



UNIONE EUROPEA
Fondo Sociale Europeo



DOCTORATE SCHOOL
INDUSTRIAL INNOVATION ENGINEERING
XXXVII CYCLE

**THERMAL MANAGEMENT OF
HIGH POWER DENSITY COMPONENTS
FOR SUSTAINABLE MOBILITY:
DEVELOPMENT OF COMPUTATIONAL METHODS
AND CASE STUDIES**

Scientific Tutor:

Prof. Ing. Diego Angeli

Candidate:

Mattia Grespan

School Dean:

Prof. Ing. Franco Zambonelli

UNIVERSITY OF MODENA AND REGGIO EMILIA
ACADEMIC YEAR 2023/2024

Abstract

This work presents the development of modelling methodologies for the thermal analysis of powertrain and cooling system components for sustainable vehicles. In particular, the analysis is directed at the most relevant components in terms of generation and transfer of heat, namely: (i) electric motors, and the power-electronics components associated with their control; (ii) compact heat exchangers. This PhD thesis was co-funded within the framework of the National Operational Programme Research and Innovation 2014-2020 (CCI 2014IT16M2OP005), ESF REACT-EU resources, Action IV.4 "Doctorates and research contracts on innovation topics" and Action IV.5 "Doctorates on Green topics".

The study of overall systems and complex components is addressed by developing a generalised computational tool, based on the lumped parameters approach. This toolbox is based on three main pillars: the representation of the physical domain through the graph theory, an advanced management system for units of measurement, and a generalised formulation for source and sink terms. These general building blocks are used to implement a solver for one-dimensional flow within networks of pipes, and a solver for networks of thermal resistances. The latter is used to develop thermal models for an integrated motor drive composed by a synchronous motor and a multilevel power converter, where thermal resistances are derived from geometric data, empirical correlations, and a dedicated Computational Fluid Dynamics model for the cooling jacket. The effects of the thermal properties of the materials comprised in the assembly on thermal performance are assessed by means of a comprehensive parameter sensitivity analysis. This study was carried out as part of the European project SCAPE, which aims to improve the performance and standardisation of power electronics components for next generation electric vehicles.

An in-depth analysis is also carried out on compact heat exchangers, with particular reference to plate-fin heat exchangers. Heat exchangers, are essential components of any cooling system; here, they are analysed by developing an original approach solely based on

physical principles and established models. This methodology combines detailed numerical models for the thermal-hydraulic analysis of extended surfaces and manifolds, with global models aimed at accurately evaluating overall head losses and heat transfer rates in plate-fin heat exchangers. The approach is validated by comparing numerical and experimental results on several configurations, producing maximum relative deviations of $\pm 20\%$. The thermal analysis of heat exchangers is further expanded by a parametric analysis on offset-strip fins, performed with the aim of deriving generalised and continuous response surfaces for head losses and heat transfer rates, suitable for the design and optimisation of plate-fin heat exchangers. These response surfaces extend over a wide Reynolds and Prandtl number range ($50 \leq Re \leq 12000$, $0.71 \leq Pr \leq 190$). In addition, a novel empirical model for the Prandtl number scaling exponent is proposed. The application of the response surfaces to the analysis of complete heat exchangers issued mean and maximum deviations of $\pm 7.8\%$ and $\pm 20\%$, respectively.

The modelling methodology developed for offset-strip fins is subsequently applied to wavy fins. Preliminary results on heat transfer and head losses associated with eighty geometric configurations and a single flow rate are employed to derive response surfaces, where the dependence on the Reynolds number is modelled using scaling laws available in the literature. The validity of this approach is assessed by comparing numerical results with experimental data on complete heat exchangers. In particular, the use of the derived response surfaces issued average relative deviations between numerical and experimental results on heat transfer and head losses of $\pm 12.1\%$ and $\pm 10.3\%$, respectively.

The methodologies developed in this work represent practical and effective approaches suitable for the thermal analysis of complex parts and components of modern powertrains for cleaner, more sustainable mobility. Low-order approaches were combined with three-dimensional numerical analyses to reduce the required computational effort, to improve practicality and flexibility of the developed methodologies, which could prove advantageous in industrial contexts. In addition, the high degree of generality of the presented approaches makes them suitable for the analysis of powertrain and cooling components with any architecture.

Contents

1	Introduction	1
1.1	Traction motors	4
1.1.1	Architectures and working principles	4
1.1.2	Heat generation mechanisms	8
1.1.3	Thermal management	9
1.2	Inverters and power electronics components	15
1.3	Heat exchangers and extended surfaces	19
1.4	Aim and scope	20
1.5	Structure and organisation	22
2	Reduced order modelling of physical systems	23
2.1	Introduction	23
2.1.1	Background and aims	23
2.1.2	Literature survey	25
2.1.3	Methodology	27
2.2	Domain discretisation	29
2.3	Dimensioned values and units of measurement	32
2.3.1	Operations between dimensioned values	33
2.4	Generalised modelling of source and sink terms	35
2.5	Development of a one-dimensional flow solver	36
2.5.1	Governing equations	36
2.5.2	Discretisation schemes	36
2.5.3	Modelling of components of networks of pipes	46
2.5.4	Solution algorithm	54
2.6	Implementation of a thermal network solver	62

2.6.1	Governing equations	62
2.6.2	Modelling schemes for thermal resistances	64
2.6.3	Boundary conditions	64
2.6.4	Solution algorithm	65
2.7	Concluding remarks	67
3	Thermal analysis of an integrated motor drive	69
3.1	Introduction	69
3.1.1	Background and aims	69
3.1.2	Literature survey	71
3.1.3	Modelling approach	74
3.2	Case study	76
3.3	Numerical methods	79
3.3.1	Two-dimensional thermal model	79
3.3.2	Three-dimensional thermal model of a single slot	86
3.3.3	Complete 3D thermal model	89
3.3.4	Boundary conditions and implementation	90
3.3.5	Validation of advective resistances and thermal diodes	91
3.3.6	Numerical model of the cooling jacket	92
3.3.7	Parameter sensitivity analysis	95
3.4	Results and discussion	105
3.4.1	Validation of the 2D motor thermal model	105
3.4.2	Cooling jacket	105
3.4.3	System level results	107
3.5	Concluding remarks	108
4	Physics-based modelling of plate-fin heat exchangers	111
4.1	Introduction	111
4.1.1	Motivations and aims	111
4.1.2	Literature survey	113
4.1.3	Modelling approach	115
4.2	Case study	117
4.3	Numerical methods	120
4.3.1	Single fin model	120
4.3.2	Numerical model for entrance effects	123
4.3.3	Head losses in inlet and outlet manifolds	125
4.3.4	Global models	129
4.4	Experimental apparatus and procedures	131
4.5	Results and discussion	133
4.5.1	Single fin model	133

4.5.2	Entry region	141
4.5.3	Head losses in inlet and outlet manifolds	144
4.5.4	Global models	146
4.6	Concluding remarks	148
5	Parametric analysis of offset-strip fins	153
5.1	Introduction	153
5.1.1	Engineering and technical background	153
5.1.2	Literature survey	155
5.1.3	Modelling approach	158
5.2	Problem statement	158
5.3	Numerical methods	160
5.3.1	Governing equations	160
5.3.2	Boundary conditions	162
5.3.3	Spatial discretisation	162
5.3.4	Mesh sensitivity analysis	162
5.4	Design of Experiment	165
5.5	Numerical results	167
5.6	Derivation of response surfaces	172
5.6.1	Equivalent Darcy friction factor	173
5.6.2	Colburn factor	178
5.6.3	Validation	182
5.7	Concluding remarks	186
6	Parametric analysis of wavy fins	189
6.1	Introduction	189
6.1.1	Engineering and technical background	189
6.1.2	Literature survey	191
6.1.3	Modelling approach	192
6.2	Problem statement	193
6.3	Numerical methods	194
6.4	Design of Experiment	195
6.5	Preliminary numerical results	198
6.6	Derivation of preliminary response surfaces	198
6.7	Concluding remarks	203
7	Conclusions	205
A	Numerical methods	211
A.1	Upwind differencing	211

A.2	Midpoint rule	212
A.3	Implicit Euler method	213
B	Offset-strip fins	215
B.1	Implementation of the non-dimensional framework	216
	Bibliography	217

CHAPTER 1

Introduction

In the European Union the transportation sector is responsible for 76% of the Greenhouse Gases (GHG) emissions [1], with passenger cars and the automotive industry accounting for almost half of the entire road transport sector share [2]. By the year 2050, the EU has planned to reach net-zero levels of green house gasses emissions [3], in order to comply with the 1.5 °C target set in the Paris Agreement [4]. In the last twenty years, regulations on pollutants and GHG emissions have guided the development of Internal Combustion Engines (ICEs), whose efficiency and impact on the environment have improved considerably thanks to advancements in key technologies like variable valve timing, downsizing, and efficient turbocharging [5]. Recently, the increasingly stringent limits set by emission standards have forced automotive manufacturers to replace traditional internal combustion engines with hybrid and full electric powertrains, as they allow for a substantial reduction of pollutants and GHG emissions [6] and efficiency improvements, particularly when powered by renewable energy sources [7, 8].

Electric Vehicles (EVs) encompass a wide range of vehicles presenting different powertrain architectures. Battery Electric Vehicles (BEVs) are pure EVs where electric energy is supplied by a large high voltage battery, able to guarantee an average driving range of 378 km as of 2025 [9]. This figure is achieved by electric vehicles equipped with battery packs with capacities ranging from 58 kWh (Peugeot e-408) to 85 kWh (Jaguar I-Pace EV400), while at the top-end of the market vehicles like the Mercedes-Benz EQS 450 and Lucid Air Grand Touring can drive for close to 700 km on a single charge, employing battery packs with a capacity of approximately 110 kWh. The powertrain of an EV is completed by the traction motor/s, which are usually of the synchronous or induction type, and the power converter and control unit. The latter converts the Direct Current (DC) supply coming from the battery to a suitable Alternating Current (AC) power supply to power the motor/s. The architecture of a typical EV is represented in the schematic of Figure 1.1. Hybrid Electric

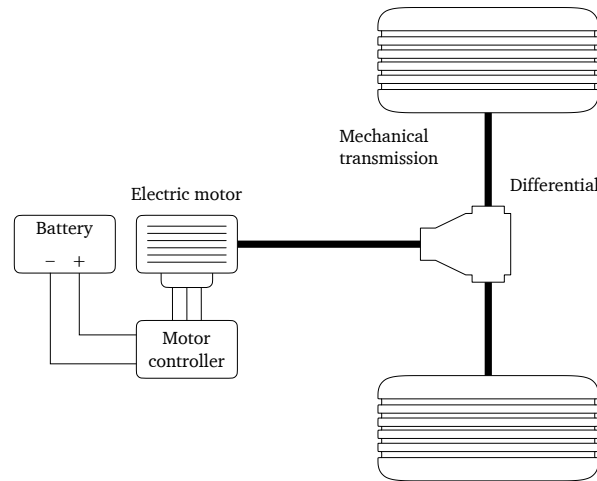


Figure 1.1. Example of a typical architecture of the power train of an electric motor, which includes a high voltage battery, a motor controller, an electric motor and a mechanical transmission. In this schematic the motor controller includes the necessary electronic component needed for power conversion.

Vehicles (HEVs) integrate traditional ICEs with electric motors and small battery packs to improve fuel efficiency and reduce emissions. This is achieved by addressing the main shortcomings of traditional engines, specifically their poor fuel efficiency in transient conditions and their inability of recuperating kinetic energy during braking, which is quite frequent in urban areas.

HEVs are classified on the basis of the architecture of their powertrains, particularly on the relative connections between the ICE, the generator, the motor, and the wheels. In series-hybrid vehicles (Figure 1.2a) the wheels are driven directly by the electric motor, while the ICE is completely decoupled from the axles and used only to charge the battery by driving a generator [10]. With this arrangement the internal combustion engine can theoretically always run at its most efficient point as it is completely disconnected from the wheels. However, both the ICE and electric motor must be sized to meet the complete power demand of the vehicle. This shortcoming is addressed by parallel-hybrid configurations (Figure 1.2b), where the electric motor and the ICE are connected by a mechanical transmission, allowing for both machines to provide mechanical power to the wheels. In this case, the generator is not needed and both machines can be designed with lower power capacity [11], thus reducing volume and cost. However, the fuel efficiency of the ICE is typically lower compared to a series arrangement, as the engine is mechanically connected to the wheels. The advantageous features of series and parallel arrangements are combined in the series-parallel architecture (Figure 1.2c), where a second motor/generator unit is used to charge the battery when a surplus of mechanical power is available from the ICE [12]. Figure 1.2d shows an example of the complex hybrid arrangement, which is a variant of the series parallel architecture, where the electric coupling between ICE and electric motor is achieved by means of the battery.

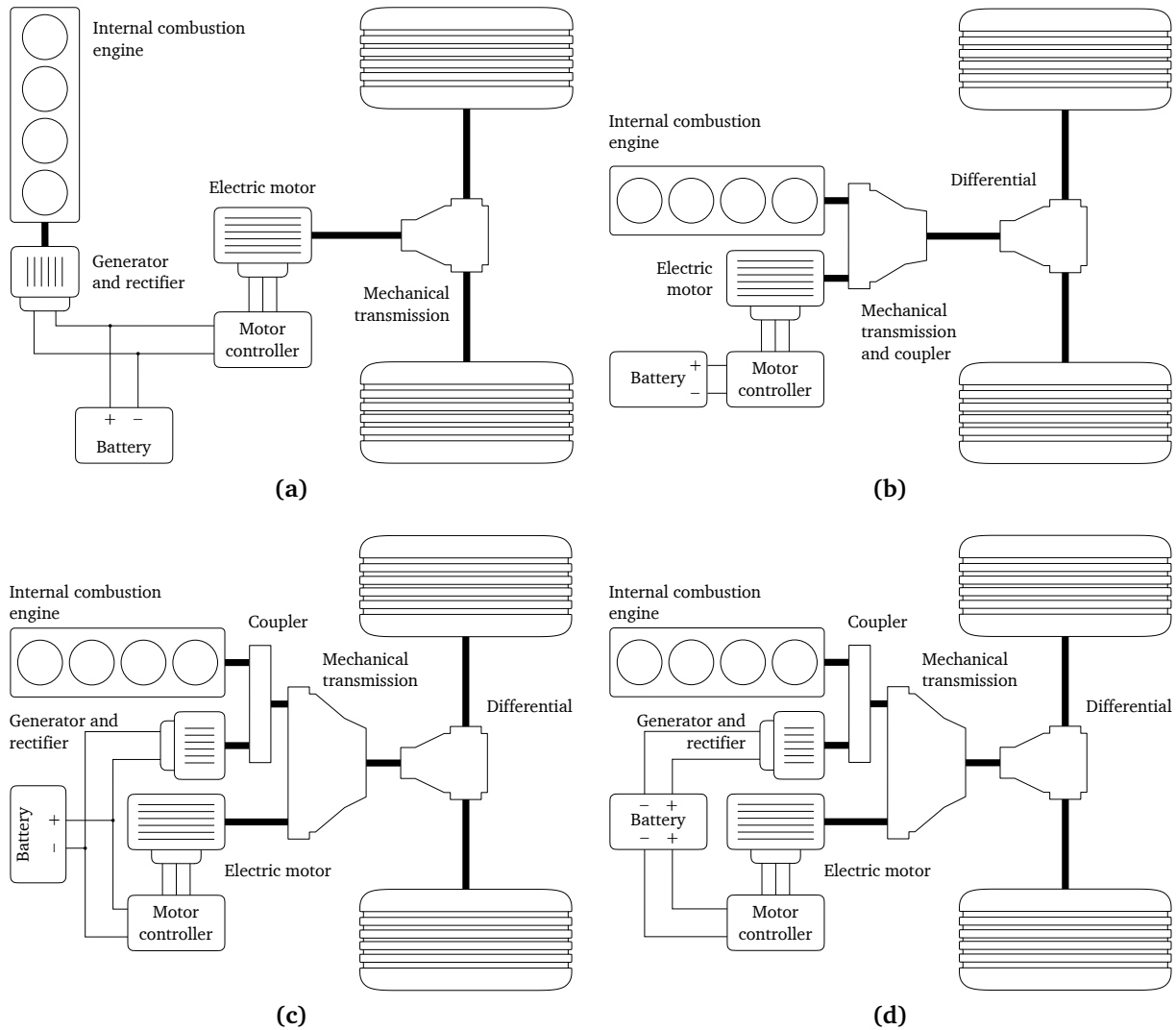


Figure 1.2. Schematics of the most relevant powertrain architectures of hybrid vehicles: (a) series hybrid, (b) parallel hybrid, (c) series-parallel hybrid, and (d) complex hybrid.

An additional subset of HEVs is constituted by Plug-In Hybrid Electric Vehicles (PHEVs), which present powertrain configurations analogous to those discussed for conventional hybrid vehicles, with the added possibility of being charged through a standard AC outlet. PHEVs are equipped with battery packs with roughly ten times the capacity of those used for traditional HEVs, allowing for a far higher fully electric range and reduced fuel consumption [13]. A special kind of PHEV is the Extended Range EV (ER-EV). ER-EVs are a variant of series-hybrid vehicles that can be charged through the grid and include a small internal combustion engine used to charge the battery and extend the electric-only range if needed. However, the so-called range extender is not powerful enough to propel the vehicle by itself and can be used only as backup. Of course, the magnitude of the benefits in terms of GHG emissions and fuel consumption depend on how often the range extender is used.

A further class of EVs are Fuel Cell Electric Vehicles (FCEVs). These are electric vehicles where electric energy is supplied by a fuel cell that consumes compressed hydrogen from

a tank and oxygen from the atmosphere, producing water as a waste product. In these vehicles, a small battery pack is used to manage the dynamic power demand, while the fuel cell covers the mean demand, as it is most efficient when working in steady-state conditions. In this case, the advantages in terms of emissions reductions heavily depend on the method used for hydrogen production. Indeed, while hydrogen is often regarded as an emission-free energy source, in reality it is only an energy carrier whose production involves considerable emissions of CO₂ if carried out using fossil fuels as a base [14].

In the following, important powertrain components of EVs relevant to the present work are described, along with a brief overview of the state of the art associated with their cooling techniques. In this overview, battery packs are not discussed in detail as they are not relevant to the specific case studies considered in this thesis.

1.1. Traction motors

1.1.1. Architectures and working principles

A common aspect of any EV, regardless of the architecture, is the presence of an electric power plant, which, as previously anticipated, is constituted by one or more electric motors, an energy storage system and the power and control electronics. While electric motors for stationary industrial applications are generally designed and optimised for steady-state operation at constant speed, traction machines for EVs must be devised to operate efficiently over a wide range of working points including low-speed high torque acceleration, high-speed low torque steady cruising, and anything in between [10, 15]. This requirement poses additional design and control challenges for traction motors compared to stationary electric machines.

In electric vehicles, many types of electric motors are used, including conventional and brushless DC motors, induction and synchronous motors, and switched reluctance motors [16]. While almost the entirety of the light-duty EV industry has switched to Permanent Magnet Synchronous Motors (PMSMs), in an effort to improve efficiency and specific power output [17], DC motors are still prominent in low-power applications like electric bikes and scooters [18], thanks to their advanced degree of technological maturity, their ease of control [10, 19], and the high torque figures produced at low rpm [15]. In DC motors, torque is produced as a result of the interaction between currents flowing in axial conductors placed on the rotor, with the radial magnetic field produced by permanent magnets or coils positioned on the stator, as shown in Figure 1.3. The conductors on the rotor are arranged as a set of identical coils, often referred to as the armature, that are placed in slots on the surface of the rotor. The design of the armature and motor must be tailored to avoid excessive heating of the conductors resulting from frequent and repeated short-cycle operations [20].

