of layers, where each one is a thin cross-section of the components generated by a computer-aided design (CAD) file. The material is stratified in different ways depending on the AM technology. LPBF consists of a metal powder melting with the use of a high power laser. AM technology's remarkable advantage is the possibility of realizing complex channels directly inside the component without assembling external cooling pipes. The direct integration of the cooling channels in the final component is becoming a key enabling technology in many different applications, among those: aerospace, automotive, biomedical, electronic thermal management, robotics, and energy. Arie *et al*. (2018) carried out an experimental investigation about heat exchangers for power plants and observed a 30%-40% improvement as compared to conventional wavy fins, louvered fins, and plain fins.

However, AM enables the design of novel internal enhanced surfaces using pins or lattice structures inside channels to improve the heat transfer coefficient. Brooks and Brigden (2016) investigated the use of AM to produce cooling channels with lattice internal structures showing interesting results. Among the different fields, fusion energy is an important topic of research in which the cooling components are relevant because the heat fluxes to be handled are an order of magnitudes higher as compared to common applications and the geometrical constraints limit the design possibilities of the traditional manufacturing technologies. The AM is an attractive solution that is under investigation and studied: for example, Seltzman and Wukitch (2020) developed and tested lower hybrid current drive RF launchers manufactured via LPBF of copper alloy.

The Divertor Tokamak Test facility (DTT) is an Italian project aimed at testing the physics and technology of alternative power for DEMO and ITER reactors (Ambrosino *et al*. 2021). In DTT, the neutral beam injector (NBI) is an important system that accelerates the deuterium beam and gives it the correct direction inside the tokamak reactor. To fulfil these functions NBI is equipped with acceleration grids, which are involved by huge localized heat fluxes. To avoid fatigue structural failures of acceleration grids, cooling channels inside grids are useful to keep the temperature under critical values. Agostinetti *et al*. (2016) designed optimized cooling channels for MITICA (Megavolt ITER Injector & Concept Advancement) grids, with Nozzle Island Cooling Enhancement (NICE) shape (Fig. 1). This optimization, which requires a complicated process chain, considered conventional technologies but it can be considered a challenging AM application.

Although this fast increase in the applicability of the cooling channels, the prediction of fluid flow can become tricky and lead to incorrect results. Kirsch and Thole (2018) studied and separated the effect of roughness and wall shape, comparing the k-ε turbulence model between LPBF and SLA (Stereolithography). The effect of roughness is a complex issue, due to the lack of control of the roughness of the manufactured metal surfaces. The current CFD tools simulate the channel as smooth ones and allow to specify the actual roughness but, unfortunately, the roughness of 3D printed channels is not known *a priori*, because the shape is complex and it is not easy to measure internal surfaces. Thus, the predictive capabilities of these numerical tools are typically unsatisfactory.

In this paper, MITICA grids cooling channels are considered to prepare novel suitable channels designed for AM. This paper critically studies the reliability of the numerical tools to simulate the fluid flow behaviour inside channels obtained via AM, showing the major limitations of the standard approaches to accurately predict the pressure drops in straight and complex channels. The turbulent flow of water inside different channels of increasing geometry complexity, built via LPBF is experimentally investigated and the collected results are used to calibrate the numerical models. The proposed novel procedure to design cooling channels based on metal AM can become a general guideline to avoid possible misleading results due to the still unpredictable surface roughness of the channels obtained via AM.

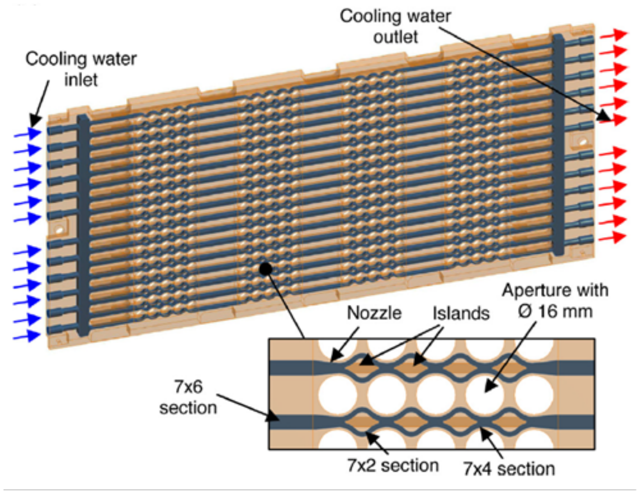

Figure 1: Acceleration Grid of MITICA experiment (Agostinetti *et al*., 2016) with the layout of the cooling system and different available sections of cooling channels.

2. MATERIALS AND METHODS

Klingaa *et al*. (2020) reported that the surface roughness of AM samples depends on the orientation of internal surfaces with respect to the build direction. In the present study, all the different channel geometries were manufactured keeping the same build direction. A vertical build orientation was selected, because, as stated by Klingaa *et al*. (2020), it leads to the lowest and most uniform surface roughness in a straight channel. This study takes part in the framework of the fusion energy program and it aims at developing an efficient cooling management system embedded in the DTT acceleration grid (**Errore. L'origine riferimento non è stata trovata.**). Different channel geometries were adopted for AM technology to maximize the heat transfer to reject the huge amount of heat generated by particles beams when passing through the grid, inspired by Agostinetti *et al*. (2016). **Errore. L'origine riferimento non è stata trovata.** reports a drawing of the acceleration grid of the MITICA experiment, the zoom of a single channel with the NICE design is reported with main quotes. The cross sectional area of NICE for the MITICA channel is a rectangular cross section of 4x7 mm. In this study, the reference channel can be considered the trade-off between the maximum dimensions available from MITICA design to improve heat transfer area and the minimum fillet radius to minimize the pressure drop. Fig. 2 reports a cross section for the reference straight channel design in this study.

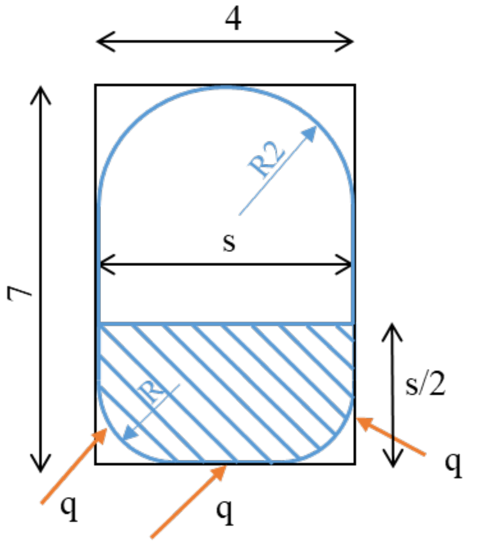

Figure 2: Cross section of straight reference cooling channel. In black, the MITICA cooling channel design section, and in blue the section developed for this study are reported.

2.1 Samples manufacture

As shown in Fig. 3, the different channels were manufactured including two pressure taps (p_1 and p_2) which were used to join the samples to the differential pressure transducer. As it can be noted from the drawings of the three channels (Fig. 3b-c-d), the pressure taps are located 23 mm after the inlet and before the outlet to measure only the frictional pressure drops, avoiding the abrupt contraction and expansions pressure drops. The NICE and duned shapes are tuned to AM design, thus the NICE geometry is different from MITICA one.

The reference straight channel (Fig. 3b) was used to identify experimentally the actual absolute surface roughness ε and calibrate the developed numerical model. Differently, the other two channel geometries, with increased complexity named NICE (Fig. 3c) and duned (Fig. 3d), which are meant to maximize the heat transfer area and the heat transfer performance of the cooling management system, were used to evaluate the predictive capabilities of the developed numerical tool and to tune the proposed design procedure. The reference straight channel presents a hydraulic diameter d_h=5.31 mm, while the duned and NICE ones are equal to 5.24 mm and 3.92 mm, respectively. Table 1 reports the most important geometrical characteristics of the samples.

The samples were manufactured in a vertical direction with an EOSINT M280 machine, using pure copper powder (Cu 99.95%) with a distribution of D10=8.1 µm, D50=18.7µm, and D90=36.4µm. Machine parameters were set to guarantee the highest material density, as reported by Bonesso *et al*. (2020): layer thickness of 20 µm, hatching distance of 0.09 mm, laser power 370 W, and a scan speed of 400 mm/s. The inert gas was nitrogen and the oxygen concentration in the chamber was controlled to be always less than 0.5%.