In conventional or brushed DC motors, electric power is delivered to the armature by means of carbon brushes sliding on a commutator, which is made up of a set of insulated copper segments mounted on a cylindrical support. The commutator ensures that current

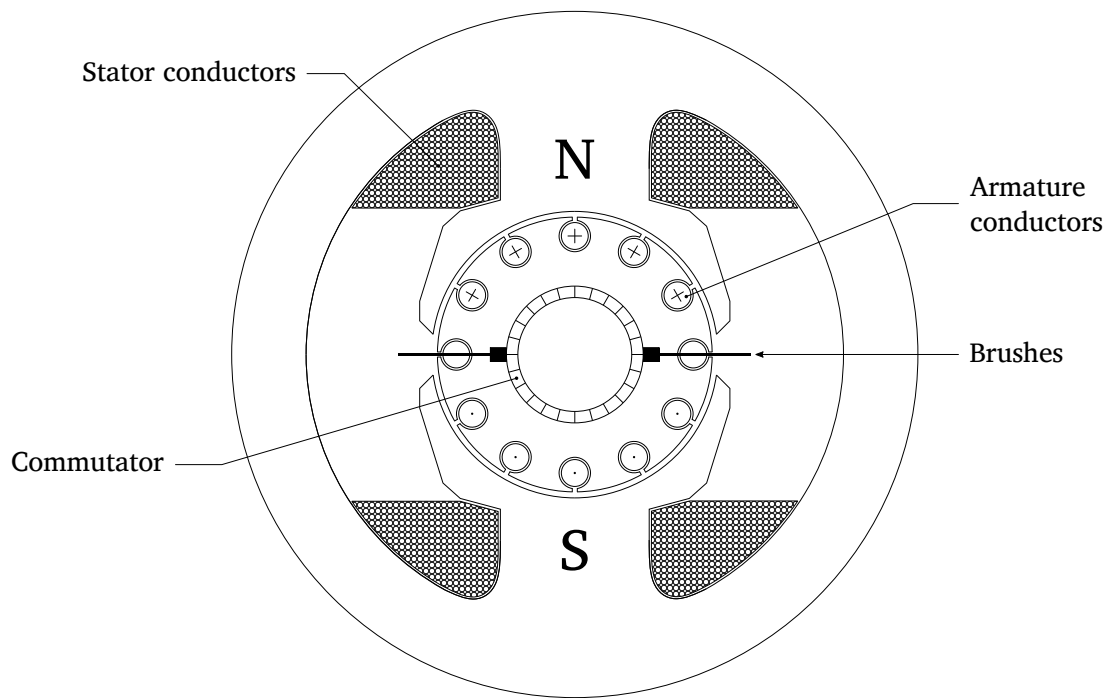


Figure 1.3. Simple schematic including the most relevant components of a conventional DC motor: the stator and rotor conductors, the commutator, and brushes. Adapted from [16].

flowing in the rotor coils has always the correct orientation with respect to the magnetic field generated by the stator. In this type of motor all the mechanical power and hence all electric power must flow through the brushes, which causes considerable losses due to the high velocity electric contact that they form with the commutator. In addition, brushes wear down and must be regularly serviced and replaced [16]. These aspects inherently limit the efficiency of DC motors, which are usually replaced with induction or synchronous motors when efficiency is of primary importance. Nevertheless, DC motors are still a valid candidate for low power and cost-effective EVs, where overall efficiency is sacrificed in favour of simplicity of design and ease of control.

The major disadvantages associated with DC motors are overcome by commutatorless electric motors, among which induction or asynchronous motors represent the most mature technology [10]. The absence of a commutator greatly improves efficiency and reliability, due to the avoidance of the associated electric losses and the removal of parts undergoing major wear, leaving the bearings supporting the rotor as the only components requiring mechanical maintenance. Induction Motors (IMs) have approximately the same size and weight as their DC counterparts, assuming equal power output and the same base speed [16]. However, IMs are less expensive due to their much simpler construction. Nevertheless, these cost considerations do not account for the expenses associated with the variable-frequency inverter which is needed to achieve speed control on induction motors, while in DC motors rotational speed can be changed simply by altering the voltage. Similarly to DC motors, induction machines generate torque due to the flow of axial currents in

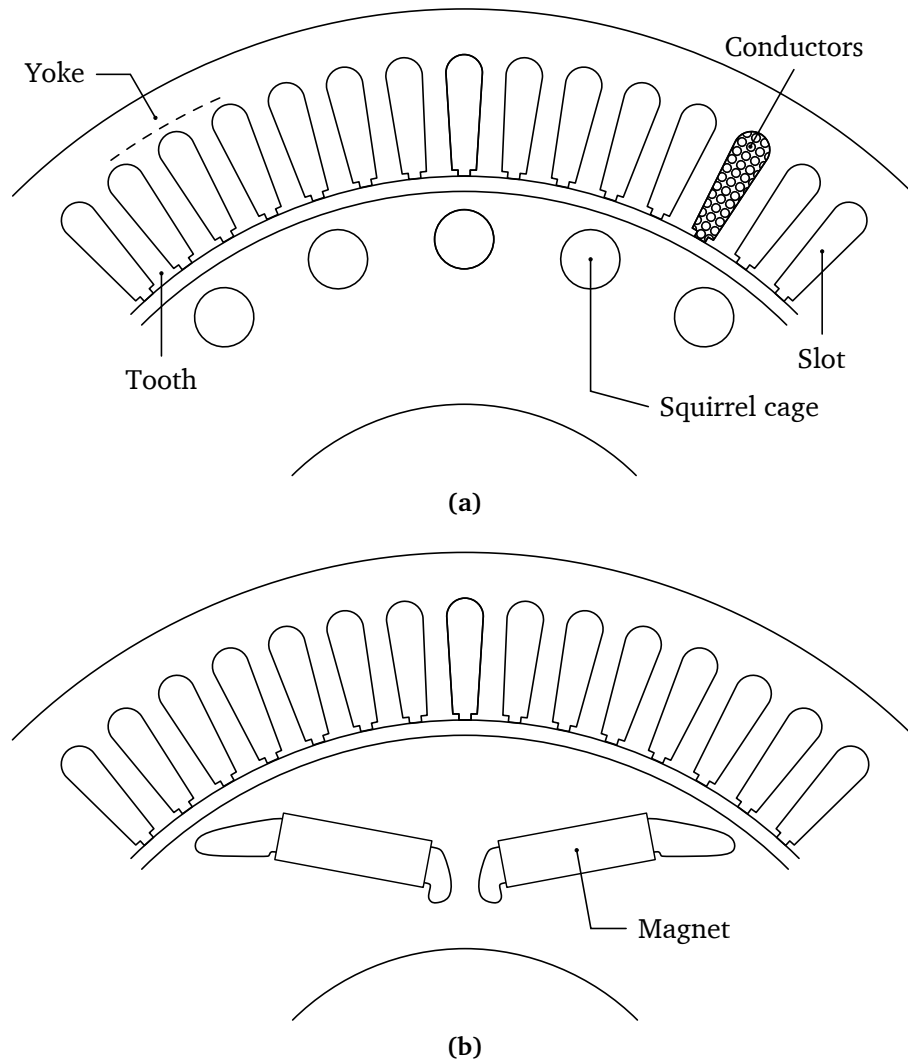


Figure 1.4. Schematics of typical stator and rotor architectures of induction motors (a) and synchronous machines equipped with permanent magnets (b).

the rotor windings interacting with the radial magnetic field generated in the stator. However, this time the torque-generating currents do not flow through a commutator but are induced by the stator windings, which also generate a rotating magnetic field from an alternating current power supply usually presenting three phases. Most IMs feature a squirrel cage rotor, where conductors are constituted by aluminium or copper bars embedded in slots in the laminations and shorted on both sides by end-rings, as depicted in the schematics of Figure 1.5. This rotor architecture is at the core of the simplicity and sturdiness typical of induction motors. While classically wound rotor IMs are also available, they do not see much use, particularly in electric vehicles, due to their poor power density compared to squirrel cage motors. Compared to other motor technologies, IMs may provide lower acceleration figures, however modern control strategies can improve this issue [15]. Induction motors are regularly employed in contemporary electric vehicles such as the Tesla Model 3 and Audi e-Tron SUV [21]. However, the market share of IMs in electric vehicles is not expected to grow significantly, as manufacturers are turning to Permanent Magnets Synchronous Mo-

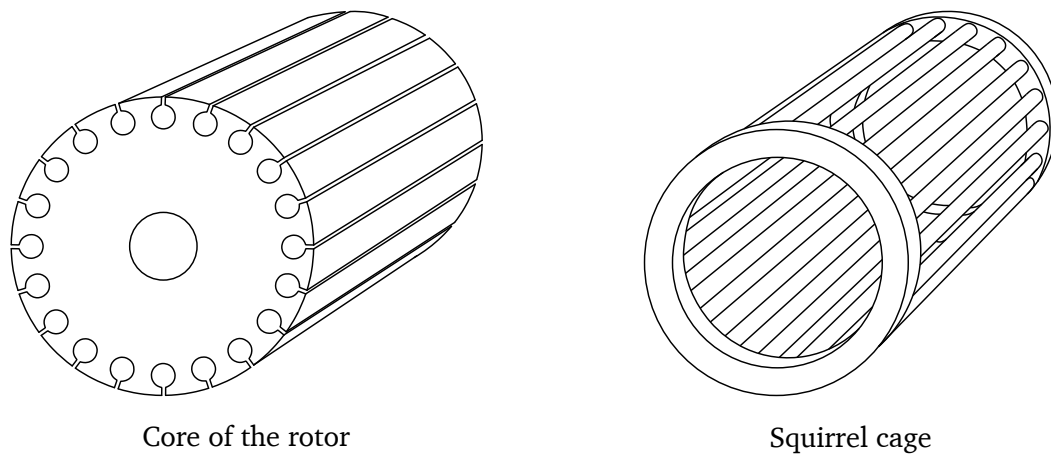


Figure 1.5. Schematics representing the core of the rotor and squirrel cage of an induction motor. Adapted from [16].

tors (PMSMs) in an effort to further improve efficiency and specific power output [22]. Still, induction motors represent a viable and rare-earths-free alternative to PMSMs.

Synchronous motors present distributed stator windings, which generate a rotating magnetic field from a three-phase power supply, exactly as induction motors. However, instead of a cage-like winding, SMs either have DC-powered rotor coils or permanent magnets generating a magnetic field on the rotor, that allows for its synchronization with the rotating magnetic field produced by the stator. Indeed, in synchronous motors the rotor and its magnetic field rotate at exactly the same speed as the rotating magnetic field produced by the stator, which is determined by the number of poles of the motor and the frequency of the supply. Naturally, the magnetic field of the rotor tends to align itself with the one of the stator giving rise to a torque at shaft of the motor. The magnitude of the torque produced is dependent on the shift between the two fields, which increases with the load on the shaft [16].

The use of rare-earths-based permanent magnets for the excitation allows for noticeable efficiency improvements over conventional wound rotor machines and induction motors, as PMs do not use any power, they do not require a commutator, and they produce very small losses. However, the scarcity and high price of rare earth materials considerably drives up the cost of these machines. As a result, Switched Reluctance Motors (SRMs) are gaining traction as a rare-earths-free alternative to PMSMs, thanks to recent improvements in electronic power converters and control strategies. Figure 1.6 shows the architecture of a typical SRM, where both the stator and rotor feature salient poles, and the rotor is free of coils or permanent magnets, allowing for easier and less expensive manufacturing. In addition, the lack of permanent magnets improves the robustness of the machine, which can reliably operate at high temperatures without risking the demagnetisation of the PMs as in synchronous motors [23]. The stator windings are made up by a set of coils wound around the salient poles. In these motors, torque is produced by the force aligning the rotor with the currently

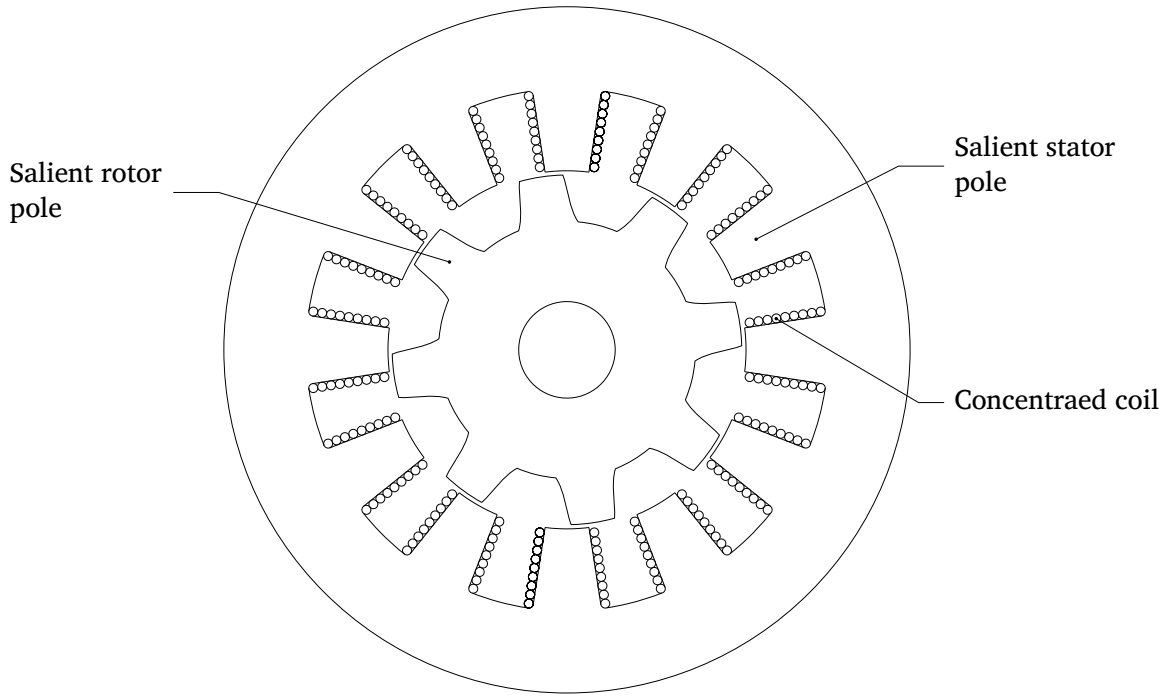


Figure 1.6. Architecture of a typical switched reluctance motor, where the stator windings are concentrated at each salient pole, and the rotor is free from magnets and conductors. Adapted from [16].

exited coil, as a result of the difference of reluctance of the magnetic field lines between the aligned and non-aligned positions [23]. Therefore, power converters must deliver power to the coils sequentially, with a control strategy that limits torque ripple.

1.1.2. Heat generation mechanisms

Although electric motors used for EVs and industrial applications may present very different architectures and working principles, they produce heat according to the same mechanisms. The two most relevant contributions to the overall heat generation are due to Joule effect losses in the copper windings and iron laminations, while secondary contributions are caused by losses in the permanent magnets (if present), ventilation losses in the air gap, and mechanical losses caused by friction in the bearings supporting the shaft. Copper losses, which are particularly relevant in high torque motors, are generated in the windings of both stator and rotor, except for PMSMs and SRMs which do not feature windings on the rotor. The heat produced by Joule effect dissipation in the conductors is affected by proximity and skin effects, which determine uneven current density distributions on the conductors cross-sections leading to increased electric resistance [24].

Generally speaking, the losses produced by a single conductor can be expressed as:

$$\dot{Q}_w = K R_e I^2 = K \gamma \frac{L}{S} I^2 \quad (1.1)$$

where R_e is the electric resistance of the conductor and K is the AC/DC loss ratio, which

expresses the contributions to the losses given by proximity and skin effects. The value of the AC/DC loss ratio is influenced by several factors including the geometry and position of the conductor, phase frequency, and temperature. These effects are often evaluated by means of empirical models [25] or Finite Element analyses [26, 27]. In the active section of the machine, i.e. along the axial development of stator and rotor, large K -values should be expected due to the high magnetic field strength, whereas in the end windings region proximity effects are almost negligible and AC losses are mostly influenced by the skin effect, which is produced by the conductor's own flux linkage [25].

Iron losses are produced in the stator and rotor laminations due to hysteresis effects and eddy currents produced by the time-varying magnetic field [28]. These effects become particularly relevant as the phase frequency increases, i.e. when the motor operates at high rotational velocities, which is typical of traction motors used in vehicular applications.

1.1.3. Thermal management

In any electric motor an increase in mechanical power available at the shaft coincides with an increment of losses in the windings and core laminations, due to growing phase current and/or frequency. The resulting overtemperatures, which can be substantial during high-power operation, increase the thermal stresses on the machine and winding losses, reduce the strength of permanent magnets [29], and can ultimately lead to the failure of the insulation of the windings and the demagnetisation of PMs [30]. The winding and slot insulation is usually composed by wire enamel, a dielectric slot liner, and potting material [31]. Wire enamels are made up of polymers stable up to high temperatures [32, 33], which are susceptible to accelerated breakdown due to repeated overvoltage scenarios [34]. The slot liner is made up insulating paper, while the potting material used to fill air gaps between conductors and the slot, improving the equivalent thermal conductance of the slot, is usually constituted by epoxy, silicone, and urethane based resins [35]. Potting materials are employed in both the slots and the end winding region, where resins with high thermal conductivity are used to lower the thermal resistance between the conductors and the case [36–38]. All these materials are classified on the basis of their insulating class, which determines the maximum temperature that an insulating material can withstand without experiencing accelerated ageing or wear. Table 1.1 reports a selection of insulating classes along with their allowable maximum winding temperature. Respecting the temperature limits set by the insulation class is of utmost importance, as the lifespan of the machine can be more than halved if the winding temperature exceeds the thermal class limit by 10 °C [39]. In addition to lifespan concerns, high temperatures can decrease power output [30] and efficiency as both DC and AC winding losses increase with temperature [40].

The significant impact of overtemperatures on the performance, efficiency and reliability of electric motors requires an accurate assessment of the thermal management of these machines, and a careful design of their cooling system. The architecture of cooling systems

designed for electric motors is strongly influenced by the size of the motor, its specific power output, and the type of application in which it is used. Motors with relatively small specific power output figures of 4 kW/kg or less are often cooled through natural and forced air ventilation [30]. The use of air cooling systems is widespread for both asynchronous and synchronous motors, especially in high speed applications, where large rotational velocities favour convective heat transfer between internal components [35]. In air cooling systems, heat transfer effectiveness is primarily improved by increasing the overall heat transfer area, by employing fins on the external surface of the case [41] and carefully designed ducts and channels for internal air circulation [42, 43]. The heat transfer enhancement provided finned casings is strongly influenced by their number of fins [44] and their geometric parameters [45]. On the other hand, the orientation of the fins has a comparatively lower impact on thermal performance, particularly when considering the temperature distribution in the end windings [46].

The thermal management of the rotor and possible permanent magnets of air-cooled motors can be improved by tailoring the design of the laminations and shaft. The air flow rate between stator and rotor can be improved by adding longitudinal slots to the core of the rotor, as shown in Figure 1.7a, increasing the heat transfer area and local heat transfer rates [43]. Other dedicated cooling approaches for the rotor involve the use of a hollow shaft possibly equipped with internal fins to cool the rotor from the inside out. An example of this system is shown in Figure 1.7b [42], where the airflow within the hollow shaft is also exploited to cool the power converter components, which are mounted circularly around the intake/exhaust tube.

Airflow over external fins and across internal channels is often provided by radial or axial fans mounted on the shaft of the motor. Radial fans are regularly employed in ventilated motors designed for various uses thanks to their simple design, rugged construction, minimum dust accumulation, and the ability to provide consistent performance regardless of the direction of rotation [47]. In unidirectional applications radial impellers can be substituted by axial fans, which can provide efficiency improvements due to their better dynamic behaviour and the reduction of mechanical energy losses linked to sharp changes of the direction of the flow and recirculation inside the fan shroud [48]. The thermal performance of ventilated motors can be improved by optimising the geometric design of the fan and internal and external cooling channels [49–51]. Nevertheless, any electromagnetic efficiency improvement brought by the reduction of winding and core temperatures must be carefully weighted against the possible increase of mechanical power required by the fan, as

Table 1.1. Insulation classes and their respective temperature limits.

Insulation class	Y	A	E	B	F	H	N	R	S
Temperature limit [°C]	90	105	120	130	155	180	200	220	240

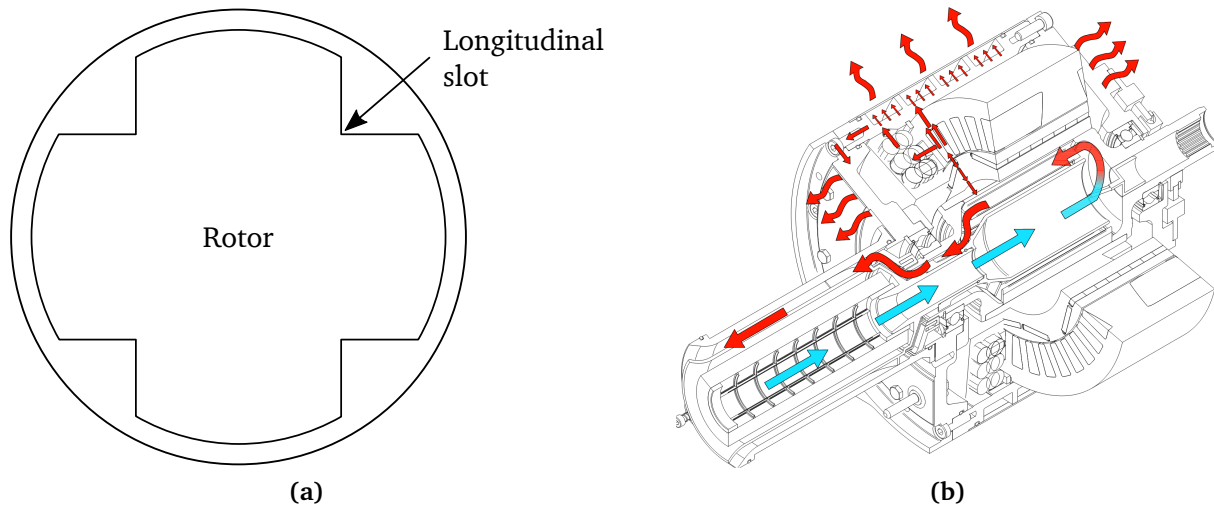


Figure 1.7. Example of a rotor with longitudinal slots to enhance heat transfer in the air gap (a) and a schematic of an electric motor with an air cooled rotor through the use of a hollow shaft (b). Adapted from [43] and [42], respectively.

windage losses associated with the impeller have a relevant effect on the nominal efficiency of ventilated motors [48].

Although air-cooling systems are a viable option for cooling traction motors, especially in the case of cost-minded and low-power applications, modern requirements for high power and compactness have favoured the widespread use of liquid cooling systems in the global EV industry. Depending on the application, liquid-based cooling systems can involve several subsystems using different working fluids to meet cooling requirements. Nevertheless, most of these systems are based on aluminium casings including circumferential, helical, or axial channels used to cool the stator and windings. In the absence of a dedicated cooling sub-system for the rotor, the latter is cooled indirectly by the cooling jacket by heat conduction through the air gap and stator laminations. In general, cooling channels with circumferential or helical development are preferred over axial ducts as they generally lead to comparatively lower head losses at the same heat transfer area [53]. However, axial channels can still be used effectively in applications with modest thermal requirements, as these channels can produce very small pressure drops if a substantial reduction of the overall heat transfer area can be tolerated [54].

The heat transfer performance of the cooling channels heavily depends on the geometric parameters of the ducts and the flow rate [55, 56], similarly to the finned surfaces used in air cooling systems. In general, heat transfer effectiveness improves as the hydraulic diameter of the channels decreases, i.e. as the number of passes and the average fluid velocity increase [57]. In addition, as in all forced-flow problems, at fixed geometry and working fluid the heat transfer coefficient increases with growing flow rate [58]. However, identifying the best strategy to improve heat transfer performance is a complicated task, as a reduction in the hydraulic diameter is always associated with an increase in pressure

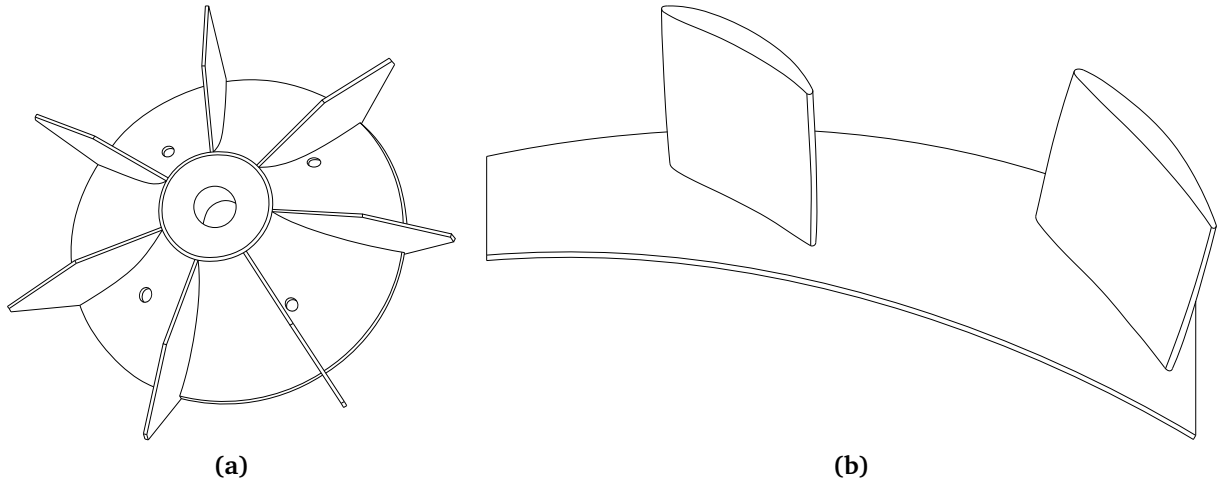


Figure 1.8. Schematics showing examples of a radial fan (a) and the blade detail of an axial fan (b). Adapted from [52] and [48], respectively.

drop and thus a decrease in the flow rate processed by the pump. Furthermore, in some configurations it may be more advantageous to adopt larger hydraulic diameter channels in order to have a greater available flow rate [57]. Therefore, the design of the geometry and distribution of the cooling channels must be carried out for each specific configuration, taking into account the characteristic curve of the pump, and the details of the entire cooling circuit.

The performance of cooling-jacket-based systems can be improved by closely integrating the cooling channels into the design of the electric machine. Figure 1.9a shows an example of these integrated design approaches, where longitudinal slots on the external surface of the stator iron are used in conjunction with an external sleeve to form cooling channels where coolant is in direct contact with the back iron, avoiding the contact and conduction thermal resistances associated with the internal wall of a conventional liquid cooled case [59], allowing for noticeable improvements in terms of thermal performance [61, 62]. Greater improvements in terms of heat transfer performance and electric loading can be achieved by integrating cooling channels near teeth and slots, and within the windings themselves [63]. For instance, copper pipes can be placed at the base of slots to greatly reduce the thermal resistance between the heat source, the winding, and the cooling fluid [60], as shown in the schematic of Figure 1.9b. Similarly, cooling channels can be arranged around the windings sides to achieve analogous effects [64]. More advanced solutions are represented by direct cooling approaches, where windings are cooled directly either by using dedicated cooling channels in direct contact with the copper [65–67] or by employing internally cooled litz wire to form the stator coils [68–70]. While direct cooling arrangements pose significant challenges in terms of implementation and manufacturing [71], they constitute a very promising approach for the reduction of hot spot temperatures in the windings, as more than 80% of winding losses are transferred directly to the cooling fluid [72]. However, the

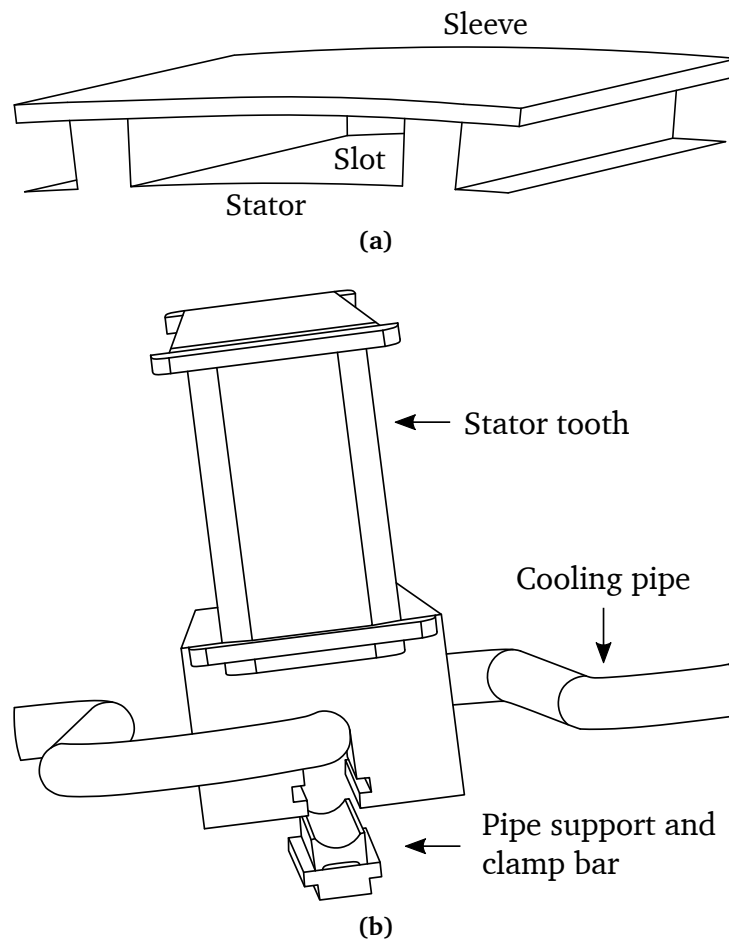


Figure 1.9. Examples of longitudinal slots integrated into the stator laminations to improve cooling (a) and cooling pipe embedded at the base of a tooth (b). Adapted from [59] and [60], respectively.

disadvantages inherently associated with internally cooled wires, such as high coolant pressure drop and higher current density due to the reduced cross-section of the wires, must be carefully evaluated when considering these options [71].

Alternative means of improving the thermal management of the stator conductors, particularly the end windings, are represented by spray cooling and jet impingement methods, which allow for substantial improvement in current density and power output [73]. These approaches rely on the spraying of a cooling fluid, usually a dielectric oil in case of electric motors, onto a target surface that needs to be cooled [74]. Generally, nozzles are mounted onto the end plates of the case, having unrestricted access to spray oil onto the surface of the windings [75, 76]. Regardless of the type of nozzle employed and their arrangement, the thermal performance of spray systems mostly depends on the oil flow rate, whereas the secondary flows introduced by the rotation of the rotor only slightly affect the local temperature of the end windings. The temperature discrepancies between different sections of the end windings due to the movement of the rotor tend to be less pronounced at high oil flow rates [77].

Dielectric oils are also often employed in dedicated cooling solutions for the rotor, allowing for a noticeable decrease of the temperature of the magnets [78] or rotor conductors if present. Similarly to the previously mentioned air-based cooling system for the rotor, oil is usually introduced into the machine through a hollow shaft via a stationary tube, flowing towards the outlet within the annulus formed by the tube and the hollow shaft [79]. The heat transfer effectiveness of these systems is strongly enhanced when the rotational speed of the rotor is high [80], suggesting that these systems may be better suited for high velocity motors while they might have a comparatively lower impact in high-torque slow-speed motors.

Besides traditional air or liquid cooling systems, alternative heat transfer enhancement approaches based on heat pipes and Phase Change Materials (PCMs) can be employed in electric motors. Heat pipes are passive devices characterised by high heat transfer rates, resulting from the phase change of the fluid contained within them [81]. At the evaporator the liquid absorbs heat turning to vapour, subsequently it flows to the condenser where thermal energy is released as a result of the condensation of the vapour. Finally, the condensed liquid is returned to the evaporator thorough a wick by capillary action [82]. In electric motors, the evaporator is usually embedded inside the machine close to the most relevant heat sources such as the stator windings [83], while the condenser is usually placed outside the motor and connected to a liquid or air cooled heat sink [84]. This allows for a substantial reduction of the equivalent thermal resistance between the heat sources and the cooling system, significantly improving the performance of the motor [85]. Similar but less advanced systems are based on the use of heat guides, which are nothing more than plates made up of a highly thermally conductive material such as copper. Similarly to heat pipes, heat guides are used to improve heat transfer in key regions such as the winding assembly, where the equivalent thermal conductivity is very low [86].

Phase change materials can be employed to improve the performance of electric machines during transient load cycles, thanks to their energy storage capabilities [87]. Specifically, the high latent and specific heat of PCMs allows for a more uniform temperature distribution over time, with lesser peaks and troughs, reducing thermal stresses and allowing for longer operation at high power levels [88]. Phase change materials are usually integrated into the case of the machine in purposely designed pockets [89]. The effectiveness of PCMs can be improved by placing them close to the heat sources, i.e. in the stator windings by using hollow conductors [90].

The wide variety of cooling systems and techniques involved with electric motors developed to date highlights the importance of thermal management for the design and operation of these machines. Furthermore, the stark differences between the aforementioned cooling approaches calls for flexible analysis, design, and verification tools capable of modelling largely different physical phenomena with sufficient accuracy.

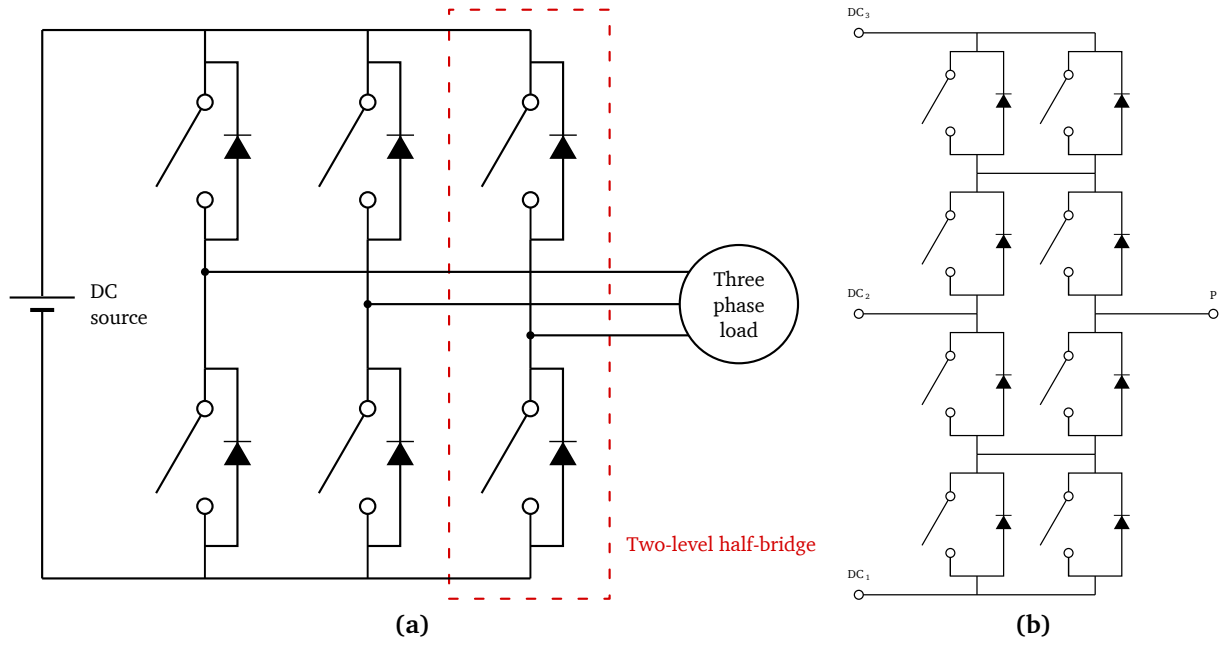


Figure 1.10. Schematic of a conventional three-phase voltage source inverter (a), and a single phase of a multilevel power converter (b). In these schematics the power semiconductors are represented as ideal diodes and switches. Adapted from [94].

1.2. Inverters and power electronics components

Inverters are among the most important components in electric vehicles [91], as they allow for the conversion of the DC power supply coming from the battery to an AC source suitable for the operation of electric motors [92]. Ideally, the voltage of the resulting AC supply would present a perfect sinusoidal waveform; however, the use of switches determines a waveform that only approximates a sine wave by succession of rectangular segments [16]. Nevertheless, the inductive nature of the load given by electric motors allows for the production of a smooth current waveform [93].

Most of the electric vehicles currently in production employ three-phase Voltage Source Inverters (VSIs) using Insulated Gate Bipolar Transistor (IGBTs), due to their high efficiency and low cost [95]. Conventional three-phase VSIs are constituted by three two-level half-bridges connected in parallel to the DC supply [94], as shown in the schematic of Figure 1.10a. Each half-bridge is composed by two controlled switches with their accompanying antiparallel diodes, which are used as protection against possible overvoltages caused by inductive loads [16]. All the power semiconductors used in conventional VSIs are affected by both conduction and switching losses. Conduction losses are associated with on-states, where the semiconductor behaves as a constant resistance usually referred to as $R_{DS(on)}$ [93]. These losses are evaluated using Joule's first law:

$$\dot{Q}_{cd} = R_{DS(on)} I^2. \quad (1.2)$$

On the other hand, switching losses are related to the transitions between on and off states, where both voltage and current are high. These losses are proportional to the switching delays associated with turn-on and turn-off operations and to the switching frequency, which is usually quite high to improve the dynamic performance and reduce the size of the semiconductors [96]. The computation of switching losses is a complicated task, as the transition between states is a highly dynamic process where the state of the semiconductor changes continuously. However, a basic estimate can be obtained from the following relation [96]:

$$\dot{Q}_{sw} = \frac{1}{2} V_{off} I_{off} (t_{off} + t_{on}) f_{sw} + \frac{1}{2} C_{oss} V_{off}^2 f_{sw}, \quad (1.3)$$

where t_{on} and t_{off} are the time delays associated with the turn on and turn off transitions, respectively. The switching frequency is denoted by f_{sw} , while C_{oss} is the output capacitance of the semiconductor. The first term of Equation (1.3) is linked to the losses caused by the time delays associated with the switching transitions, while the second accounts for the dissipation of the energy stored in the output capacitance of the semiconductor that takes place during turn-off transitions.

Switching losses can be mitigated by designing power switches with faster turn-on and turn-off transitions, and, therefore, greater switching frequencies. This also allows for the reduction of the size and losses associated with filters present in most power conversion devices [97]. However, semiconductors with optimised switching characteristics often perform poorly in continuous conduction scenarios, leading to large conduction losses. For these reasons, it is necessary to identify a design trade-off that allows for the overall reduction of conduction and switching losses. This often involves optimising each individual semiconductor for the specific application in which it will be used, resulting in a large engineering and design costs.

The challenges inherently linked with the design of conventional VSIs using two-level half-bridges have prompted the development of alternative architectures for power inverters. Among these innovative designs, Switching-Cell Arrays (SCAs) represent a promising approach to improve the standardisation of power semiconductors and reduced design costs. Multilevel power converters based on SCAs revolve around single switching cells highly optimised for specific voltage levels, that allow for a better distribution of switching and conduction losses across many power switches, leading to efficiency improvements. In addition, the improved standardisation coming from the use of a generalised basic building block, the switching cell, allows for considerable savings in terms of cost. However, these advantages come at the cost of a greater number of switches and a more complex control strategy compared to conventional VSIs [94], as shown in the schematic of Figure 1.10b.

In all power converters or any device using power electronic components, regardless of their architecture, the thermal management of each individual power device plays a critical role in ensuring the efficiency and reliability of these devices. Although power semiconduc-

tors are efficient devices that produce small losses in relation to the electric power processed, their size, which is continuously decreased through the years in order to improve power density, determines very large local heat fluxes as 500 W/cm^2 [98]. Moreover, thermal management issues account for more than 50% of the failures of integrated circuits [99], as the mean lifetime of an individual transistor sharply decreases with growing temperature values [100] with possible sudden failures if the maximum junction temperature is exceeded [92].

Most of the power converters installed into contemporary production vehicles are based on conventional VSIs presenting liquid cooled IGBTs. The inverter in the 2016 Chevrolet Volt [101, 102] features dual three-phase voltage source inverters, and a third low-power unit to drive the electric oil pump. The switching devices are power modules manufactured by Delphi Technologies including a silicon IGBT and freewheeling diode in an electrically insulated package that can be cooled on both sides. These modules are cooled by two identical liquid cooled copper heat sinks, which are manufactured by injection moulding. In these heat sinks heat transfer is enhanced by means of rectangular channels with alternating dimples on the sidewalls. The contact thermal resistance between the chips and the coolers is reduced by the application of a Thermal Interface Material (TIM). The heat sinks cooling the IGBTs are integrated into a system that runs throughout the case of the inverter cooling the AC busbars and the capacitor assembly. The inverter itself is mounted on top of the transmission case, so that the AC bus bars can be directly connected to the electric motor, thus avoiding the use of phase cables, in an effort to improve efficiency and reliability, and to reduce the electromagnetic interference coming from the inverter [95].

Another inverter design by General Motors is the one from Cadillac CT6 [103, 104]. Also in this case the power converter is made up of two main inverters for the two electric motors and one for the engine oil pump. The two motors are fully integrated into the drive train as part of the electric variable transmission, which allows for full CVT operation by using one of the two EMs as a generator to convert a part of the energy from the engine into electric energy, thus driving the other motor. The two EMs and the ICE are connected to the wheels by planetary gear sets, to ensure smooth operation. The inverters are made up by power modules manufactured by Hitachi including a two-level half bridge. In each power module the silicon IGBTs, diodes, and the necessary spacers are directly soldered together and to a copper lead frame. The latter is encapsulated by resin and surrounded by an insulating sheet. These components are included in an aluminium case with pin fins on both the front and back surfaces. The power modules assembled in this way can be directly inserted into cooling passages without the need for any TIM, allowing for substantial cooling improvements.

The Power Control Unit (PCU) in the fourth generation Toyota Prius [107, 108] is installed on top of the transaxle transmission as in the Chevrolet Volt. This allows for a noticeable weight reduction thanks to the length reduction of high voltage cables, and leads to

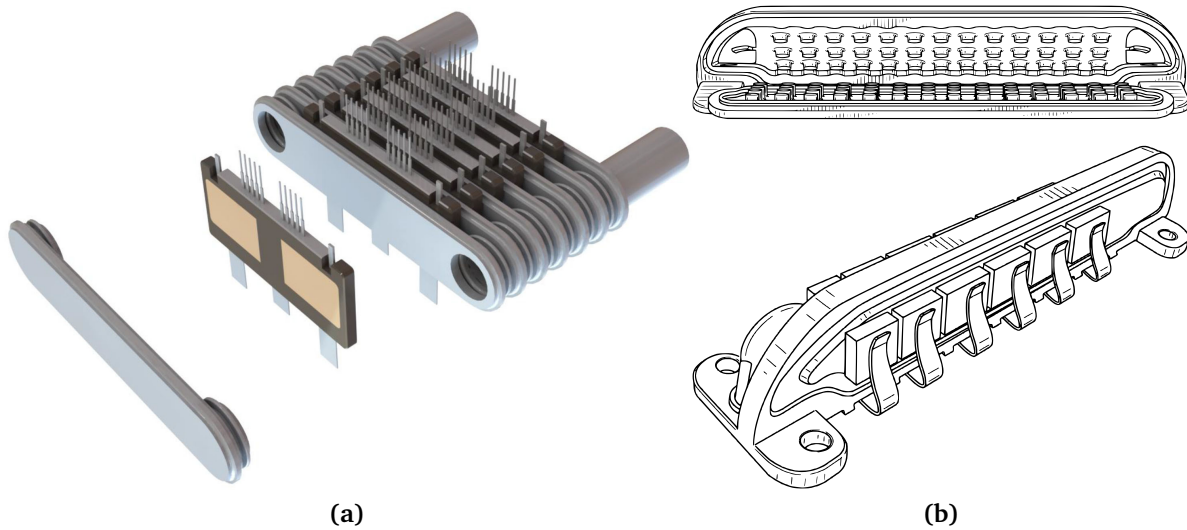


Figure 1.11. Examples of dedicated cooling solutions for the power semiconductors employed in the inverters of the Toyota Prius [105] (a) and Tesla Model S [106] (b).

a more robust assembly that is less likely to lose insulation in case of a front collision. The designers focused their attention on reducing the vibrations transferred from the engine and gearbox by using appropriate vibration resistant electronic components and suitable assembly techniques. The PCU includes the inverter for the motor-generator, the boost converter, and the DC/DC converter. The silicon IGBTs and freewheeling diodes constituting the inverter are assembled in power modules each making up one leg of the inverter. These power modules are stacked together to form full units. Water cooled heat sinks are placed in between the modules, along with a layer of thermal grease, as shown in Figure 1.11a. The details on the cooling channels geometry are not given, however the previous generation of heat sinks featured simple straight channels [109]. Further improvements regarding weight reduction and compactness were achieved by integrating the smoothing capacitor with the bus bar modules.