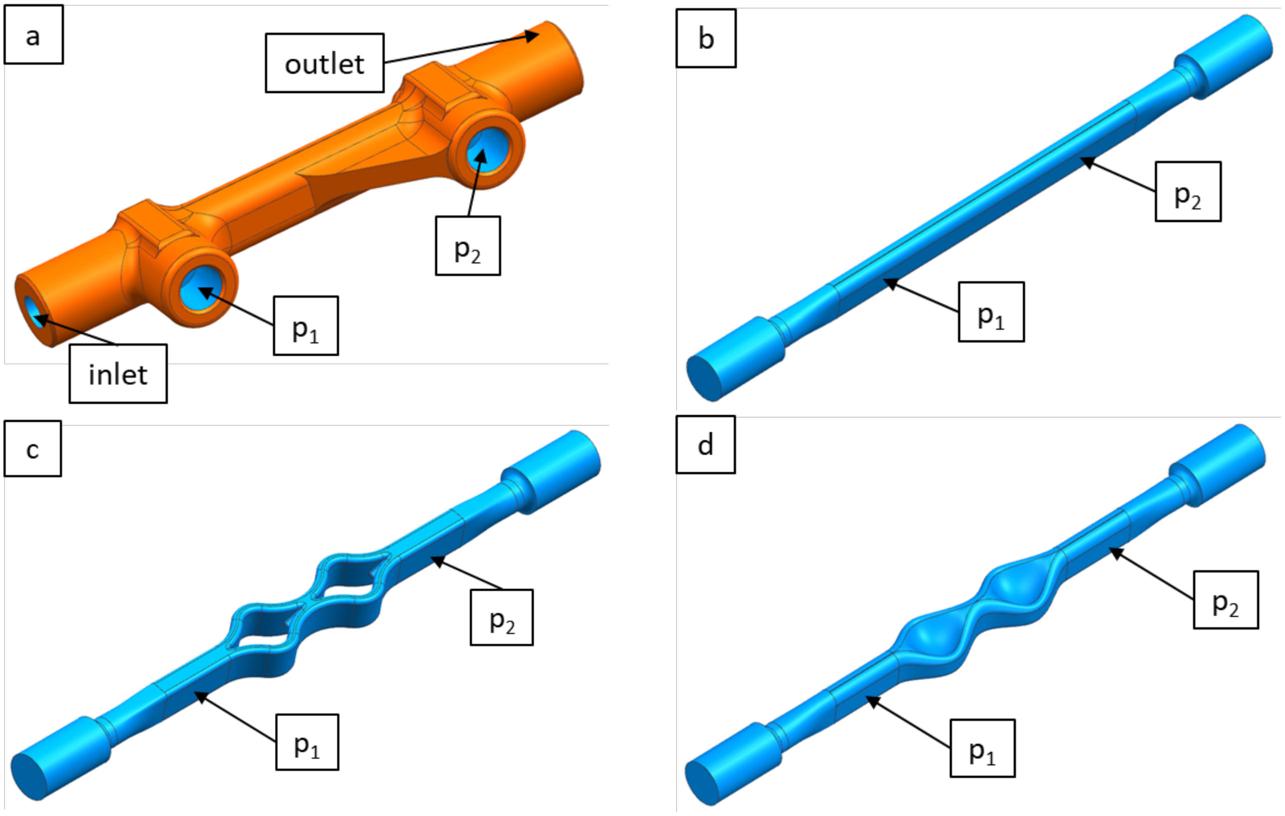

Figure 3: Channels shapes. In (a) the external samples layout. In (b), (c), and (d) the internal channel shape of reference straight, NICE, and duned channel respectively.

Table 1: Main geometrical characteristics of samples.

	Reference straight channel	**duned channel**	**NICE channel**
Wetted area (p_1-p_2) [mm^2]	1061	1385	1647
Fluid flow volume (p_1-p_2) [mm^3]	1409	1814	1612
Measuring distance (p_1-p_2) [mm]	54	54	54
Total sample length [mm]	100	100	100
Cross sectional area at p_1 [mm^2]	26.09	26.09	27.45
Wetted perimeter at p_1 [mm]	19.64	19.64	20.63
Hydraulic diameter [mm]	5.31	5.24	3.92

2.2 Experimental setup

The hydraulic tests were run in an experimental setup located at the Department of Management and Engineering of the University of Padua. The test bench comprises a water loop in which the fluid temperature and flow rate can be independently set and controlled. The test bench is equipped with a differential pressure transducer with an uncertainty (k=2) of ±0.065% of the full scale (FS= 1 bar), while the water temperature was monitored using calibrated T-type thermocouples with an accuracy of ±0.05 K. The tests were run by varying the water flow rate from 1 to 14 l/min and multiple curves were collected by increasing and decreasing the water flow rate, to successfully verify the repeatability of the results.

3. EXPERIMENTAL RESULTS

In Fig. 4, the experimental results collected for the three different channels are reported. Pressure drop measured values are in agreement with a second-order polynomial function with respect to water flow rates.

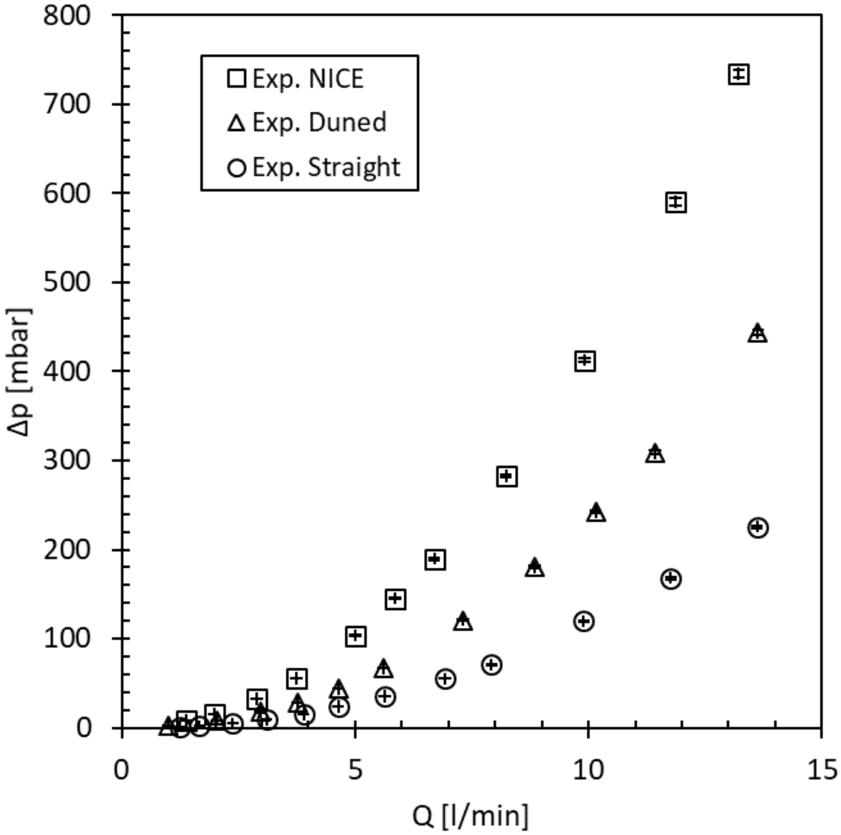

Figure 4: Pressure drop test results plotted with respect to water flow rate.

The flow regime depends on the Reynolds number, which is given by:

$$Re = \frac{\rho u d_h}{\mu} \tag{1}$$

The tests were run at a mean water temperature of 28.5°C and the related thermophysical properties are ρ=996.03 kg m^{-3} and μ=825.125 μPa s. Thus, in the experimental tests, the Reynolds number varies from 5080 to 55765. These values ranged from the transitional flow regime (i.e. typically between Re=2000 and Re=10000) to turbulent flow (i.e. Re>10000).

The friction factor *f* of the reference straight channel may be computed from the measured values of pressure drop and water flow rate with the Darcy equation (2):

$$\Delta p = f \frac{l}{d_h} \rho \frac{u^2}{2} \tag{2}$$

considering the flow length equal to the distance between the two pressure taps, l=54 mm, and computing nominal velocity u on the known cross sectional area from Table 1. In **Errore. L'origine riferimento non è stata trovata.**, the values of the experimental friction factor as a function of the Reynolds number are reported for the straight channel. A detailed error analysis was performed following Kline and McClintock (1953); it was estimated that the uncertainty (k=2) on the values of the friction factor was less than ±2.0% for Re>28000, below ±7.3%, for 28000<Re<12000 and then it increases being ±13.0% at Re=9800, ±31.4% at Re=6850, and ±68% at Re=5100.