In the 2012 Nissan Leaf, the power switches of the inverter are directly attached to the busbars, by means of an intermediate copper-molybdenum layer used to alleviate the stresses originating from the different thermal expansion coefficients of the chips and the bus bars [110]. The bus bars themselves are fixed to the heat sink by means of an insulating pad, that features a high thermal conductivity. The heat sink is made out of aluminium and features nine rectangular cooling channels arranged in a snake-like shape featuring 180 degree bends. The motor casing is made out of low pressure cast aluminium and features three plane circumferential cooling channels that are connected by axial ducts [111]. A similar arrangement is found in the power converter of the 2014 Honda Accord Hybrid, where the power semiconductors are soldered directly to the liquid cooled heat sink by means of a ceramic substrate. The cold plate presents a rectangular fin array. Soldering power switches directly to their cooler ensures a superior thermal performance compared to

the use of insulating thermal pads as in the Nissan Leaf [110]. However, the larger thermal capacity of the Cu-Mo plates used in the inverter of the Nissan Leaf allows for better heat redistribution during transient load cycles.

Unlike most of the power converters mentioned so far, the inverter of the Tesla Model S does not feature power cards or modules, but instead is made up of individual TO-247 IGBT packages [95]. Each switch is made up by six transistors in parallel for a total amount of 36 packages. Figure 1.11b shows a module of the inverter of the Tesla Model S, which comprises a group of twelve IGBTs that are fixed to a two piece cast aluminium heat sink by metal clips. The heat sink is composed of two halves, that are brazed or welded together [106].

1.3. Heat exchangers and extended surfaces

Heat exchangers are important components in any cooling system, especially those found in electric vehicles. Indeed, heat exchangers designed for electric vehicles must not only provide an adequate heat transfer effectiveness to ensure the correct and safe operation of the main powertrain components, such as motors and power converters [112], but must also be optimised to minimise the energy consumption of the cooling system itself [113]. These strict requirements on energy consumption stem from the limited energy capacity of the main battery pack. Modern energy storage systems for electric vehicles are often based on Li-ion cells, as they offer the highest figures of volumetric energy density, between 0.3–0.6 MJ/l, among the available chemistry types [5]. Still, they fall short when compared to traditional fuels like petrol and diesel, that feature energy density values of 34.2 MJ/l and 38.6 MJ/l, respectively. This difference translates into a substantial range discrepancy between EVs and traditional ICE-based vehicles [7, 114], which is further magnified if the sub-systems required to operate the vehicle, such as the cooling system, are inefficient. Hence, traditional sizing and design procedures based on worst-case assumptions are no longer appropriate as they may lead to configurations with unfeasible packaging and excessive energy consumption [115].

In electric vehicles, Plate-Fin Heat Exchangers (PFHEs) are often employed as a more effective alternative to conventional plate heat exchangers [116], which are part of the heat pump systems commonly used to recuperate heat from batteries and other components to reduce the energy demand linked to the thermal management of the cabin [117]. PFHEs are constructed by successively stacking aluminium sheets and sidebars to form rectangular channels where finned sheets are introduced to improve heat transfer effectiveness. Figure 1.12 shows examples of PFHE configurations regularly employed in vehicular applications. These coolers are often categorised as compact heat exchangers due their high heat transfer area over volume ratio, which allows them to perform effectively even when small temperature differences, typical of heat recovery processes, are involved [118, 119]. The inherent flexibility coming with the architecture of PFHEs allows for a wide variety of possible geometric configurations for the finned surfaces. Among these, Offset-Strip Fins (OSFs)

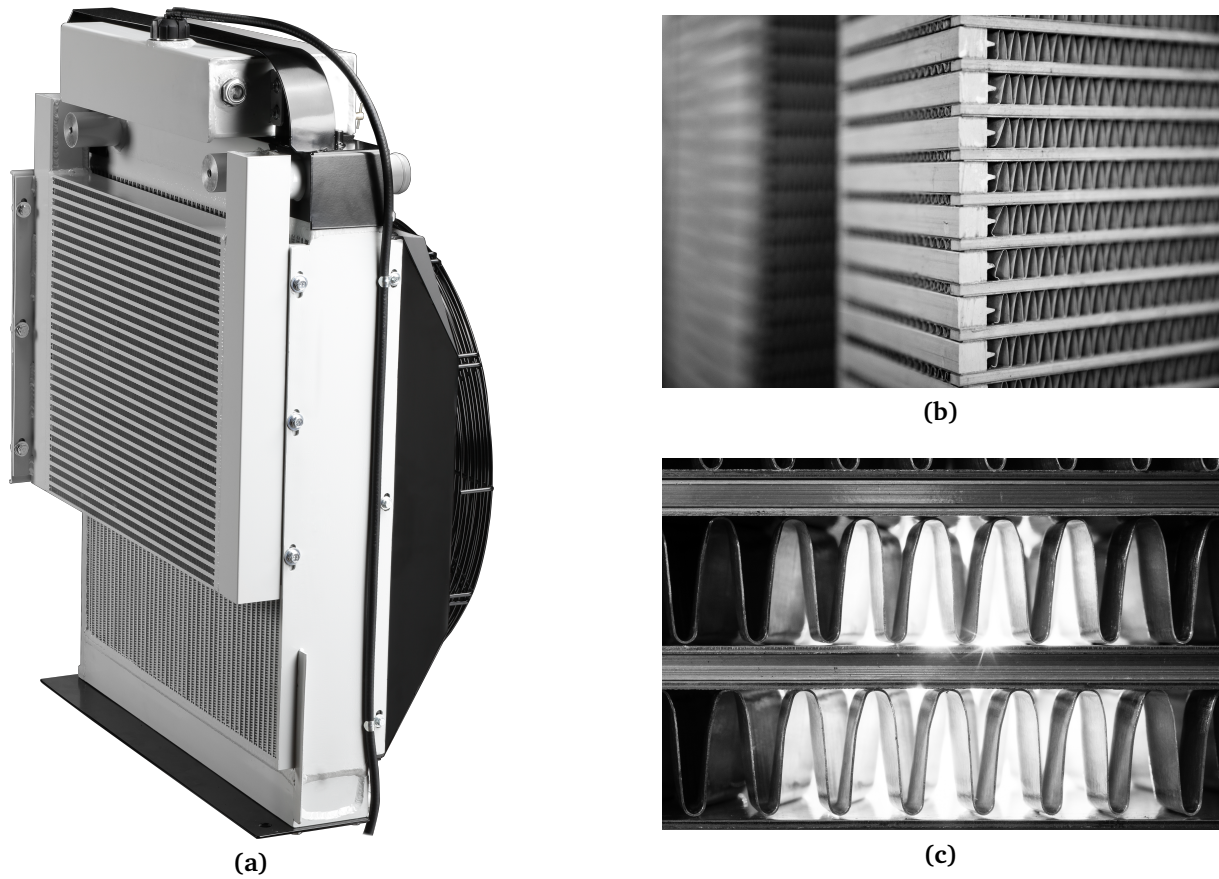


Figure 1.12. Pictures showing a cooling unit composed by multiple compact heat exchangers (a), a PFHE core equipped with offset-strip fins and wavy fins (b), and a close-up of a channel presenting wavy fins (c). Pictures provided by VEMA S.r.l.¹.

are one of the most prominent geometries, thanks to their large surface area and heat transfer coefficient. In addition, their great mechanical stability allows for a reduction of the size and weight of the support structures of the heat exchangers [120]. Wavy Fins (WFs) are also used extensively in plate-fin heat exchangers, thanks to their superior resistance to clogging [121] in harsh environments, compared to offset-strip fins.

The performance of OSFs and WFs, as of that any extended surface, is highly dependent on geometric and flow parameters, which can be varied to achieve a virtually infinite number of configurations. Therefore, designing efficient heat exchangers requires the careful consideration of the direct and indirect influence of the parameters.

1.4. Aim and scope

The complexity of the cooling systems used in electric vehicles, and the marked importance of their effectiveness and energy efficiency on the overall performance of EVs, underline the importance of considering the cooling system at the very early stages of the design of electric vehicles. To this end, accurate and computationally efficient tools are needed to rapidly

¹VEMA Industries S.r.l., Via Guido Rossa 8, 43058 Sorbolo-Mezzani, Italy.

evaluate the feasibility of a given architecture and assess the effects of design changes. In general, these types of computational tools are based on lumped parameter frameworks, thanks to their flexibility, low computational effort, and ease of use. At the same time, the detailed analysis of individual components may require more advanced and computationally intensive approaches like Computational Fluid Dynamics (CFD) and Finite Element Analysis (FEA). However, the computational effort often associated with these approaches makes performing a large number of runs impractical or even unfeasible. It follows that the tools best suited for the design and verification of cooling systems for EVs may be a combination of reduced-order methods and higher order approaches, where lumped parameter models are used for the bulk of the analysis with limited use of higher order methods to improve spatial accuracy where necessary.

This thesis presents the development of multiscale modelling approaches for the analysis of relevant components of EVs compounding reduced order approaches for the evaluation of their overall performance with CFD models to improve the resolution and accuracy of results where needed.

Initially, a generalised methodology for the development and analysis of reduced-order models is developed and implemented in Python language. This toolbox is based on three main pillars: the representation of the physical domain through the graph theory, an advanced management system for units of measurement, and a generalised formulation for source and sink terms. These general building blocks are used to implement a solver for one-dimensional flow within networks of pipes, and a solver for networks of thermal resistances. The latter is then used to develop thermal models for an integrated motor drive composed by a synchronous motor and a multilevel power converter, where thermal resistances are derived from geometric data, empirical correlations, and a dedicated Computational Fluid Dynamics model for the cooling jacket. These thermal models are used to evaluate the thermal performance of the proposed cooling solution and to evaluate the effects of different choices of constitutive material on thermal management, through a comprehensive sensitivity analysis. The thermal analysis of the integrated motor drive was carried out as part of the European project SCAPE, which aims to improve the performance and standardisation of power electronics components for next generation electric vehicles.

The rest of this thesis is dedicated to the analysis of the thermal-hydraulic performance of compact heat exchangers, given the relevance of their thermal performance and energy efficiency on the overall efficiency and range of EVs. At first, a multiscale methodology for the analysis of PFHEs is presented. This approach compounds detailed CFD models for the extended surfaces and manifolds with reduced-order models to evaluate the overall heat transfer rates and head losses of compact heat exchangers. This methodology is presented through the analysis of a case study including several configurations of PFHEs with different extended surfaces and working fluids. Subsequently, the generality and capabilities of this approach are expanded by developing generalised response surfaces for

the thermal-hydraulic performance of offset-strip fins, which allow for the extension of the possible PFHE configurations that can be studied using the aforementioned multiscale approach. The response surfaces for heat transfer performance and head losses are obtained using a Design Of Experiment (DOE) procedure, which is performed by numerical methods. Numerical results are used as a base for a curve fitting procedure to determine the coefficients of the polynomial response surfaces. In addition, a novel modelling technique for the Prandtl number scaling coefficient is presented. Finally, the response surfaces are validated by comparing numerical and experimental results on the heat transfer rates and head losses of complete heat exchangers.

The methodology developed for the analysis of offset-strip fins is also applied to wavy fins, to study the dependence of their thermal-hydraulic performance on dimensional and flow parameters. Preliminary numerical results associated with eighty geometric configurations and single flow rate are used to derive response surfaces, using analogous techniques to those employed for OSFs, where the dependence on the Reynolds number is modelled using scaling laws available in the literature. This approach is validated by comparing numerical results obtained from the response surfaces with numerical and experimental outcomes on single fins and complete heat exchangers, respectively.

1.5. Structure and organisation

This thesis is organised as follows. Chapter 2 covers the development and implementation of the generalised approach for the reduced order modelling of physical systems. The main pillars of the methodology, specifically the spatial discretisation schemes, the management of units of measurement, and the generalised representation of source terms are described in detail. In addition, Chapter 2 presents the development of solvers for networks of pipes and thermal networks.

Chapter 3 covers the thermal analysis of an innovative integrated motor drive for EVs comprising a synchronous motor and switching cell array power converter. The study is carried out through lumped parameter thermal networks using the numerical methodologies developed in Chapter 2.

Chapter 4 presents the development of the generalised modelling approach for heat exchangers, which compounds CFD models and reduced order approaches for an accurate evaluation of the thermal-hydraulic performance. This methodology is presented through the analysis of a case study comprising several configurations of heat exchangers. The study about the thermal performance of compact heat exchangers and extended surfaces is further expanded in Chapters 5 and 6, where heat transfer and friction properties of offset-strip and wavy fins are investigated through a comprehensive parametric analysis using a design of experiment approach, respectively. Chapter 7 reports the final concluding remarks of this thesis.

Reduced order modelling of physical systems

2.1. Introduction

2.1.1. Background and aims

Designing systems and devices to meet modern standards of efficiency and reliability expected by the market requires careful analysis of individual sub-systems and a careful study of overall system performance. The latter is often evaluated by means of lumped parameter methods, thanks to their versatility, ease of use, and a low computational effort. These computational tools are widely available as commercial software packages, with limited open and free alternatives, which are usually developed and tailored for very specific applications. Commercial computational tools usually feature practical user interfaces that facilitate their use, thereby improving productivity. However, these software packages are often closed from the point of view of the implementation, hiding from their users the modelling assumptions and numerical methods used to obtain the solution of the physical problem at hand. Furthermore, these aspects complicate possible expansions of the codebase needed to address particular problems or applications.

This chapter presents the development of a computational framework for the development and analysis of lumped parameters models of physical systems. This toolbox is developed with the highest possible degree of generality, so that this methodology can be adapted to the solution of any physical problem and to facilitate possible future interactions with higher order methods for the study of specific details and components.

NOMENCLATURE I

Latin letters

A	incidence matrix
B	upwind matrix operator
c_p	specific heat [$\text{J kg}^{-1} \text{K}^{-1}$]
c	vector of coefficients associated with the SI base units
C	heat capacity [JK^{-1}]
C_r	heat capacity rate [WK^{-1}]
C	diagonal matrix of the heat capacity rates of branches [WK^{-1}]
$\vec{d}$	Euclidean distance vector [m]
d	vector of unsteady terms in the continuity equation
D_h	hydraulic diameter [m]
<i>e</i>	error
e	vector of exponents associated with the SI base units
$\dot{E}_v$	mechanical losses due to viscous effects [W]
<i>f</i>	equivalent Darcy friction factor [–]
<i>g</i>	gravitational acceleration [m s^{-2}]
<i>G</i>	thermal conductance [WK^{-1}]
<i>h</i>	heat transfer coefficient [$\text{W m}^{-2} \text{K}^{-1}$]
<i>H</i>	volumetric heat generation coefficient [W m^{-3}]
I_{bou}	mask of boundary nodes
I_{int}	mask of internal nodes
<i>k</i>	constant source coefficient
k	vector of know terms in the mechanical energy balance [Pa]
<i>L</i>	length [m]
L	matrix of coefficients of the energy balance of thermal networks [WK^{-1}]
<i>m</i>	linear source coefficient

M	diagonal matrix of elements $\rho_j S_j$ [kg m^{-1}]
<i>n</i>	amount of
N	matrix operator of sum over branches
$\vec{n}$	normal unit vector
O	orientations of branch velocities [–]
<i>p</i>	fluid pressure [Pa]
<i>P</i>	total fluid pressure [Pa]
<i>q</i>	heat flux [W m^{-2}]
$\dot{Q}$	heat [W]
r	vector of known terms of the energy balance of thermal networks [W]
<i>R</i>	thermal resistance [KW^{-1}]
Re	Reynolds number $u D_h / \nu$ [–]
s	vector of known terms in the thermal energy balance [W]
<i>S</i>	surface area [m^2]
<i>t</i>	time [s]
<i>T</i>	temperature [K]
<i>u</i>	streamwise velocity component [m s^{-1}]
<i>U</i>	overall heat transfer coefficient [$\text{W m}^{-2} \text{K}^{-1}$]
$\vec{v}$	fluid velocity [m s^{-1}]
<i>V</i>	volume [m^3]
<i>W</i>	mechanical power provided by pumps and fans [W]
W	matrix of coefficients of the thermal energy balance [WK^{-1}]
$\vec{x}$	position in a Cartesian system
Y	matrix of velocity coefficients in the mechanical energy equation [Pa s m^{-1}]
<i>z</i>	gravimetric height [m]

NOMENCLATURE II*Greek letters*

β	concentrated loss coefficient [—]
γ	conversion coefficient
ε	effectiveness [—]
ε_r	emissivity [—]
λ	thermal conductivity [$\text{W m}^{-1} \text{K}^{-1}$]
ν	kinematic viscosity [$\text{m}^2 \text{s}^{-1}$]
ρ	density [kg m^{-3}]
σ	generalised source term
σ_B	Stefan Boltzmann constant [$\text{W m}^{-2} \text{K}^{-4}$]
τ	viscous stress tensor
ϕ	flow rate coefficient
φ	generic field
χ	generic numerical value
ψ	head coefficient

Subscripts

b	branches
ext	external
E	egress
i	node i
I	inlet
int	internal
j	branch j
L	losses
n	nodes
P	pump
ud	unidirectional

Superscripts

n	n -th time step
-----	-------------------

2.1.2. Literature survey

One of the most used toolboxes for the reduced-order modelling of complex systems is Simulink [122], thanks to its straightforward block diagram interface, it easily allows for the development of complex multi-domain models encompassing different physics types. The Simscape™ Fluids™ library includes the components models and governing equations for the modelling of one-dimensional fluid systems. The nodal values of fluid properties are propagated using upwind schemes, while energy fluxes are computed by means of smoothed upwind schemes. Streamwise heat diffusion is taken into account by means of a thermal conductance coefficient. Time advancement is handled by backwards Euler and trapezoidal schemes, with the former being employed right at the start of a transient analysis and when single components undergo discrete internal changes, such as the opening of a valve. Sparse matrices algorithms are available, and are employed automatically depending on the topology of the model. The FORTRAN 90 based GT-SUITE [123] is a computational tool that offers a similar block diagram interface to Simulink, and it is integrated with 3D CFD and FE tools for detailed components modelling. It is often used for the analysis of internal combustion engine and their ancillary systems. In GT-SUITE individual components are suitably divided in sub-volumes to the desired discretisation level. The continuity, momentum, and energy equations are solved on a 1D staggered grid, where scalar fields are defined at the centre of the aforementioned sub-volumes, while vector field are defined at their interface [124]. Time integration can be handled with explicit schemes with adaptive time stepping to satisfy the Courant condition. Similar functionalities are offered by Amesim and Dymola [125]. The latter is an integrated development tool based on the Modelica

language [126], built around a block-diagram style interface similar to that of other commercial tools of this type. Dymola employs a hierarchical object-oriented architecture to model complex systems by combining several detailed sub-models, allowing for improved code re-usability. These models include entries from the Modelica standard library and other open source and commercial libraries, as well as user defined models. Modelica [127], the Dymola back-end, is an object-oriented declarative language for modelling of complex multi-domain systems developed by the non-profit Modelica Association, which also curates the development of the standard library containing about 1400 components models and 1200 functions and subroutines. The Modelica Fluid library provides component models for one-dimensional fluid flow and heat transfer in pipe networks including valves, vessels, and fluid machines. The standard solver implements the DASSL method [128, 129] which employs backward differentiation formulas for the approximation of time derivatives and Newton's method for the solution of the system of algebraic equations. Other available solution algorithms are based on the Euler, Runge-Kutta, Newton-Raphson, and other numerical methods. Another programming language purposely built for numerical analysis is Scilab [130], developed by INRIA and ENPC researchers. Scilab is an interpreted language based on FORTRAN and C that offers a compact and straightforward syntax around the use and manipulation of matrices. Scilab includes large numerical libraries dedicated to the main branches of scientific computing such as linear algebra, numerical integration, and optimisation procedures. Scilab includes Xcos, a toolbox for the modelling and analysis of dynamical systems. It features a block-based architecture, as the previously cited software packages, and it offers features and capabilities comparable to Simulink. It features many ODE and DAEs solvers such as Crank-Nicolson, several Runge-Kutta methods, LSodar, CVode, and others.