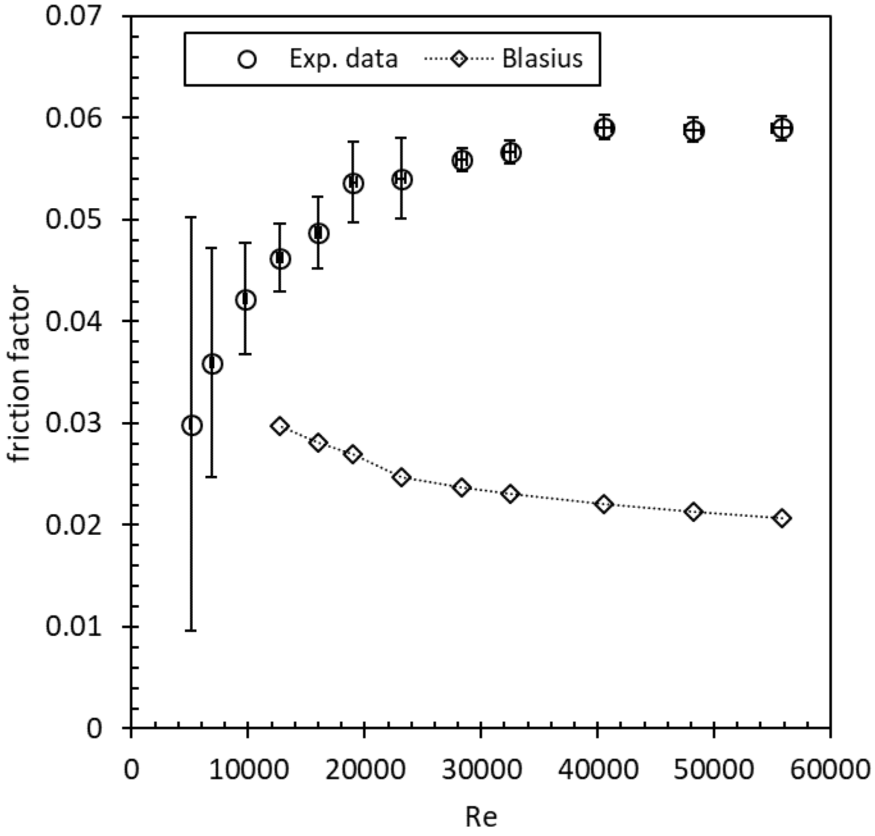

Figure 5: Friction factor of reference straight channel, from experimental results and Blasius theoretical model.

The same Fig. 5 also reports the values of the friction factor evaluated with the Blasius (1908) equation for smooth channels:

$$f = 0.184\, Re^{-0.2} \quad \text{if} \quad Re > 20000 \tag{3}$$

$$f = 0.3164\, Re^{-0.25} \quad \text{if} \quad 10000 < Re < 20000 \tag{4}$$

It clearly appears that the experimental friction factors show a completely different behaviour as compared to those computed for the equivalent smooth straight one. As described before, this is due to the channel obtained via AM is not "smooth" and it presents a non-negligible surface roughness, which deeply affects the fluid flow. The Reynolds number varies from 5000 to 50000; in this range for rough channels, the transition between laminar to turbulent implies an increase of the friction factor, which then tends to a constant value that depends on the value of the relative roughness, as also reported in Moody diagram (1945).

The main idea of the method proposed here is to use the experimental results collected for the reference straight channel to estimate the wall absolute roughness ε and then calibrate the numerical model. As it can be seen from Fig. 5, for Reynolds greater than 40000, the friction factor becomes almost independent from Reynolds number and it tends to a value around 0.059, which is almost three times higher than that calculated with Blasius equations.

In this work the Colebrook-White model (1939), with equation (5), can be used to evaluate the wall absolute roughness ε of the channel:

$$\frac{1}{\sqrt{f}} = -2\log\left(\frac{\varepsilon}{3.71\, d_h} + \frac{2.51}{Re\sqrt{f}}\right) \tag{5}$$

The average computed value of wall absolute roughness is 0.168 mm at Reynolds numbers 40564, 48185, and 55765.

4. NUMERICAL MODEL

4.1 Numerical model development and validation

The application of AM is taking place in many sectors and is having a huge diffusion in the heat transfer field. However, when dealing with metal AM, the main properties of the material and surface (e.g. thermal conductivity, surface roughness) may largely differ from the expected values. This can lead to undesired results; Fig. 5 reports the friction factor, which can be more than 2 times lower as compared to the experimental one.

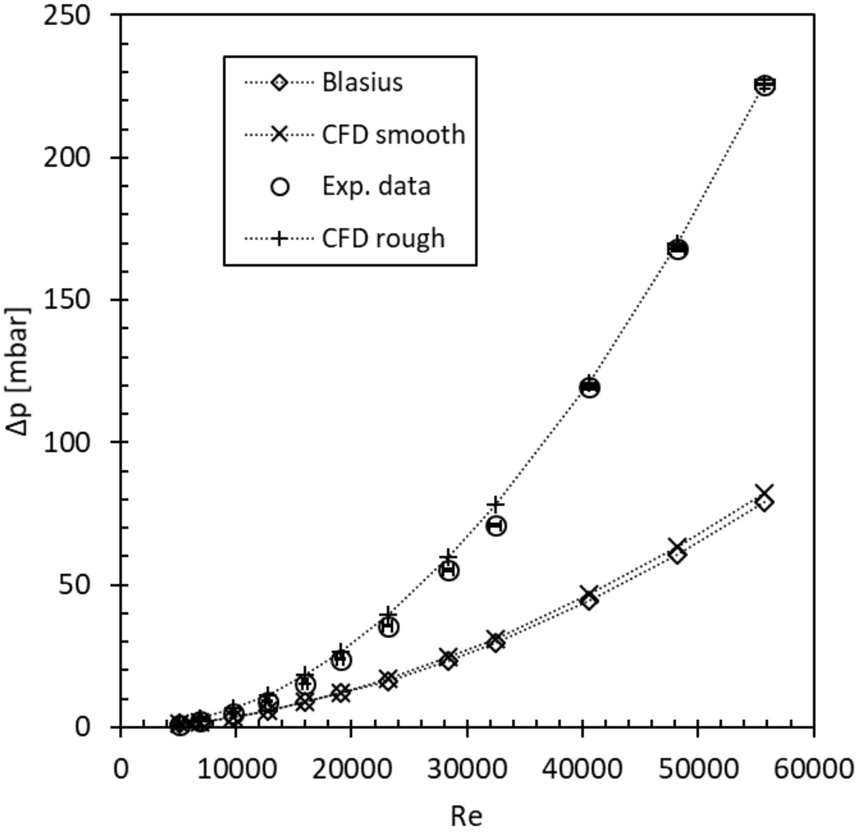

Figure 6: Pressure drop of reference straight channel: experimental data, data evaluated for the smooth channel with Blasius equations (5) and (6), CFD results for smooth and rough channels after calibration.

Ansys Fluent v20.1 software was used to develop the numerical model. The flow was assumed to be three-dimensional, steady-state and incompressible, with constant thermo-physical properties of the fluids which were estimated at the experimental water mean temperature of 28.5°C. Tetrahedral element type was used to better mesh complex geometry of different channel shapes. The k-ε turbulence model was adopted to solve Navier-Stokes and continuity equations. The k-ε turbulence model is a well-known, reliable turbulence model, which brings the simulations solution to convergence.

A mesh sensitivity analysis was run considering the reference straight channel and a deviation below 1% was chosen as a threshold in pressure drop computation. Three meshes were investigated: the first one with 159515 elements, the second with 261795 elements, and the third with 383891 elements. The relative deviations between the estimated pressure drops were -2.9% and -0.7%, respectively.

The model was first validated against the data calculated with the Blasius equations (5) and (6); as reported in **Errore. L'origine riferimento non è stata trovata.**, the developed model was able to fairly predict the calculated results with an average deviation of 4.9%.

Boundary conditions of the numerical model are set for the highest possible water flow rate: at the inlet mass flow rate, $\dot{m}$=0.229 kg/s, at the outlet (pressure value is zero), sand grain roughness height value of 0.168 mm that was calculated from the experimental results. As described before, the Standard k-ε (SKE) turbulence model with Scalable Wall Function (SWF) was selected.

As shown in Fig. 6, the calibrated numerical model can accurately predict the experimental data for Re>40000, when the friction factor tends to a constant value. For lower Reynolds numbers, the relative deviations slightly increase but the results are satisfactory. It is worth highlighting that the rating operating conditions for the cooling management system of the electron beam grids set Reynolds numbers greater than 40000. The experimental and numerical analyses were extended to lower values of the Reynolds number to understand the feasibility and the accuracy of the proposed method. In any case, considering the uncertainty analysis reported before, numerical results show an excellent agreement.

4.2 Numerical application

The proposed method allowed for the calibration of the numerical model for the reference straight channel leading to fair results, but this work aims to identify a suitable procedure to accurately estimate the fluid flow and pressure drops in AM devices, in which the channels are commonly complex and integrated into the system. Thus, the case study proposed here can give huge information about the suitability of the numerical models to be reliable design tools. For these reasons, using the same mesh parameters, same turbulence model and boundary conditions (i.e. maximum water flow rate at the inlet), and imposing at the wall the experimentally estimated value of surface absolute roughness, the values of the pressure drop for the other two samples were numerically estimated. The results revealed that when dealing with complex channels, the computed relative deviations were not negligible, being -23.3% and -30.1% for duned and NICE channels, respectively. In the entire experimental results, the calibrated model tends to underestimate the pressure drops measured for these channels and this can be related to both the different geometrical properties of the channels and to a different wall absolute roughness on the non-vertical walls. It can be stated that the calibrated model for the reference straight channel is neglecting some physical interactions.