In the scientific computing landscape Python [131, 132] has become a powerful tool thanks to its expressive syntax, comparable to executable pseudocode [133], its rich type system, and the vast amount of available libraries. Unlike other languages and tools specifically designed for array-based computations such as FORTRAN and MATLAB, Python is a general purpose language, offering native types and a syntax structure that are well suited for solving problems unrelated to array operations like string manipulations, graph traversals, finite state machines, or branch and bound algorithms [134]. Through its high level interface Python is also capable of exposing low level libraries written in FORTRAN, C, and C++ for performance-sensitive tasks as array based operations and linear algebra algorithms. NumPy [135] and SciPy [136] are the most widely used wrappers of this kind for scientific computing applications. The former provides a sophisticated n-dimensional array object with rich semantics, serving as the base of most if not all scientific computing workflows; in addition NumPy includes several mathematical routines, random number generators, linear algebra algorithms, Fourier transforms, and more. The latter, SciPy, provides a large selection of fundamental algorithms for optimisation, integration, interpolation, eigenvalue

problems, algebraic equations, differential equations, statistics and many others. In addition, it provides fast sparse linear algebra functions that are well suited for the efficient solution of large systems of equations. The standard interactive features of Python are improved by the IPython shell [137], which forms the back-end of most interactive environments, providing access to OS-level actions and commands, inspection of variables, and graphical utilities for 2D and 3D plots.

Python's features make it an excellent language for developing tools for analysing dynamical systems. PyPSA - Python for Power System Analysis - is an open-source toolbox for simulating and optimising modern power and energy systems [138]. These are modelled as networks of sources, loads, and distribution components. The code solves energy conservation at each node of the network by using the Newton-Raphson method. TESPpy - Thermal Engineering Systems in Python - is an open-source Python library for the analysis of thermal processes and systems [139]. This toolkit features network topology representation where the steady-state mass and energy conservation equations are solved at each node. Pressure differentials in pipes are evaluated by means of the Darcy-Weisbach equation. Fluid properties are computed by means of the CoolProp library [140]. The resulting system of equations is solved by means of the Raphson method as in PyPSA.

In some cases, particular problems require the development of dedicated software packages and tools that offer improved features with respect to their generalised counterparts. Apros [141] is a computational tool specifically developed for the analysis of thermal processes in nuclear and conventional power plants. In Apros the two-phase flow governing equations, formulated either by means of a five equation drift flux model or a full six equation method, are discretised by following a staggered Finite Volume approach. The physical domain is represented by a grid of nodes and branches. The resulting algebraic equations are solved by an iterative procedure. COBRA-TF [142] is similar tool specifically developed for the thermal analysis of water cooled nuclear reactors. In CTF the two-phase flow is solved by using a staggered Finite Volume approach, and a variant of the SIMPLE algorithm [143]. RELAP5 [144], implemented in FORTRAN, was developed for the transient analysis of light water reactor coolant systems during several operational scenarios. An analogous software package is the first-order finite volume code CATHARE [145–147], which also features a 3D module to model multidimensional effects in specific components.

2.1.3. Methodology

The methodology developed in this chapter is based upon three main pillars. These are invariant with respect to the specific physical problem tackled with the present numerical approach, and refer to core aspects as the representation of the domain, the management of units of measurement, and the modelling schemes used to represent non-linear physical phenomena by linearised source and sinks terms which can be assigned to topology entities. The three main pillars at the basis of the present methodology are:

- i. Spatial discretisation is based on the use of the graph theory, by representing the computational domain as set of nodes connected by directed and weighted branches. Nodes are points in space defined by their coordinates and a unique identifier, which can take the form of a numeric index or a text string. This type of identifier is used to uniquely define each branch, whose description also includes the identifiers of its starting and ending nodes. Information on nodes and branches is stored in associative arrays based on their identifiers. Once the topology of the whole system is defined the connections between nodes and branches are collected in the incidence matrix, which is then used as a based for the assembly and solution of the system of algebraic equations involved in the solution of the problem at hand.
- ii. An advanced management system for units of measurement is devised to check the dimensional consistency and to ensure correct propagation of the units in all operations involving physical quantities. This system is based on two arrays presenting the exponents and coefficients associated with the seven base units of the SI system, respectively. Exponents are used to define the type of physical quantity, while coefficients refer to the metric prefixes used in the SI system. An instance of these arrays is assigned to each scalar, vector, and matrix representing physical quantities. When an operation is performed dimensional consistency and units propagation are performed by comparing the exponents associated with the operands and applying suitable operations between them when needed. The vector of coefficients is used for the conversion of an arbitrarily defined physical quantity to its base-units counterpart. This procedure is performed as a first step of the solution of any physical problem, to avoid operations between inconsistent multiples or submultiples. Finally, the results are converted back to the set of units initially defined by the user.
- iii. The contributions to the source and sink terms often appearing in the governing equations of physical phenomena given by nodes and branches are expressed by a generalised linear formulation, where the independent variable is the value of the field associated with the considered governing equations and network entity. The coefficients of this linear model are suitably defined to represent the desired physical phenomena, managing any non-linear behaviour by iteratively computing the coefficients considering field values at the previous iteration of the solution process.

The generalised core functionalities presented above are used to develop solvers for one-dimensional flow in networks of pipes and for lumped parameter thermal networks. The former is developed on the basis of a staggered Finite Volume approach. The domain composed by nodes and branches is divided in Control Volumes (CVs) according to two different schemes, which are applied concurrently. Branch CVs include the volume of fluid associated with branches, and they are taken as a reference for the solution of the conservation of mechanical energy. On the other hand, the conservation equations for mass and

thermal energy are solved on node control volumes, which encompass half of the volume of fluid of the branches connected to each node. The pressure and velocity fields are uncoupled by a variant of the SIMPLE algorithm [143], which is used as basis to implement solvers for steady-state and transient problems. Components typically present in pipe networks such as pumps and valves are modelled by linearised sources and sinks of mechanical energy.

The solver for networks of thermal resistances presents a much more straightforward implementation, as the graph-based approach for the representation of the domain is particularly well suited for these problems. Each node of the network models the mean temperature value of the region of the domain identified by it, while branches, representing the thermal resistances, are used to model the heat transfer paths between temperature nodes. The values of the thermal resistances are computed iteratively as part of the solution algorithm to account for possible temperature dependencies linked to material or fluid properties, and natural convection or thermal radiation phenomena. This solver is used extensively in Chapter 3 for the thermal analysis of an integrated motor drive composed by a six-phase synchronous motor equipped with permanent magnets and its power converter, which presents a switching cell array architecture.

All the tools and algorithms presented in this Chapter are implemented in a Python 3 environment in the form of a structured stand-alone library, for improved ease of use and portability. While Python does not offer the computational speed of FORTRAN or C it has been nevertheless selected for this application thanks to its flexibility and straightforward syntax, which allow for lower development lead times and easier expandability compared to traditional compiled languages.

2.2. Domain discretisation

The spatial discretisation of any given physical domain is handled by means of a directed and weighted graph, which consists in a set of nodes that are connected by directed branches or edges. Figure 2.1 shows an example of a directed graph, where branches are represented as arrows to clearly indicate their direction.

Each node represents a point in space and is defined by a unique identifier, the *key*, which can be either an integer index, as in Figure 2.1, or a string of text. Additionally, each node possesses a set of generic coordinates:

$$\vec{x} = (x_1, x_2, x_3). \quad (2.1)$$

Two nodes can be connected by a branch, that is uniquely identified by its key and the coordinates of its starting and ending nodes, $\vec{x}_s$ and $\vec{x}_e$, respectively. The direction and length of a branch are derived as follows:

$$\vec{d} = (x_{1,e} - x_{1,s}, x_{2,e} - x_{2,s}, x_{3,e} - x_{3,s}), \quad (2.2)$$

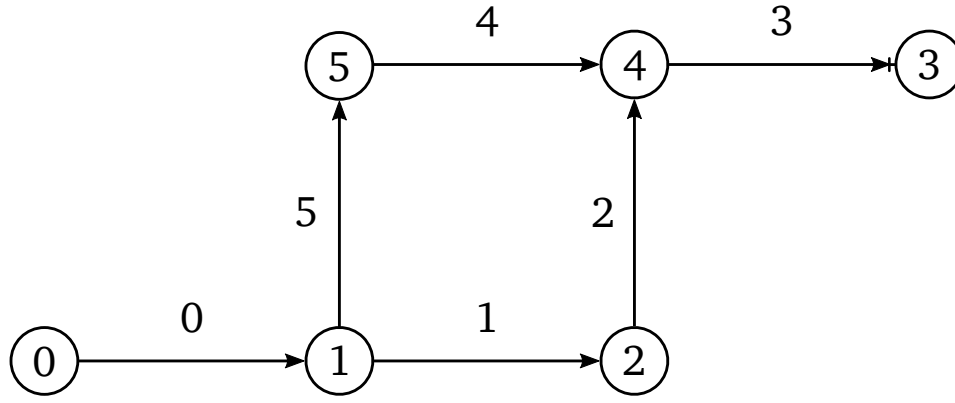


Figure 2.1. Example of a simple directed graph, where nodes and branches are identified by integer indices.

$$L = |\vec{d}|. \quad (2.3)$$

Moreover, branches may possess a series of additional properties depending on the particular physical properties under examination. For instance, in the context of networks of pipes, values of hydraulic diameter and cross-sectional area are assigned to each branch to solve the governing equations of fluid flow. The connections between nodes and branches are represented in a compact way by the elements of the $n_n \times n_b$ incidence matrix A :

$$A_{ij} = \begin{cases} 1 & \text{if node } i \text{ is the starting point of branch } j \\ -1 & \text{if node } i \text{ is the ending point of branch } j \\ 0 & \text{if there is no interaction between node } i \text{ and branch } j \end{cases}, \quad (2.4)$$

or, equivalently:

$$A_{ij} = \begin{cases} 1 & \text{if branch } j \text{ comes out from node } i \\ -1 & \text{if branch } j \text{ enters into node } i \\ 0 & \text{if there is no interaction between branch } j \text{ and node } i \end{cases}. \quad (2.5)$$

The use of Equations (2.4) and (2.5) results in the following incidence matrix for the graph in Figure 2.1.

$$A = \begin{pmatrix} 1 & 0 & 0 & 0 & 0 & 0 \\ -1 & 1 & 0 & 0 & 0 & 1 \\ 0 & -1 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & -1 & 0 & 0 \\ 0 & 0 & -1 & 1 & -1 & 0 \\ 0 & 0 & 0 & 0 & 1 & -1 \end{pmatrix} \quad (2.6)$$

Equations (2.4) and (2.5) are valid when considering branches that establish a bidirectional connection between their starting and ending nodes. These bidirectional branches can be

used to model the majority of physical systems. However, in some instances, unidirectional branches may be required to model specific systems or components. An example of such a component is the electrical diode, which allows for the passage of current only along one direction. The connections formed by unidirectional branches, like branch 3 in Figure 2.1, can be included in the incidence matrix as follows.

$$A_{\text{ud},ij} = \begin{cases} 1 & \text{if node } i \text{ is the starting point of unidirectional branch } j \\ 0 & \text{if node } i \text{ is the ending point of unidirectional branch } j \\ 0 & \text{if there is no interaction between node } i \text{ and branch } j \end{cases} \quad (2.7)$$

Equations (2.4) and (2.7) are used to obtain A_{ud} , which is the variant of the incidence matrix that explicitly includes unidirectional connections. The unidirectional incidence matrix of the network in Figure 2.1 is:

$$A_{\text{ud}} = \begin{pmatrix} 1 & 0 & 0 & 0 & 0 & 0 \\ -1 & 1 & 0 & 0 & 0 & 1 \\ 0 & -1 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & -1 & 1 & -1 & 0 \\ 0 & 0 & 0 & 0 & 1 & -1 \end{pmatrix}. \quad (2.8)$$

Typically, A is sparse, i.e. most of its elements are zero, since in most networks each node is usually connected to another single node or to a restricted group of nodes, thus not having any interaction with the rest of the nodes present in the network. Thus, A is stored using a Compressed Sparse Row (CSR) scheme, which includes three arrays: one for the non-zero elements of A , a second one for the indices of the columns of said elements, and a third array of integers where the i -th entry points to the start of row i in the first two arrays [148]. As an example, the three arrays making up the CSR representation of the incidence matrix in Equation 2.8 are reported in Table 2.1. The CSR format allows for the use of fast algorithms for the inner product between matrices and vectors, and the solution of linear systems. However, it is static, meaning that it is not easy to add or modify elements of the matrix. The data structures and algorithms for sparse linear algebra used for the development of the present numerical tool are available in the SciPy library [136].

Table 2.1. Representation of the matrix in Equation (2.8) in Compressed Sparse Row format.

Values	1	-1	1	1	-1	1	-1	1	-1	1	-1
Column indices	0	0	1	5	1	2	2	3	4	4	5
Row pointers	0	1	4	6	6	9	11				

2.3. Dimensioned values and units of measurement

Solving physical problems requires careful consideration of the units of measurement of the physical quantities involved, to ensure the consistency of the operations performed. Mismatches in the units of measurement can lead to serious design error, with potentially catastrophic consequences, as in the famous case of NASA's Mars Climate Orbiter [149]. To mitigate and possibly avoid these problems, an advanced management system for units of measurement is included in the present numerical toolbox. This framework is developed with the aim of performing operations between dimensioned quantities, while checking the consistency of their units of measurement, and ensuring the correct propagation of the units of the operands to the result. In addition, a practical text-based user interface is developed to simplify the definition of dimensioned quantities.

This framework is based on the use of two arrays, one for coefficients and one for exponents, that are associated with the International System of Units. An instance of these arrays is associated to each scalar, vector, and matrix involved in the solution of the governing equations of any physical problem, in order to define their units of measurement. The vectors of coefficients and exponents are:

$$\mathbf{c} = (c_{\text{kg}}, c_{\text{m}}, c_{\text{s}}, c_{\text{K}}, c_{\text{mol}}, c_{\text{A}}, c_{\text{cd}}) , \quad (2.9)$$

$$\mathbf{e} = (e_{\text{kg}}, e_{\text{m}}, e_{\text{s}}, e_{\text{K}}, e_{\text{mol}}, e_{\text{A}}, e_{\text{cd}}) , \quad (2.10)$$

where each subscript identifies the SI base unit to which each coefficient and exponent are referred. This system easily allows for the definition of multiples and sub-multiples of the base units, as well as derived and imperial units as these can be expressed by prescribing suitable values for $\mathbf{c}$ and $\mathbf{e}$.

A unit of measurement can be defined by directly specifying the values of the vectors of coefficients and exponents, or through a string of text formatted as follows:

$$\langle c1 \rangle \langle u1 \rangle^{\langle e1 \rangle} \langle c2 \rangle \langle u2 \rangle^{\langle e2 \rangle} \dots$$

The above string includes two dimensional groups, each formed by a coefficient symbol $\langle c \rangle$, a unit symbol $\langle u \rangle$, and a numerical exponent $\langle e \rangle$. Coefficients can be inserted directly as numerical values, or through the respective symbol of the desired multiple or sub-multiple according to the SI system, as in Table 2.2. For example, square millimetres and Pascal can be defined as mm^2 and $\text{kg m}^{-1} \text{s}^{-2}$, respectively. Alternatively, derived units like Pascal, Watt, and others can be directly introduced in the text string by linking their symbol to their representation in SI base units:

$$\begin{aligned} \text{Pa:} & \quad \text{kg m}^{-1} \text{s}^{-2} \\ \text{W:} & \quad \text{kg m}^2 \text{s}^{-3} \end{aligned}$$

this information is stored in a dedicated associative array. The text strings used to define units of measurement are translated into the vectors of coefficients and exponents by means

of a recursive parsing function. The latter substitutes the symbols for derived and imperial units with their base units representation and associates each symbol with its respective numerical value.

2.3.1. Operations between dimensioned values

Operations between dimensioned quantities involve both the numerical value and the units of measurement. While operations between numerical values are straightforward, those between units of measurement must be suitably defined to ensure the correct propagation of the units, and to avoid invalid operations between incompatible units. The definitions of the operations between dimensioned quantities are provided in the following by considering scalar values. These operations are extended to dimensioned vectors and matrices by performing the numerical operations using an element-wise approach, while executing the operations between units just once.

2.3.1.1. Conversion to SI base units

The conversion of a dimensioned value with generic units to its SI base counterpart is performed by multiplying the numerical value χ by the conversion coefficient γ_{SI} :

$$\chi_{\text{SI}} = \gamma_{\text{SI}} \chi. \quad (2.11)$$

The term γ_{SI} is obtained from the product of the coefficients elevated to their corresponding exponents:

$$\gamma_{\text{SI}} = \prod_{k=1}^7 c_k^{e_k}. \quad (2.12)$$

After the conversion of the numerical value the components of $\mathbf{e}$ are uniformly set to 1, while the exponents are left unchanged.

2.3.1.2. Algebraic sum

The algebraic sum between two dimensioned values can be performed only when the units of the two values are consistent, i.e. when their $\mathbf{e}$ vectors present the same values. If this condition is not met an error message is issued to the user letting him know that an

Table 2.2. Symbols and values associated with the multiples and sub-multiples of the SI system.

Sub-multiples	symbol value	p 10^{-12}	n 10^{-9}	mu 10^{-6}	m 10^{-3}	c 10^{-2}	d 10^{-1}
Multiples	symbol value	da 10^1	h 10^2	k 10^3	M 10^6	G 10^9	T 10^{12}

invalid operation was attempted. The example below shows the error message issued when attempting to sum a value of length in meters to a value of time in seconds.

UnitsArithmeticError: Units [m] and [s] are not compatible
for the operation: +

If the two dimensioned values present the same e -values the algebraic sum of the numerical values is performed as follows:

$$\chi_3 = \gamma_1 \chi_1 + \gamma_2 \chi_2 . \quad (2.13)$$

If the two dimensioned values feature the same vectors of coefficients, γ_1 and γ_2 are set to 1. Otherwise, χ_1 and χ_2 are converted to SI based units using γ_{SI} obtained from Equation (2.12). The newly obtained dimensioned quantity features the same exponents as the operands and the suitable coefficients depending on the condition mentioned above.

2.3.1.3. Multiplication

Multiplications between dimensioned quantities can always be performed regardless of their units, as long as their correct propagation is ensured. The numerical value resulting from the operation is obtained as follows:

$$\chi_3 = \gamma \chi_1 \chi_2 , \quad (2.14)$$

where γ is set to 1 or $\gamma_{SI,1} \gamma_{SI,2}$ depending on whether the coefficients of the two quantities are equal or not. The exponents associated with the resulting dimensioned quantity are obtained as follows:

$$\mathbf{e}_3 = \mathbf{e}_1 + \mathbf{e}_2 . \quad (2.15)$$

2.3.1.4. Division

Division operations are defined along the same lines as multiplications. The numerical operation is defined as:

$$\chi_3 = \gamma \frac{\chi_1}{\chi_2} , \quad (2.16)$$

where γ is set to 1 or $\gamma_{SI,1} / \gamma_{SI,2}$ using the same criteria used for multiplications. The exponents associated with the resulting dimensioned quantity are:

$$\mathbf{e}_3 = \mathbf{e}_1 - \mathbf{e}_2 . \quad (2.17)$$

2.3.1.5. Power

When performing power operations there is no need for units conversion as these operations involve a single dimensioned quantity. The coefficients and exponents obtained by elevating a dimensioned quantity to the power ξ are:

$$\mathbf{c}_2 = \mathbf{c}_1^\xi, \quad (2.18)$$

$$\mathbf{e}_2 = \mathbf{e}_1 \xi. \quad (2.19)$$

2.4. Generalised modelling of source and sink terms

Source and sink terms often appear in the equations governing most physical problems. In the framework of the present computational tool, source and sink terms can be associated with both nodes and branches depending on the physical problem at hand. The contribution given by each node or branch is expressed by using a generalised linear semi-implicit scheme:

$$\sigma_r = m_r \varphi_r + k_r, \quad (2.20)$$

where the subscript r stands for the index of the node or branch that produces the source term σ_r . The term φ_r represents the value of a generic field associated with the specific network entity under consideration. This formulation allows for easily modelling any linear or non-linear physical phenomena, by concentrating non-linearity in the coefficients m_r and k_r , and evaluating them using field values related to the previous iteration of the employed solution algorithm.

For instance, when modelling convective heat transfer phenomena related to a given branch j , these coefficients can be obtained by comparing Equation (2.20) with Newton's law of cooling:

$$\sigma_j = m_j T_j + k_j = \dot{Q} = U_j S_{\text{ref},j} (T_\infty - T_j), \quad (2.21)$$

leading for the following definitions for m_j and k_j :

$$m_j = -U_j S_{\text{ref},j}, \quad (2.22)$$

$$k_j = U_j S_{\text{ref},j} T_\infty. \quad (2.23)$$

By using the approach highlighted above, source terms modelling fluid flow and heat transfer in networks of pipes are developed in Section 2.5.3.