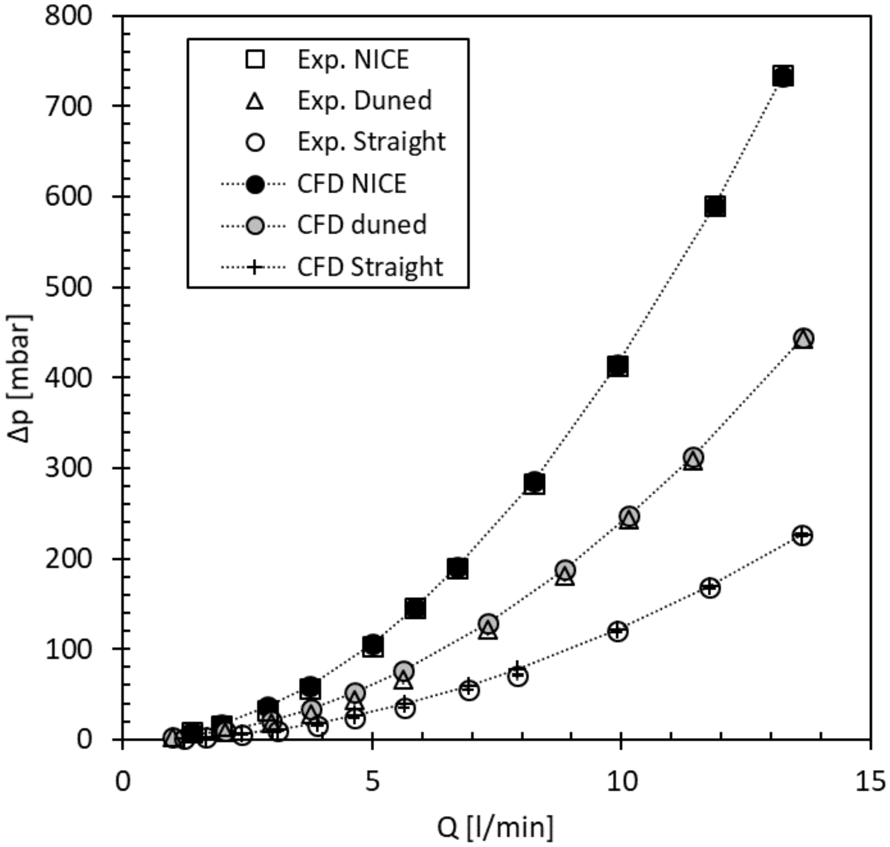

Figure 7: CFD, after single calibration of wall roughness for each channel, and experimental results of reference straight, duned, and NICE channel.

Considering a Non-Equilibrium Wall Function (NEWF) with the SKE turbulence model, some more accurate results were obtained. NEWF emphasizes the channel geometrical effect in the turbulent vortices but does not keep the roughness effect. The results are better in the duned channel, and different in the NICE one. In the duned channel, for Re>30000 the maximum deviation is -8.7% compared with the experimental pressure drop. Instead, in the NICE channel, the pressure drop results are accurate just for Re<20000, but not where the turbulent regime increases: the "roughness effect" in experimental data is present, while SKE with NEWF is not able to predict it.

The surface roughness of the samples built via AM depends upon the different orientations that surfaces create with the main build direction. This means that the wall roughness is not uniform in all the duned and NICE internal surfaces. Since it might not be possible to estimate the absolute surface roughness for all the non-vertical surfaces, a simplified procedure can be suggested to improve the predicting capabilities of the numerical model. In the case of NICE and duned channels, for the non-vertical surfaces, an average value of the absolute roughness is assigned on basis of the calibration of the pressure drop at the maximum Reynolds number. For the vertical surfaces, the previous value of 0.168 mm for absolute roughness was kept.

As reported in Fig. 7, following this procedure, the numerical model has an acceptable variability with experimental results for all the investigated channels.

It came out that for the duned channel, using an SWF and SKE, a wall absolute roughness value of 1.25 mm was computed for the non-vertical surfaces. For example, the deviation at the maximum water flow rate was 0.14%, while considering the entire data set, the average value of the relative deviation was 7.8%.

Considering the NICE channel, a value of 0.62 mm for the wall absolute roughness was estimated for the non-vertical surfaces. Similarly, to what has already been presented for the duned channel, the SKE model with SWF is able to accurately predict the experimental pressure drop, showing a relative deviation of -0.23% at the highest water flow rate. Considering the entire data set, the average value of the relative deviation was 2.5% and the maximum deviation was 9.7%.

5. CONCLUSIONS

This paper presents the capabilities and limitations of the numerical tools to simulate the fluid behaviour inside complex shape cooling channels made via AM. Two wall functions were investigated and the results were compared to choose the best prediction of experimental data.

A novel calibration method was developed to tune a numerical CFD model, starting from the pressure drop measurements collected for turbulent flow. From the experimental pressure drops collected for a reference straight channel, the values of the absolute surface roughness ε can be estimated using the Colebrook-White model by equation (5). This value can be used in the SWF parameters applied to the SKE turbulence model. The CFD results revealed that an accurate prediction of the experimental results for the straight channel can be achieved with a relative deviation that becomes negligible for Re>30000 and it can be considered more than satisfactory at lower Reynolds numbers.