2.5. Development of a one-dimensional flow solver

2.5.1. Governing equations

Fluid flow within networks of pipes and ducts is governed by macroscopic balance equations of mass, mechanical energy, and thermal energy. These equations, expressed in integral form, read:

$$\int_V \frac{\partial \rho}{\partial t} dV + \int_S \rho \vec{v} \cdot \vec{n} dS = 0, \quad (2.24)$$

$$\begin{aligned} \frac{d}{dt} \int_{V(t)} \rho \left(\frac{v^2}{2} + g z \right) dV = & - \int_S \rho \left(\frac{v^2}{2} + g z \right) (\vec{v} - \vec{v}_s) \cdot \vec{n} dS \\ & - \int_S p \vec{v} \cdot \vec{n} dS + \int_S (\tau \vec{v}) \cdot \vec{n} dS \\ & + \int_V p \nabla \cdot \vec{v} dV - \int_V \tau : \nabla \vec{v} dV, \end{aligned} \quad (2.25)$$

$$\begin{aligned} \frac{\partial}{\partial t} \int_V \rho c_p T dV + \int_S \rho c_p T \vec{v} \cdot \vec{n} dS = & + \int_S -\vec{q} \cdot \vec{n} dS + \int_V H dV \\ & + \int_V \tau : \nabla \vec{v} dV. \end{aligned} \quad (2.26)$$

Equation (2.25) is obtained by multiplication of the momentum balance equation by the velocity field $\vec{v}$ and integration over a moving Control Volume (CV) $V(t)$ using the Leibniz formula. In Equation (2.25) $\vec{v}_s$ is the velocity of the moving vents of the control volume.

The governing equations are solved using a low-Mach-number approach. The fluid is considered incompressible under the assumption that fluid velocity is always far lower than the speed of sound, which is roughly 1480 m/s for water and 340 m/s for air. However, variations of fluid density resulting from temperature and pressure changes are taken into account. The density field is computed from a state equation or by linear interpolation of tabled data as a function of pressure and temperature, then the governing equations are solved on the basis of this imposed density field. This approach leads to a two-way coupling between the velocity and temperature fields.

2.5.2. Discretisation schemes

The fields of density, velocity, pressure, and temperature are obtained by numerically solving the flow governing equations using a Staggered Finite Volume approach. The domain is subdivided in control volumes by employing two discretisation schemes concurrently. Two distinct types of spatially fixed control volumes are defined:

- **Branch control volume:** it includes the fluid volume inside the branch, and it features a uniform arbitrary cross-section over its length. The vents are placed in correspondence of the branch starting and ending nodes, and they are assumed planar and

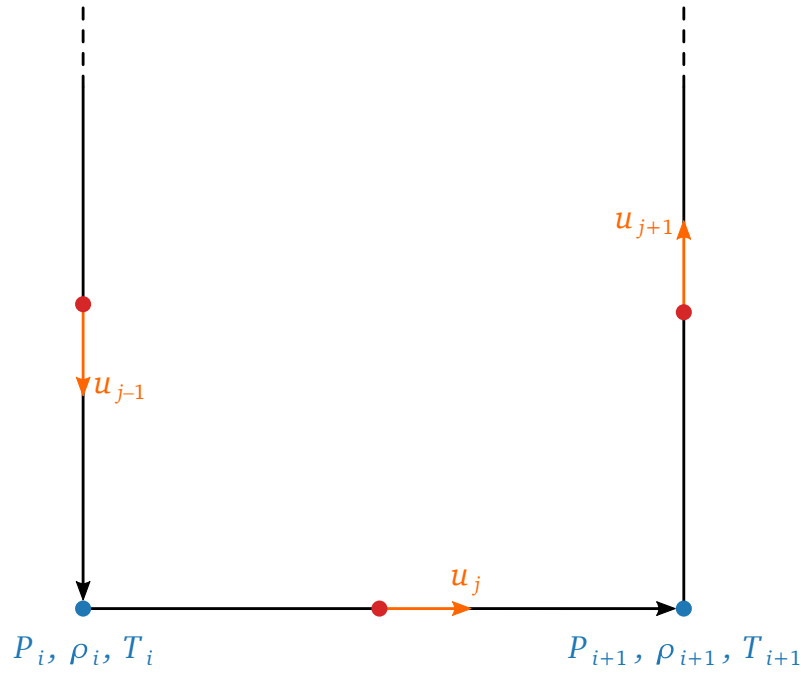


Figure 2.2. Examples of staggered arrangement of nodes and branches, where blue points stand for nodes, while red ones represent the centre of branches.

perpendicular to the axis of the branch. The normal unit vectors associated with the vents are assumed positive if pointing outwards. Figure 2.3a shows an example of a branch control volume with a round cross-section, where blue points represent the nodes, while the red one stands for the centre of the branch.

- **Node control volume:** it comprises half the volume of fluid associated with the branches connected to a node of the network. The vents of the control volume are located in correspondence of the centre points of the branches, and planar and normal to the axes of the branches. An example of this control volume is represented in Figure 2.3b.

Equations (2.24) and (2.26) are applied to each node CV of the network. Conversely, the mechanical energy global balance, which is expressed by Equation (2.25), is applied at each branch control volume. This approach leads to the definition of a staggered grid where the average values of fluid total pressure, density and temperature (P, ρ, T) are defined at each node, while the mean velocity u is assigned to branches, positive if oriented branch-wise. An example of this arrangement is shown in Figure 2.2.

Nodal field values are transferred to branches by using an upwind scheme of first order accuracy, whose details are discussed in Appendix A.1. This scheme is applied by defining the following operator:

$$B = 0.5 \left(|A^T| + A^T O \right), \quad (2.27)$$

where O is a $n_b \times n_b$ diagonal matrix indicating the orientations of branch velocities:

$$O_{jj} = \frac{u_j}{|u_j|}. \quad (2.28)$$

The operator $\mathbf{B}$ is essentially a modified version of the transposed incidence matrix: for any given row, positive elements are kept if fluid velocity of the related branch is positive, meaning that the upwind node is the starting point of the branch considered, as in Equation (2.4). Instead, negative elements are deleted. When fluid velocity is negative, positive elements are deleted, while negative ones are kept in absolute value, meaning that the upwind node is the end point of the branch under examination.

The operator $\mathbf{B}$ is applied to the generic nodal field φ_n as follows:

$$\varphi_b = \mathbf{B} \varphi_n. \quad (2.29)$$

In the following, the methodology used for the discretisation of the governing equations is discussed in detail.

2.5.2.1. Macroscopic mass balance

Mass conservation is enforced at each node control volume of the network. Fluid velocity is assumed as aligned with the normal unit vectors at the vents of the CVs. The latter are considered planar and fixed in space. The integrals in Equation (2.24) are solved using the midpoint method, which is described in detail in Appendix A.2. Thus, the integral averages of fluid velocity and density are substituted with their midpoint values, introducing a second order error. Equation (2.24) applied to the i -th node becomes:

$$\frac{\partial \bar{\rho}_i}{\partial t} V_i + \sum_j^{n_E} \bar{\rho}_j \bar{u}_j S_j - \sum_j^{n_I} \bar{\rho}_j \bar{u}_j S_j = 0, \quad (2.30)$$

where over-bars indicate values taken at the centre point of the surface or volume under consideration. S_j is the cross-sectional area of branch j , while n_I and n_E stand for the number of branches entering and exiting node i , respectively. u is the velocity component oriented as the branches. By rearranging the terms of Equation (2.30) and integrating in time between two subsequent time steps t^n and $t^{n+1} = t^n + \Delta t$, the following is obtained:

$$(\bar{\rho}_i^{n+1} - \bar{\rho}_i^n) V_i = \int_n^{n+1} \left(\sum_j^{n_I} \bar{\rho}_j \bar{u}_j S_j - \sum_j^{n_E} \bar{\rho}_j \bar{u}_j S_j \right) dt. \quad (2.31)$$

Time integration on the right side of Equation (2.31) is carried out by using the implicit Euler method of first order accuracy (see Appendix A.3):

$$\frac{\bar{\rho}_i^{n+1} - \bar{\rho}_i^n}{\Delta t} V_i = \sum_j^{n_I} \bar{\rho}_j^{n+1} \bar{u}_j^{n+1} S_j - \sum_j^{n_E} \bar{\rho}_j^{n+1} \bar{u}_j^{n+1} S_j. \quad (2.32)$$

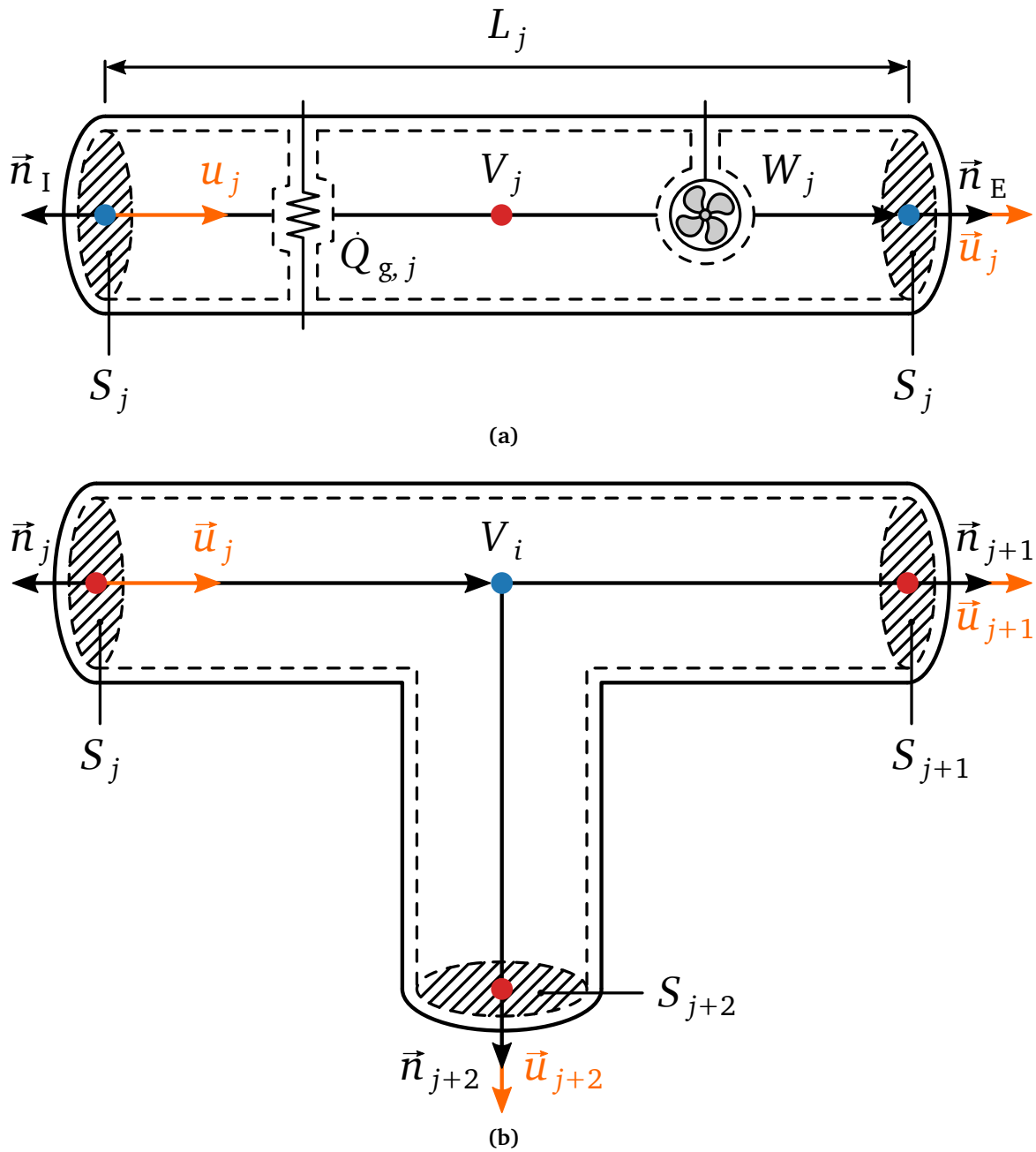


Figure 2.3. Example of a branch control volume (a) and node control volume (b), where blue points stand for nodes while the red ones represent the centres of branches.

The sign of mass flux terms is managed by introducing elements of the incidence matrix. According to Equation (2.5), exhaust branches connected to a given node i are associated to values of 1 of the A_{ij} coefficients, while inlet ones are related to values of -1. Equation (2.32) becomes:

$$\sum_{j=1}^{n_b} A_{ij} \bar{\rho}_j^{n+1} \bar{u}_j^{n+1} S_j = -\frac{\bar{\rho}_i^{n+1} - \bar{\rho}_i^n}{\Delta t} V_i. \quad (2.33)$$

Equation (2.33) is applied to each node of the network, leading to a system of n_n equations:

$$\mathbf{A} \mathbf{M} \mathbf{u}^{n+1} = -\mathbf{d}, \quad (2.34)$$

where $\mathbf{u}^{n+1}$ is the array of branch velocities calculated at the time step $n+1$, while $\mathbf{M}$ is a $n_b \times n_b$ diagonal matrix of elements

$$M_{jj} = \bar{\rho}_j^{n+1} S_j, \quad (2.35)$$

and $\mathbf{d}$ is the array of unsteady terms:

$$d_i = \frac{\bar{\rho}_i^{n+1} - \bar{\rho}_i^n}{\Delta t} V_i \quad (2.36)$$

In case of steady-state flow Equation (2.34) becomes:

$$\mathbf{A} \mathbf{M} \mathbf{u} = \mathbf{0}. \quad (2.37)$$

2.5.2.2. Macroscopic mechanical energy balance

The volume integral on the left-hand side of Equation (2.25) is solved by using the midpoint rule of second order accuracy:

$$\frac{d}{dt} \int_{V(t)} \rho \left(\frac{v^2}{2} + gz \right) dV \approx \frac{d}{dt} \left[\rho_j \left(\frac{\bar{u}_j^2}{2} S_j L_j + g \bar{z}_j V_j \right) \right]. \quad (2.38)$$

The advective terms are solved following the same procedure:

$$\begin{aligned} \int_S \rho \left(\frac{v^2}{2} + gz \right) (\vec{v} - \vec{v}_s) \cdot \vec{n} dS &\approx \bar{\rho}_E \left(\frac{\bar{u}_j^2}{2} + g \bar{z}_E \right) \bar{u}_j S_j \\ &\quad - \bar{\rho}_I \left(\frac{\bar{u}_j^2}{2} + g \bar{z}_I \right) \bar{u}_j S_j, \end{aligned} \quad (2.39)$$

$$\int_S p \vec{v} \cdot \vec{n} dS \approx \bar{p}_E \bar{u}_j S_j - \bar{p}_I \bar{u}_j S_j - \dot{W}_{p,j}, \quad (2.40)$$

where $\dot{W}_{p,j}$ stands for the work per unit time given by fluid pressure to impermeable walls. The work per unit time performed by viscous forces is expressed as $\dot{W}_{\tau,j}$:

$$\int_S (\boldsymbol{\tau} \cdot \vec{v}) \cdot \vec{n} dS = \dot{W}_{\tau,j}. \quad (2.41)$$

Compression power, i.e. the term of Equation (2.25) containing the divergence of velocity is neglected, as, in a lumped parameters framework, its effects are usually compounded in other terms as the work performed or extracted by fluid machines and head losses. Equations (2.39) to (2.41) are introduced into Equation (2.25) leading to the following relation:

$$\begin{aligned} \frac{d}{dt} \left[\bar{\rho}_j \left(\frac{\bar{u}_j^2}{2} S_j L_j + g \bar{z}_j V_j \right) \right] = & - \left(\bar{\rho}_E \frac{\bar{u}_j^2}{2} + \bar{p}_E + \bar{\rho}_E g \bar{z}_E \right) \bar{u}_j S_j \\ & + \left(\bar{\rho}_I \frac{\bar{u}_j^2}{2} + \bar{p}_I + \bar{\rho}_I g \bar{z}_I \right) \bar{u}_j S_j \\ & + \dot{W}_{p,j} + \dot{W}_{\tau,j} - \dot{E}_{v,j}. \end{aligned} \quad (2.42)$$

In Equation (2.42) $\dot{E}_{v,j}$ stands for the energy per unit time dissipated by viscous effects. Equation (2.42) is simplified by introducing the total pressure of the fluid, which referred to the i -th node reads:

$$P_i = \bar{\rho}_i \frac{\bar{u}_i^2}{2} + \bar{p}_i + \bar{\rho}_i g \bar{z}_i. \quad (2.43)$$

Thus, Equation (2.42) becomes:

$$\frac{d}{dt} \left(\bar{\rho}_j \frac{\bar{u}_j^2}{2} + \bar{\rho}_j g \bar{z}_j \right) V_j = P_I \bar{u}_j S_j - P_E \bar{u}_j S_j + \dot{W}_j - \dot{E}_{v,j}, \quad (2.44)$$

where $\dot{W}_j = \dot{W}_{p,j} + \dot{W}_{\tau,j}$ is the work per unit time done or extracted by mechanical devices, such as pumps or fans, as shown in Figure 2.3a. Time integration is again carried out by using the implicit Euler method of first order accuracy (see Appendix A.3).

$$\begin{aligned} \frac{V_j}{\Delta t} \left[\bar{\rho}_j^{n+1} \left(\frac{(\bar{u}_j^{n+1})^2}{2} + g \bar{z}_j \right) - \bar{\rho}_j^n \left(\frac{(\bar{u}_j^n)^2}{2} + g \bar{z}_j \right) \right] = & + P_I^{n+1} \bar{u}_j^{n+1} S_j \\ & - P_E^{n+1} \bar{u}_j^{n+1} S_j \\ & + \dot{W}_j^{n+1} - \dot{E}_{v,j}^{n+1} \end{aligned} \quad (2.45)$$

All terms of Equation (2.45) are divided by the branch volumetric flow rate $\bar{u}_j^{n+1} S_j$. The division is exact for all terms related to the time step $n + 1$. However, for the one referred

to t^n , a first-order error is introduced:

$$e = \frac{(u^n)^2}{u^n} - \frac{(u^n)^2}{u^{n+1}} = \frac{du}{dt} \Delta t - \left(\frac{du}{dt} \right)^2 \frac{\Delta t^2}{u^{n+1}} + \dots \quad (2.46)$$

Equation (2.45) becomes:

$$\begin{aligned} \frac{L_j}{\Delta t} \left[\bar{\rho}_j^{n+1} \left(\frac{\bar{u}_j^{n+1}}{2} + g \bar{z}_j \right) - \bar{\rho}_j^n \left(\frac{\bar{u}_j^n}{2} + g \bar{z}_j \right) \right] = & + P_I^{n+1} - P_E^{n+1} \\ & + \Delta p_{P,j}^{n+1} \\ & - \Delta p_{L,j}^{n+1}, \end{aligned} \quad (2.47)$$

where $\Delta p_{P,j}^{n+1}$ and $\Delta p_{L,j}^{n+1}$ stand for the pressure differentials given by pumps of fans, and viscous losses, respectively. These terms are now substituted with generalised source and sink terms, according to Equation (2.20).

$$\begin{aligned} \frac{L_j}{\Delta t} \left[\bar{\rho}_j^{n+1} \left(\frac{\bar{u}_j^{n+1}}{2} + g \bar{z}_j \right) - \bar{\rho}_j^n \left(\frac{\bar{u}_j^n}{2} + g \bar{z}_j \right) \right] = & + P_I^{n+1} - P_E^{n+1} \\ & + \sigma_{P,j}^{n+1} \\ & - \sigma_{L,j}^{n+1} \end{aligned} \quad (2.48)$$

By introducing the coefficients m and k , and reordering the terms of Equation (2.48), the mechanical energy macroscopic balance applied at the j -th branch becomes:

$$\begin{aligned} P_I^{n+1} - P_E^{n+1} = & + \left(m_{L,j}^{n+1} - m_{P,j}^{n+1} + \frac{\bar{\rho}_j^{n+1} L_j}{2 \Delta t} + \frac{\bar{\rho}_j^{n+1} g \bar{z}_j L_j}{\Delta t} \right) \bar{u}_j^{n+1} \\ & + \left(-k_{P,j}^{n+1} - \frac{\bar{\rho}_j^n \bar{u}_j^n L_j}{2 \Delta t} - \frac{\bar{\rho}_j^n g \bar{z}_j L_j}{\Delta t} \right). \end{aligned} \quad (2.49)$$

By recalling the definition of the incidence matrix of the network (Equation (2.4)) the n_b equations can be expressed in compact form as follows:

$$\mathbf{A}^T \mathbf{P}^{n+1} = \mathbf{Y} \mathbf{u}^{n+1} + \mathbf{k}, \quad (2.50)$$

where $\mathbf{P}^{n+1}$ is the vector of nodal total pressure values referred to the time step $n+1$, while $\mathbf{Y}$ is a $n_b \times n_b$ diagonal matrix of elements:

$$Y_{jj} = m_{L,j}^{n+1} - m_{P,j}^{n+1} + \frac{\rho_j^{n+1} L_j}{2 \Delta t} + \frac{\bar{\rho}_j^{n+1} g \bar{z}_j L_j}{\Delta t}, \quad (2.51)$$

and $\mathbf{k}$ is the vector of terms:

$$k_j = -k_{p,j}^{n+1} - \frac{\bar{\rho}_j^n \bar{u}_j^n L_j}{2 \Delta t} - \frac{\bar{\rho}_j^n g \bar{z}_j L_j}{\Delta t} \quad (2.52)$$