When applying the described procedure to the complex channels, which presented wall surfaces that were not vertical but inclined as compared to the build direction, the calibrated numerical model tended to underestimate the experimental results. The numerical analysis with different wall treatments revealed that the orientation of the surfaces deeply affects the wall roughness and, thus, the predicting capabilities of the numerical models.

Thus, the proposed calibration procedure suggests to evaluate a different average absolute wall roughness for the channel non-vertical surfaces, which once set at the SWF of the SKE, led to accurate predictions of the experimental pressure drops.

REFERENCES

Wrobel, R., Scholes, B., Hussein, A., Law, R., Mustaffar, A., & Reay, D. (2020). A metal additively manufactured (MAM) heat exchanger for electric motor thermal control on a high-altitude solar aircraft – Experimental characterisation. *Thermal Science and Engineering Progress*, *19*, 100629. https://doi.org/10.1016/j.tsep.2020.100629

Kirsch, K. L., & Thole, K. A. (2018). Isolating the effects of surface roughness versus wall shape in numerically optimized, additively manufactured micro cooling channels. *Experimental Thermal and Fluid Science*, *98*, 227-238. https://doi.org/10.1016/j.expthermflusci.2018.05.030

Arie, M.A., Shooshtari, A.H., & Ohadi, M.M. (2018). Experimental characterization of an additively manufactured heat exchanger for dry cooling of power plants. *Applied Thermal Engineering*, *129*, 187–198. https://doi.org/10.1016/j.applthermaleng.2017.09.140

Brooks, H., & Brigden, K. (2016). Design of conformal cooling layers with self-supporting lattices for additively manufactured tooling. *Additive Manufacturing*, *11*, 16–22. https://doi.org/10.1016/j.addma.2016.03.004

Seltzman, A.H., & Wukitch, S.J. (2020). Surface roughness and finishing techniques in selective laser melted GRCop-84 copper for an additive manufactured lower hybrid current drive launcher. *Fusion Engineering and Design*, *160*, 111801. https://doi.org/10.1016/j.fusengdes.2020.111801

Ambrosino, R., *et al*. (2021). DTT - Divertor Tokamak Test facility: A testbed for DEMO. *Fusion Engineering and Design*, *167*, 112330. https://doi.org/10.1016/j.fusengdes.2021.112330

Agostinetti, P., Chitarin, G., Gambetta, G., & Marcuzzi, D. (2016). Two key improvements to enhance the thermo-mechanic performances of accelerator grids for neutral beam injectors. *Fusion Engineering and Design*, *109-111*, 890-894. http://dx.doi.org/10.1016/j.fusengdes.2016.01.056

Klingaa, C.G., Dahmen, T., Baier, S., Mohanty, S., & Hattel, J.H. (2020). X-ray CT and image analysis methodology for local roughness characterization in cooling channels made by metal additive manufacturing. *Additive Manufacturing*, *32*, 101032. https://doi.org/10.1016/j.addma.2019.101032

Bonesso, M., Rebesan, P., Calliari, I., Mancin, S., Gennari, C., Pepato, A., Dima, R. (2020). Effect of particle size distribution on laser powder bed fusion manufacturability of copper. *Proceedings of the 5th Metal Additive Manufacturing Conference*, *Austria* (173-182).

Kline, S.J. & McClintock, F.A. (1953). Describing uncertainties in single-sample experiments, *Mech. Eng.*, *75*, 3–8.

Blasius, H. (1908). Grenzschichten in Flüssigkeiten mit Kleiner Reibung, *Zeitschrift für angewandte Mathematik und Physik*, *56*, 1-37.

Moody, L.F. (1944). Friction Factors for Pipe Flow, *ASME Trans.*, *66*, 671–684.

Kadivar, M., Tormey, D., McGranaghan, G. (2021). A review on turbulent flow over rough surfaces: fundamentals and theories. *International Journal of Thermofluids*, *10*, 100077. https://doi.org/10.1016/j.ijft.2021.100077.

Colebrook, C.F. (1939). Turbulent Flow in Pipes, with particular reference to the Transition Region between the Smooth and Rough Pipe Laws. *Journal of the Institution of Civil Engineers*, *11*, 133–156.

ACKNOWLEDGEMENT

The authors would acknowledge the High Performance Computing infrastructure at the Department of Management and Engineering of the University of Padova. This work was developed with the financial support of Fondazione Cassa di Risparmio di Padova e Rovigo (CariPaRo).