When the flow is steady the terms Y_{jj} and k_j become:

$$Y_{jj} = m_{L,j} - m_{p,j} \quad (2.53)$$

$$k_j = -k_{p,j} \quad (2.54)$$

2.5.2.3. Thermal energy macroscopic balance

As previously anticipated, the thermal energy macroscopic balance is applied to each node control volume of the network. The heat entering and leaving branches is accounted for in the thermal energy balance of their downstream nodes, according to the direction of the flow. Equation (2.26) applied to the i -th node CV becomes:

$$\begin{aligned} \frac{\partial}{\partial t} (\bar{\rho}_i \bar{c}_{p,i} \bar{T}_i) V_i + \sum_j^{n_E} C_{r,j} T_j - \sum_j^{n_I} C_{r,j} T_j \\ + \sum_j^{n_I} (-m_{e,j}) \bar{T}_j = \sum_j^{n_I} (k_{e,j} + \Delta p_{L,j} u_j S_j) , \end{aligned} \quad (2.55)$$

where $C_{r,j}$ is the heat capacity rate of branch j :

$$C_{r,j} = \bar{\rho}_j \bar{u}_j S_j \bar{c}_{p,j} \quad (2.56)$$

In Equation (2.55) the integral terms present in Equation (2.26) referring to the heat transferred through the surface of the branch CV and volumetric heat generation are substituted with generalised source terms:

$$\int_S -\vec{q} \cdot \vec{n} dS = m_{e,j} \bar{T}_j + k_{e,j} , \quad (2.57)$$

$$\int_V H dV = m_{e,j} \bar{T}_j + k_{e,j} . \quad (2.58)$$

In addition, heat diffusion along the branch-wise direction is neglected. Time integration is once again carried out by using the implicit Euler method of first order accuracy, as discussed

in Appendix A.3:

$$\begin{aligned} & \frac{V_i}{\Delta t} \left(\bar{\rho}_i^{n+1} \bar{c}_{p,i}^{n+1} \bar{T}_i^{n+1} - \bar{\rho}_i^n \bar{c}_{p,i}^n \bar{T}_i^n \right) + \sum_j^{n_E} C_{r,j}^{n+1} \bar{T}_j^{n+1} \\ & - \sum_j^{n_I} C_{r,j}^{n+1} \bar{T}_j^{n+1} + \sum_j^{n_I} \left(-m_{e,j}^{n+1} \right) \bar{T}_j^{n+1} = \sum_j^{n_I} \left(k_{e,j}^{n+1} + \Delta p_{L,j}^{n+1} u_j^{n+1} S_j \right). \end{aligned} \quad (2.59)$$

The sign of flux terms is once more managed by introducing elements of the incidence matrix of the network. As, according to Equation (2.5), the A_{ij} coefficients associate +1 coefficients to exhaust branches and -1 values to inlet ones.

$$\begin{aligned} & \frac{V_i}{\Delta t} \left(\bar{\rho}_i^{n+1} \bar{c}_{p,i}^{n+1} \bar{T}_i^{n+1} - \bar{\rho}_i^n \bar{c}_{p,i}^n \bar{T}_i^n \right) + \sum_j^{n_b} A_{ij} C_{r,j}^{n+1} \bar{T}_j^{n+1} \\ & + \sum_j^{n_I} \left(-m_{e,j}^{n+1} \right) \bar{T}_j^{n+1} = \sum_j^{n_I} \left(k_{e,j}^{n+1} + \Delta p_{L,j}^{n+1} u_j^{n+1} S_j \right). \end{aligned} \quad (2.60)$$

The summation of $-m_{e,j}^{n+1}$ and $k_{e,j}^{n+1}$ terms over inlet branches is carried out by defining the matrix operator N , which is based on the incidence matrix of the network:

$$N_{ij} = \frac{1}{2} \left(|A_{ij}| - A_{ij} \frac{u_j}{|u_j|} \right). \quad (2.61)$$

When the value of fluid velocity related to a given branch j is positive, the downstream node of said branch is its ending node. Therefore, in this instance, the column of N associated with branch j reports the negative values of same column of the incidence matrix taken in absolute value, while the positive coefficients are discarded. Otherwise, when the velocity of branch j is negative, negative coefficients of the associated column of the incidence matrix are discarded, while positive ones are kept.

Equation (2.60) becomes:

$$\begin{aligned} & \frac{V_i}{\Delta t} \bar{\rho}_i^{n+1} \bar{c}_{p,i}^{n+1} \bar{T}_i^{n+1} + \sum_j^{n_b} A_{ij} C_{r,j}^{n+1} \bar{T}_j^{n+1} + \sum_j^{n_b} N_{ij} \left(-m_{e,j}^{n+1} \right) \bar{T}_j^{n+1} = \\ & = \sum_j^{n_b} N_{ij} \left(k_{e,j}^{n+1} + \Delta p_{L,j}^{n+1} u_j^{n+1} S_j \right) + \frac{V_i}{\Delta t} \bar{\rho}_i^n \bar{c}_{p,i}^n \bar{T}_i^n. \end{aligned} \quad (2.62)$$

Temperature values associated with the centre points of branches are obtained from nodal ones by employing the upwind operator $\mathbf{B}$:

$$\begin{aligned} & \frac{V_i}{\Delta t} \bar{\rho}_i^{n+1} \bar{c}_{p,i}^{n+1} \bar{T}_i^{n+1} + \sum_j^{n_b} A_{ij} C_{r,j}^{n+1} \sum_i^{n_n} B_{ji} \bar{T}_i^{n+1} + \sum_j^{n_b} N_{ij} (-m_{e,j}^{n+1}) \sum_i^{n_n} B_{ji} \bar{T}_i^{n+1} \\ &= \sum_j^{n_b} N_{ij} \left(k_{e,j}^{n+1} + \Delta p_{L,j}^{n+1} u_j^{n+1} S_j \right) + \frac{V_i}{\Delta t} \bar{\rho}_i^n \bar{c}_{p,i}^n \bar{T}_i^n. \end{aligned} \quad (2.63)$$

Equation (2.63) is applied to each node control volume. The resulting system of n_n equations is:

$$\mathbf{W} \mathbf{T}^{n+1} = \mathbf{s}, \quad (2.64)$$

where $\mathbf{W}$ is a $n_n \times n_n$ matrix of elements:

$$W_{ij} = \frac{V_i}{\Delta t} \bar{\rho}_i^{n+1} \bar{c}_{p,i}^{n+1} \delta_{ij} + A_{ij} C_{r,j}^{n+1} \sum_i^{n_n} B_{ji} + N_{ij} (-m_{e,j}^{n+1}) \sum_i^{n_n} B_{ji} \quad (2.65)$$

where δ_{ij} is the Kronecker delta. $\mathbf{W}$ can also be obtained from the following relation:

$$\mathbf{W} = \mathbf{W}_u + (\mathbf{A}\mathbf{C} + \mathbf{N}\mathbf{M}_e) \mathbf{B}, \quad (2.66)$$

where $\mathbf{C}$ is the $n_b \times n_b$ diagonal matrix of the heat capacity rates of branches, computed according to Equation (2.56). $\mathbf{M}_e$ is the $n_b \times n_b$ diagonal matrix made up by the coefficients $-m_{e,j}^{n+1}$, while $\mathbf{W}_u$ is the $n_n \times n_n$ diagonal matrix unsteady terms:

$$W_{u,ii} = \frac{V_i}{\Delta t} \bar{\rho}_i^{n+1} \bar{c}_{p,i}^{n+1}. \quad (2.67)$$

In Equation (2.64) $\mathbf{s}$ is the vector of terms:

$$s_i = \sum_j^{n_b} N_{ij} \left(k_{e,j}^{n+1} + \Delta p_{L,j}^{n+1} u_j^{n+1} S_j \right) + \frac{V_i}{\Delta t} \bar{\rho}_i^n \bar{c}_{p,i}^n \bar{T}_i^n. \quad (2.68)$$

In case of steady-state flow the matrix $\mathbf{W}$ and vector $\mathbf{s}$ become:

$$W_{ij} = A_{ij} C_{r,j} \sum_i^{n_n} B_{ji} + N_{ij} (-m_{e,j}) \sum_i^{n_n} B_{ji}, \quad (2.69)$$

$$s_i = \sum_j^{n_b} N_{ij} \left(k_{e,j}^{n+1} + \Delta p_{L,j}^{n+1} u_j^{n+1} S_j \right). \quad (2.70)$$

2.5.3. Modelling of components of networks of pipes

2.5.3.1. Centrifugal pumps and fans

Centrifugal pumps and fans are machines able to increase the mechanical energy of the fluid by exploiting centrifugal forces. In general, the head given by a pump, expressed in terms of pressure, is a quadratic function of fluid velocity:

$$\Delta p_p = c_0 + c_1 u + c_2 u^2. \quad (2.71)$$

Equation (2.71) is related to a reference operational regime, or in other words to particular value of the rotational speed of the shaft. The relationship between the head, the flow rate, and the speed of the shaft is expressed by a non-dimensional curve obtained from the hydraulic similarity theory [150], assuming that the various working points lay on the same hydraulic efficiency curve:

$$\psi = c'_0 + c'_1 \phi + c'_2 \phi^2, \quad (2.72)$$

where ϕ and ψ are the flow rate and head coefficients:

$$\phi = \frac{u S}{\text{rps } D_{\text{ref}}^3}, \quad (2.73)$$

$$\psi = \frac{\Delta p_p}{\rho (\text{rps } D_{\text{ref}})^2}. \quad (2.74)$$

D_{ref} is reference length of the pump, while rps stands for the rotational speed of the shaft in revolutions per second. An example of a non-dimensional pump curve is shown in Figure 2.4. For low ϕ values the head coefficient increases up to the vertex of the parabola, then it starts decreasing. This trend tends to hinder convergence of the solution algorithm, if the flow rate coefficient takes small values for the first iterations. Therefore, Equation (2.72) is modified to obtain a smooth monotonic function to improve the convergence of the iterative solution algorithm. A second non-dimensional function can be obtained by mirroring the parabola defined by (2.72) across a horizontal axis passing through the vertex point (ϕ_v, ψ_v) , as in Figure 2.4:

$$\psi' = 2\psi(\phi_v) - c'_0 - c'_1 \phi - c'_2 \phi^2. \quad (2.75)$$

Then, a smooth monotonic function is obtained by combining Equations (2.72) and (2.75) as follows:

$$\psi'' = \begin{cases} 2\psi(\phi_v) - c'_0 - c'_1 \phi - c'_2 \phi^2 & \text{for } \phi < \phi_v \\ c'_0 + c'_1 \phi + c'_2 \phi^2 & \text{for } \phi \geq \phi_v \end{cases} \quad (2.76)$$

Equations (2.76) are used for modelling the behaviour of a centrifugal pump. From figure 2.4 it is clear that at low ϕ -values the head of the pump given by ψ'' is noticeably overes-

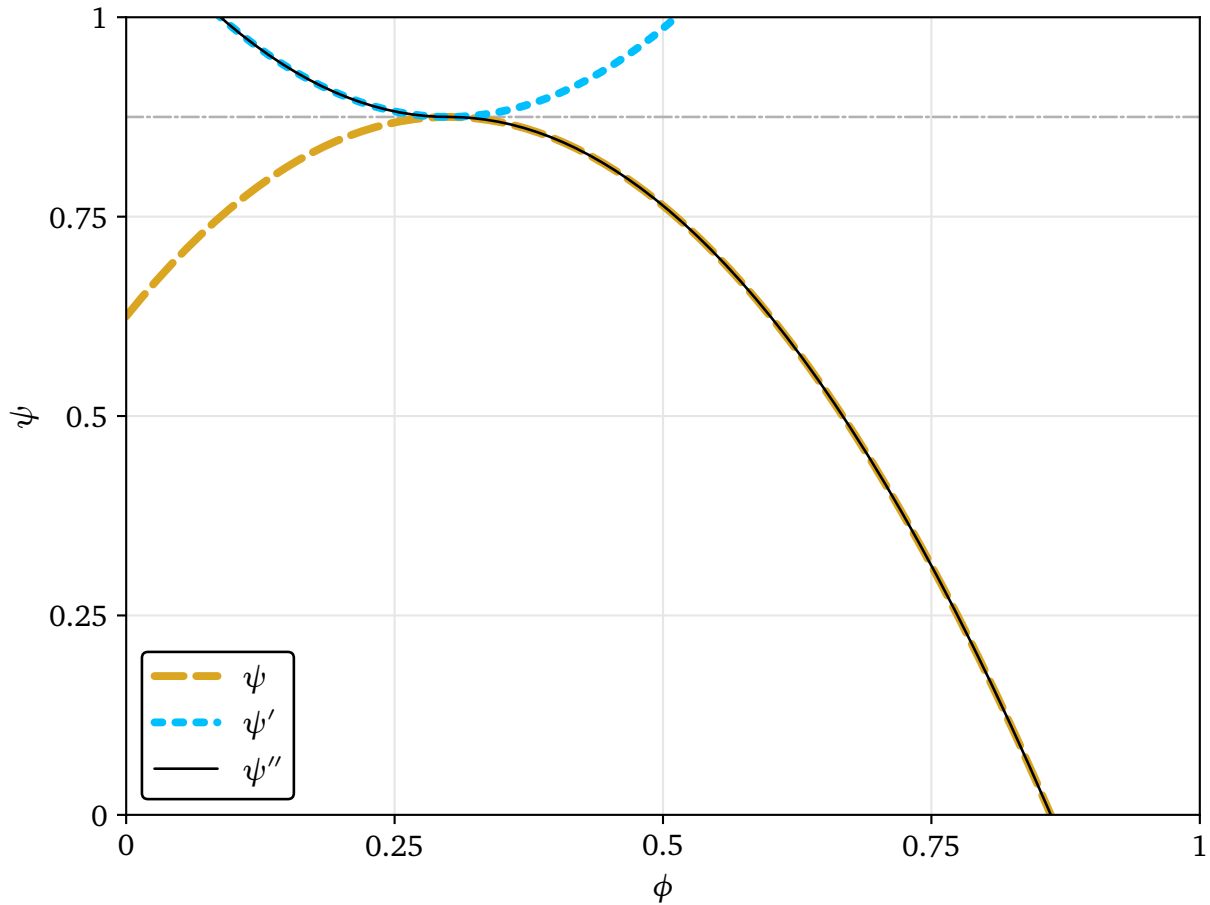


Figure 2.4. An example of the non-dimensional curves ψ , ψ' , and ψ'' used for modelling centrifugal pumps and fans.

timated. However, if the pump under examination is well-matched to its hydraulic circuit, it will operate at high flow rate coefficients ($\phi > \phi_v$), meaning that the final working point does not change due to the modifications performed on the characteristic curve.

The head given by a pump or fan is now expressed by means of the generalised formulation for source and sink terms, which is given in Equation (2.20):

$$\Delta p_{p,j}^{k+1} \approx \sigma_{p,j}^{k+1} = m_{p,j} u_j^{k+1} + k_{p,j}. \quad (2.77)$$

Equation (2.77) allows for the linearisation of Equation (2.71):

$$\psi_j'' \approx \xi_j \phi_j^k + \gamma_j, \quad (2.78)$$

where ϕ_j^k is the flow rate coefficient calculated at the k -th iteration. ξ_j is defined as

$$\xi_j = \begin{cases} -c'_{1j} - c'_{2j} \phi_j^{k-1} & \text{for } \phi_j^{k-1} < \phi_{vj} \\ c'_{1j} + c'_{2j} \phi_j^{k-1} & \text{for } \phi_j^{k-1} \geq \phi_{vj} \end{cases}, \quad (2.79)$$

while γ is:

$$\gamma_j = \begin{cases} 2\psi_j(\phi_{vj}) - c'_{0j} & \text{for } \phi_j^{k-1} < \phi_{vj} \\ c'_{0j} & \text{for } \phi_j^{k-1} \geq \phi_{vj} \end{cases}, \quad (2.80)$$

where ϕ_j^{k-1} is related to the previous iteration.

Finally, the dimensional linearised pump head is obtained from Equation (2.78) by multiplying all members by $\rho_j (rps_j D_{refj})^2$:

$$\Delta p_{pj}^k \approx \sigma_{pj}^k = \frac{\rho_j rps_j S_j}{D_{refj}} \xi_j u_j^k + \rho_j (rps_j D_{refj})^2 \gamma_j. \quad (2.81)$$

The coefficients m_{pj} and k_{pj} are:

$$m_{pj} = \frac{\rho_j rps_j S_j}{D_{refj}} \xi_j, \quad (2.82)$$

$$k_{pj} = \rho_j (rps_j D_{refj})^2 \gamma_j. \quad (2.83)$$

2.5.3.2. Viscous losses

The macroscopic mechanical energy balance, expressed by Equation (2.25), applied to a duct with uniform cross-section, in which the flow is steady and fully developed, and there are no sources of mechanical energy given by pumps or fans, reads:

$$\begin{aligned} \int_S \rho \left(\frac{v^2}{2} + gz \right) (\vec{v} - \vec{v}_s) \cdot \vec{n} dS + \int_S p \vec{v} \cdot \vec{n} dS = \\ = \int_V p \nabla \cdot \vec{v} dV - \int_V \tau : \nabla \vec{v} dV. \end{aligned} \quad (2.84)$$

Fluid density is considered constant therefore, the mechanical energy advective term and compression power cancel out:

$$\int_S p \vec{v} \cdot \vec{n} dS = - \int_V \tau : \nabla \vec{v} dV. \quad (2.85)$$

The integral on the left-hand side of Equation (2.85) is solved with the midpoint method (see Appendix A.2). As a result, the integral average of fluid velocity is substituted with its midpoint value, thus introducing a second order error. Pressure is considered uniform across vents. The term representing viscous losses is replaced with $-\dot{E}_v$:

$$\dot{E}_v = (p_I - p_E) \bar{u} S. \quad (2.86)$$

Equation (2.86) allows for the expression of viscous losses in terms of the pressure difference between the vents of the duct. Conceptually, viscous losses are divided in distributed and concentrated losses [151].

Distributed losses are associated to straight channels and pipes, they are expressed as a function of the Darcy friction factor, which is defined as:

$$f \equiv \frac{\Delta p}{L} \frac{D_h}{\frac{1}{2} \rho \bar{u}^2} . \quad (2.87)$$

By introducing Equation (2.87) in Equation (2.86) the mechanical power loss due to viscous effects is obtained:

$$\dot{E}_v = f \frac{L}{D_h} \frac{1}{2} \rho \bar{u}^3 S , \quad (2.88)$$

while the pressure drop is:

$$\Delta p_d = \frac{\dot{E}_v}{\bar{u} S} = f \frac{L}{D_h} \rho \frac{\bar{u}^2}{2} . \quad (2.89)$$

In general the friction factor is a function of surface roughness and flow regime, which is expressed by the Reynolds number:

$$\text{Re} = \frac{u D_h}{\nu} \quad (2.90)$$

In laminar flow, for $\text{Re} < 2000$, the Darcy friction factor f is independent of the surface roughness, since losses are mainly caused by fluid layers sliding with respect to each other:

$$f = \frac{64}{\text{Re}} . \quad (2.91)$$

Conversely, when the flow is turbulent, surface roughness plays an important role in the determination of f . If the micro irregularities of the surface are contained within the viscous sub-layer of the turbulent boundary layer, the surface of the duct or pipe is smooth from the hydraulic point of view. If the height of the peaks of surface roughness is higher than the one of the viscous sub-layer, friction losses increase due to pressure drag caused by local separation effects. At low Re-values the viscous sub-layer is tall enough to cover all peaks. As Re increases the viscous layer becomes thinner, thus exposing tall surface peaks to the turbulent flow. For this reason, at high Re-value the friction factor depends only on surface roughness characteristics. One of the most used correlation of the evaluation of f for rough pipes in turbulent flows is the Colebrook-White equation:

$$\frac{1}{\sqrt{f}} = -2.0 \log \left(\frac{Ra}{3.7 D_h} + \frac{2.51}{\text{Re} \sqrt{f}} \right) , \quad (2.92)$$

while the Blasius formula is employed for smooth pipes and ducts:

$$f = \frac{0.316}{\text{Re}^{1/4}} . \quad (2.93)$$

When examining components with complex geometries, which can not be directly traced back to sections of straight pipes, it is more convenient to express viscous losses by means of a concentrated loss coefficient β :

$$\Delta p_c = \beta \frac{1}{2} \rho \bar{u}^2. \quad (2.94)$$

β is usually determined by means of numerical analyses or experimental tests.

The combined distributed and concentrated losses associate with a given branch j are expressed as follows:

$$\Delta p_{L,j} = \left(f_j \frac{L_j}{D_{h,j}} + \beta_j \right) \frac{\rho_j \bar{u}_j |\bar{u}_j|}{2}, \quad (2.95)$$

where the square of the value of fluid velocity is substituted by the product $\bar{u}_j |\bar{u}_j|$, so that $\Delta p_{L,j}$ always determines a decrease of fluid pressure along the direction of fluid velocity. Equation (2.95) is a quadratic function of fluid velocity, therefore it is linearised through the generalised formulation for source and sink terms:

$$\Delta p_{L,j} \approx \sigma_{L,j}^{k+1} = m_{L,j} \bar{u}_j^{k+1} + k_{L,j} \quad (2.96)$$

where the $m_{L,j}$ and $k_{L,j}$ coefficients are obtained by comparing Equations (2.95) and (2.96):

$$m_{L,j} = \left(f_j \frac{L_j}{D_{h,j}} + \beta_j \right) \frac{\rho_j |u_j^k|}{2}, \quad k_{L,j} = 0. \quad (2.97)$$

2.5.3.3. Branch thermal properties

Heat transferred through the surface of a branch CV is given by:

$$\dot{Q}_j = \int_S -\vec{q} \cdot \vec{n} dS + \int_V H dV, \quad (2.98)$$

where $\vec{q}$ is the heat transferred through the boundaries of the control volume, while H indicates a volumetric heat source, which can be used to model a heating or cooling coil. The former of the two aforementioned contributions can be expressed using Newton's law of cooling, by assuming a uniform heat transfer coefficient and solving the associated integral using the midpoint method:

$$\dot{Q}_j = \int_S -\vec{q} \cdot \vec{n} dS = U_j S_{\text{ref},j} (T_\infty - \bar{T}_j), \quad (2.99)$$

where U_j is the global heat transfer coefficient and T_∞ is the reference temperature of the external environment. In general the $U_j S_{\text{ref},j}$ product can be expressed as function of the

equivalent thermal resistance involved with the examined heat transfer mechanisms:

$$\frac{1}{U_j S_{\text{ref},j}} = R_{\text{eq},j} . \quad (2.100)$$

For instance, when considering a round pipe exchanging heat with the environment by internal convection, wall conduction, and external convection and radiation, $R_{\text{eq},j}$ becomes:

$$R_{\text{eq},j} = R_{\text{int}} + R_w + \frac{R_{\text{ext}} R_{\text{rad}}}{R_{\text{ext}} + R_{\text{rad}}} . \quad (2.101)$$

The internal convective resistance R_{int} is given by:

$$R_{\text{int}} = \frac{1}{h_{\text{int}} \left(\frac{\Delta T_{\text{lm}}}{\Delta T} \right) S_{\text{int}}} \quad (2.102)$$

where the internal convective heat transfer coefficient h_{int} is derived from the Dittus-Boelter correlation for turbulent flow:

$$h_{\text{int}} = \frac{\lambda}{D_h} 0.023 \text{Re}^{0.8} \text{Pr}^{0.4} . \quad (2.103)$$

The conductive resistance associated with the walls of the pipe is:

$$R_w = \frac{\ln(D_{\text{ext}}/D_h)}{2 \pi \lambda_w L} \quad (2.104)$$

where D_e is the tube external diameter and λ_w is the thermal conductivity of the wall. The external convective thermal resistance R_{ext} is:

$$R_{\text{ext}} = \frac{1}{h_{\text{ext}} S_{\text{ext}}} , \quad (2.105)$$

where h_{ext} can be imposed as a boundary condition. The walls of the branch are assumed as grey bodies with emissivity ε_r , and the external surface of the branch is considered much smaller than the one of the environment. Under these assumptions the amount of heat transferred by radiation is given by:

$$\dot{Q}_{\text{rad}} = S_{\text{ext}} \sigma_B \varepsilon_r (T_{\text{ext}}^4 - T_{\infty}^4) , \quad (2.106)$$

where σ_B is the Stefan-Boltzmann constant, and T_{ext} is the external wall temperature of the branch. Equation (2.106) can be rewritten as:

$$\begin{aligned} \dot{Q}_{\text{rad}} &= S_{\text{ext}} \sigma_B \varepsilon_r (T_{\text{ext}}^2 + T_{\infty}^2)(T_{\text{ext}} + T_{\infty})(T_{\text{ext}} - T_{\infty}) \\ &= h_{\text{rad}} S_{\text{ext}} (T_{\text{ext}} - T_{\infty}) . \end{aligned} \quad (2.107)$$

This formulation leads to the definition of the radiation thermal resistance:

$$R_{\text{rad}} = \frac{(T_{\text{ext}} - T_{\infty})}{\dot{Q}_{\text{rad}}} = \frac{1}{h_{\text{rad}} S_{\text{ext}}} \quad (2.108)$$

where h_{rad} is given by:

$$h_{\text{rad}} = \sigma_{\text{B}} \varepsilon_{\text{r}} (T_{\text{ext}}^2 + T_{\infty}^2) (T_{\text{ext}} + T_{\infty}) . \quad (2.109)$$

In general, any heat transfer process associated with the walls of a branch is expressed by means of the generalised formulation for source and sink terms, defined in Equation (2.20):

$$\dot{Q}_j^{k+1} \approx \sigma_{\text{e},j}^{k+1} = m_{\text{e},j} T_j^{k+1} + k_{\text{e},j} , \quad (2.110)$$

where the superscript $k + 1$ indicates values referred to the $k + 1$ iteration of the solution algorithm. The coefficients $m_{\text{e},j}$ and $k_{\text{e},j}$ are determined as in Section 2.4:

$$m_{\text{e},j} = -\frac{1}{R_{\text{eq},j}} , \quad k_{\text{e},j} = \frac{T_{\infty}}{R_{\text{eq},j}} . \quad (2.111)$$

These coefficients are computed considering values of fluid properties and fields associated with the k iteration of the solution process.

2.5.3.4. Heat exchangers

Heat exchangers can be easily modelled by using the ε -NTU method:

$$\dot{Q}_j = \int_S -\vec{q} \cdot \vec{n} dS = \varepsilon C_{r,\min} (T_{\text{ext},\text{in}} - \bar{T}_j) = -\varepsilon C_{r,\min} \text{ITD} , \quad (2.112)$$

where $T_{\text{ext},\text{in}}$ is the inlet temperature of the external fluid. In this case, the upwind scheme used to obtain branch temperatures is particularly convenient. The temperature of the branch T_j is equal to the temperature value defined at the upwind node, which represents the inlet temperature of the internal fluid of the heat exchanger, used for the derivation of ITD. Once more $\dot{Q}_j$ is expressed by means of Equation (2.20), leading to the definition of the following source coefficients:

$$m_{e,j} = -\varepsilon C_{r,\min} , \quad k_{e,j} = \varepsilon C_{r,\min} T_{\text{ext},\text{in}} . \quad (2.113)$$

2.5.3.5. Heating and cooling elements

Heat uniformly generated or absorbed in the volume of fluid associated with a branch is modelled by a lumped heating or cooling element as shown in Figure 2.3a:

$$\dot{Q}_j = \int_V H dV = \dot{Q}_{g,j} , \quad (2.114)$$

which immediately leads to the definition of the coefficient $k_{e,j}$:

$$k_{e,j} = \dot{Q}_{g,j} . \quad (2.115)$$

2.5.4. Solution algorithm

The numerical solution to the flow of fluid within the network is attained by solving the set of algebraic equations obtained through the discretisation procedure:

$$\begin{cases} \mathbf{A} \mathbf{M} \mathbf{u}^{n+1} = -\mathbf{d} \\ \mathbf{A}^T \mathbf{P}^{n+1} = \mathbf{Y} \mathbf{u}^{n+1} + \mathbf{k} \\ \mathbf{W} \mathbf{T}^{n+1} = \mathbf{s} \end{cases} \quad (2.116)$$

The mass and mechanical energy macroscopic balances are decoupled by using an iterative procedure based on the SIMPLE algorithm [143]. In addition, at each iteration of this procedure, the temperature field is computed using Equation (2.64), and then fluid properties are updated as a function of pressure and temperature. The SIMPLE algorithm is an iterative procedure based on the pressure correction method [152], whose main steps can be outlined as follows:

- i. Velocity and pressure are divided in guessed φ^* and correction φ' fields:

$$\mathbf{u} = \mathbf{u}^* + \mathbf{u}' , \quad (2.117)$$

$$\mathbf{P} = \mathbf{P}^* + \mathbf{P}' . \quad (2.118)$$

- ii. An initial value is assigned to the guessed pressure field $\mathbf{P}^*$.
- iii. The guessed velocity field $\mathbf{u}^*$ is derived from the momentum or mechanical energy equation using the guessed pressure values $\mathbf{P}^*$.
- iv. Most likely the values of $\mathbf{u}^*$ do not satisfy continuity, as they were obtained from a guessed pressure field. Thus mass conservation is used to derive a pressure correction $\mathbf{P}'$, that once added to $\mathbf{P}^*$ will improve continuity.
- v. The procedure is repeated starting from step 3 until a satisfactory degree of convergence is reached.

2.5.4.1. Derivation of the pressure correction equation

An important step of the development and implementation of any iterative procedure based on the pressure correction method is the derivation of the pressure correction equation. The guessed velocity field is obtained by introducing the guessed pressure $\mathbf{P}^*$ in Equation (2.50):

$$\mathbf{u}^* = \mathbf{Y}^{-1} (\mathbf{A}^T \mathbf{P}^* - \mathbf{k}) . \quad (2.119)$$

A relation between the correction fields is retrieved by introducing Equations (2.117) and (2.118) into Equation (2.50):

$$\mathbf{A}^T (\mathbf{P}^* + \mathbf{P}') = \mathbf{Y} (\mathbf{u}^* + \mathbf{u}') + \mathbf{k} , \quad (2.120)$$

by pre-multiplying all members by $\mathbf{Y}^{-1}$ and considering Equation (2.119) the previous relation becomes:

$$\mathbf{u}' = \mathbf{Y}^{-1} \mathbf{A}^T \mathbf{P}' \quad (2.121)$$

To obtain the pressure correction equation, Equation (2.117) is introduced into the continuity Equation (2.34):

$$\mathbf{A} \mathbf{M} (\mathbf{u}^* + \mathbf{u}') = -\mathbf{d} , \quad (2.122)$$

then Equation (2.121) is used to obtain the final system of equations:

$$\mathbf{\Lambda} \mathbf{P}' = \mathbf{b} , \quad (2.123)$$

where $\mathbf{\Lambda}$ is an $n_n \times n_n$ matrix given by:

$$\mathbf{\Lambda} = \mathbf{A} \mathbf{M} \mathbf{Y}^{-1} \mathbf{A}^T , \quad (2.124)$$

and $\mathbf{b}$ is the vector of elements:

$$\mathbf{b} = -\mathbf{A} \mathbf{M} \mathbf{u}^* - \mathbf{d} . \quad (2.125)$$

2.5.4.2. Boundary conditions for the pressure field

At the boundary nodes of the network, fluid pressure is set to a fixed or time-varying value that is assigned to the guessed pressure field at the beginning of each iteration. By recalling Equation (2.118), an equivalent boundary condition for the pressure correction field is obtained:

$$\mathbf{P}_{\text{bou}} = \mathbf{P}_{\text{bou}}^* + \mathbf{P}'_{\text{bou}} \quad \Rightarrow \quad \mathbf{P}'_{\text{bou}} = \mathbf{0} . \quad (2.126)$$

Any boundary node of the network is connected to a single branch, meaning that in its corresponding row of the incidence matrix a single non-zero value is present, as in Equation (2.4). Therefore, boundary nodes can be identified by defining the $n_n \times n_n$ diagonal matrix $\mathbf{I}_{\text{bou}}$:

$$I_{\text{bou},ii} = 1 \quad \text{if} \quad \sum_j^{n_b} |A_{ij}| = 1 . \quad (2.127)$$

The matrix that identifies internal nodes $\mathbf{I}_{\text{int}}$ is simply obtained by considering the logical not of $\mathbf{I}_{\text{bou}}$. Pressure boundary conditions are applied to the pressure correction Equation (2.123), by modifying the matrix $\mathbf{\Lambda}$ and vector $\mathbf{b}$ with the aim of obtaining equations in the form of Equation (2.126) for each boundary node. The matrix of coefficients becomes:

$$\mathbf{\Lambda}_{\text{bc}} = \mathbf{I}_{\text{int}} \mathbf{\Lambda} + \mathbf{I}_{\text{bou}} , \quad (2.128)$$

as a result of this operation, the elements of each row of $\mathbf{\Lambda}$ related to a boundary node are zeroed ($\mathbf{I}_{\text{int}} \mathbf{\Lambda}$), then in the same rows a 1 is placed in the position corresponding to the diagonal ($+\mathbf{I}_{\text{bou}}$). The elements of vector $\mathbf{b}$ associated with boundary nodes are simply zeroed to obtain the desired boundary condition, according to Equation (2.126).

$$\mathbf{b}_{\text{bc}} = \mathbf{I}_{\text{int}} \mathbf{b} \quad (2.129)$$

The final set of equations used to obtain the pressure correction field is:

$$\mathbf{\Lambda}_{\text{bc}} \mathbf{P}' = \mathbf{b}_{\text{bc}} . \quad (2.130)$$

2.5.4.3. Boundary conditions for the temperature field

At inlet boundary nodes of the network the value of fluid temperature value is imposed, while at outlet ones a zero-gradient boundary condition is set. Boundary nodes are divided in inlet and outlet points on the basis of the sign of fluid velocity of their respective branches. These conditions are enforced by modifying Equations (2.64) to obtain the following equations for boundary nodes:

$$\begin{cases} \bar{T}_i = T_{i,\text{bc}} & \text{if node } i \text{ is an inlet point} \\ \bar{T}_i = \sum_j^{n_b} |A_{ij}| T_j & \text{if node } i \text{ is an outlet point} \end{cases} . \quad (2.131)$$

The matrix of coefficient $\mathbf{W}$ in Equation (2.64) becomes:

$$\mathbf{W}_{bc} = \mathbf{I}_{int} \mathbf{W} + \mathbf{I}_{bou} \quad (2.132)$$

The elements of the vector $\mathbf{s}$ for boundary and internal nodes are, respectively:

$$s_{bc,i} = \begin{cases} \frac{1}{2} \sum_j^{n_b} \left[|A_{ij}| (T_{i,bc} + \bar{T}_j) + \frac{\bar{u}_j}{|\bar{u}_j|} (T_{i,bc} - \bar{T}_j) \right] \\ s_i \end{cases} . \quad (2.133)$$

The final system of equations used to solve for the temperature field is:

$$\mathbf{W}_{bc} \mathbf{T} = \mathbf{s}_{bc} . \quad (2.134)$$

2.5.4.4. The steady-state flow solver

The algorithm used for steady-state problems involves the following steps:

- i. Assignment of boundary conditions and initial values: at the beginning of the procedure, boundary conditions are assigned to each boundary node, and initial values are assigned to each field, including the one for fluid properties.

$$\varphi_r = \varphi_{init} , \quad \varphi_r = \varphi_{bc,r} \quad \forall \varphi, r \quad (2.135)$$

In this instance, the choice of initial values does not affect the final solution, given the examined problem is steady. However, it may determine how many iterations are needed for the solution to converge.

- ii. Computation of source and sink terms: the m and k coefficients associated with sources and sink of mechanical and thermal energy, related to nodes and branches, are computed according to the specified models and as function of the respective values of fluid properties and fields.
- iii. Derivation of the guessed velocity field $\mathbf{u}^*$: A new guessed velocity field is obtained from the current guessed pressure field:

$$\mathbf{u}_{new}^* = \mathbf{Y}^{-1} (\mathbf{A}^T \mathbf{P}^* - \mathbf{k}) . \quad (2.136)$$

Then, the guessed velocity field is updated by introducing an under relaxation factor α , to dampen velocity oscillations across iterations:

$$\mathbf{u}^{*(k+1)} = \alpha \mathbf{u}_{new}^* + (1 - \alpha) \mathbf{u}^{*(k)} . \quad (2.137)$$

- iv. Pressure correction: the pressure correction field is obtained from the solution of Equation (2.130), and pressure is corrected by introducing an under relaxation factor:

$$\mathbf{P}' = \Lambda_{bc}^{-1} \mathbf{b}_{bc}, \quad \mathbf{P}^* = \mathbf{P}^* + \alpha \mathbf{P}'. \quad (2.138)$$

- v. Velocity correction: the velocity correlation field is obtained from the solution of Equation (2.121) and the velocity field is corrected accordingly:

$$\mathbf{u}' = \mathbf{Y}^{-1} \mathbf{A}^T \mathbf{P}', \quad \mathbf{u}^* = \mathbf{u}^* + \mathbf{u}'. \quad (2.139)$$

- vi. Computation of the error: the error associated with each iteration is computed as the maximum value of the pressure correction field, which is taken in absolute value:

$$e = \max |\mathbf{P}'|. \quad (2.140)$$

- vii. Computation of the temperature field: new temperature values are obtained by solving Equation (2.134):

$$\mathbf{T}_{new} = \mathbf{W}_{bc}^{-1} \mathbf{s}_{bc}. \quad (2.141)$$

- viii. Update of fluid properties: all fields associated with fluid properties are updated by computing new field values using the selected state equations or by interpolating tabled data as a function of pressure and temperature. Then these fields are updated using the familiar under relaxation technique:

$$\varphi_{new,i} = f(P_i^*, T_i), \quad \varphi^{k+1} = \alpha \varphi_{new} + (1 - \alpha) \varphi^k \quad \forall \varphi, i. \quad (2.142)$$

- ix. Error check: if the error associated with the current iteration e is lower than a pre-determined threshold value the calculation stops. Otherwise, the process is repeated starting from step (ii), until the aforementioned condition is satisfied.

Figure 2.5 reports the flow chart representing the steady-state flow solver including all the aforementioned steps.

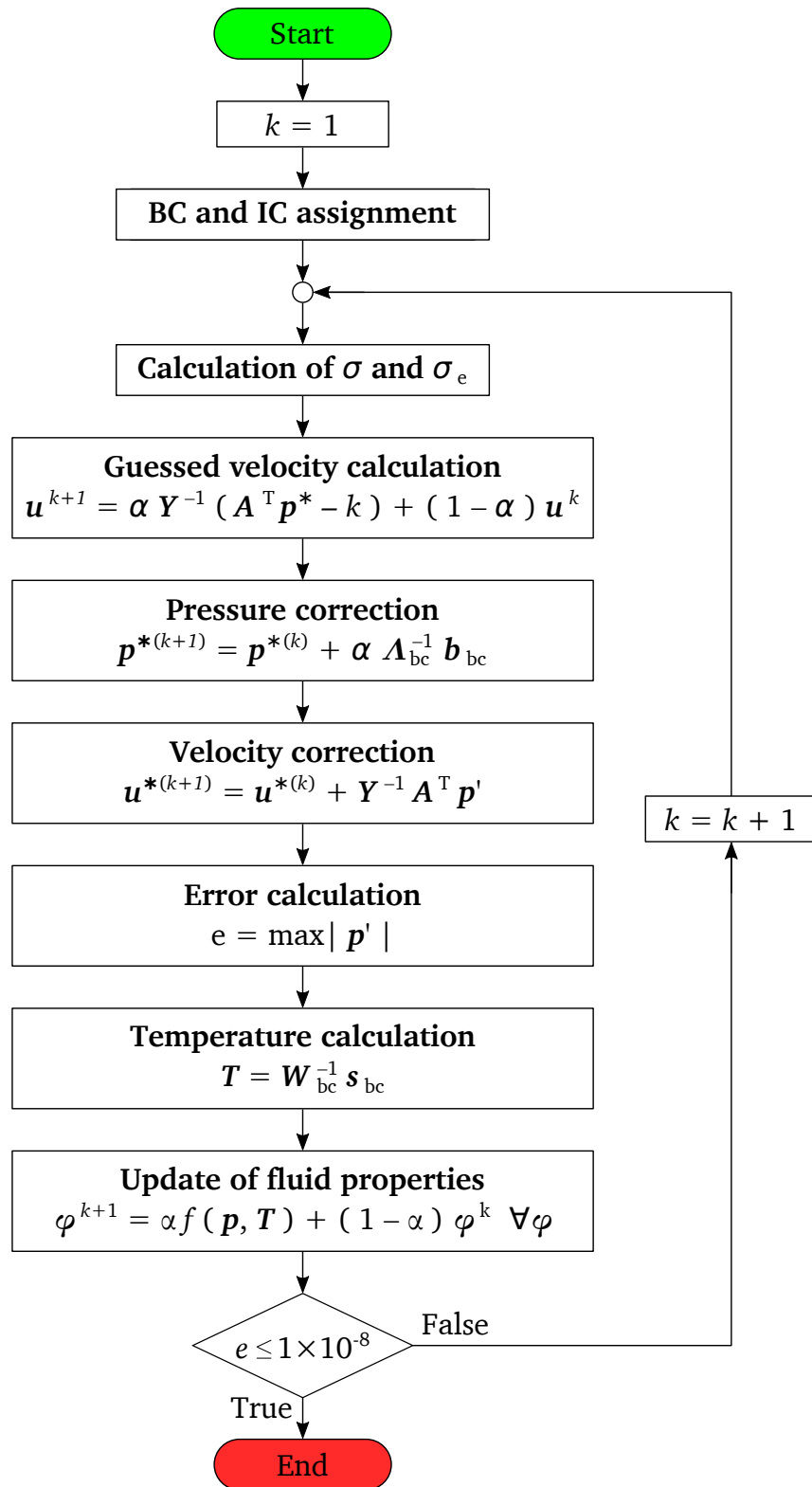


Figure 2.5. Flow chart representing the algorithm employing for the solution of steady-state flow problems, where k indicates the number of iterations.

2.5.4.5. The unsteady solver

Each time step of an unsteady case is solved using a similar procedure to the one used for steady-state problems, considering time-varying source terms and boundary conditions, along with unsteady terms in the governing equations.

- i. Assignment of initial values: these are assigned at the beginning of the process, following a procedure identical to the one described for the steady-state solver.
- ii. Assignment of boundary conditions: in this case, boundary conditions can be described by time-dependant functions, therefore they must be updated and enforced before the solution of each time step.
- iii. Solution of the time step $n+1$: the numerical solution is attained by an iterative procedure analogous to the algorithm used for steady-state cases, which includes governing equations with unsteady terms.
- iv. Time advancement:

$$t^{n+1} = t^n + \Delta t . \quad (2.143)$$

After this operation, the process starts again from step (ii) to solve the next time step. This is done until all time steps are calculated.

These steps are represented in the flow chart of Figure 2.6, where the inner loop used for the solution of each time step is contained by a dashed box.

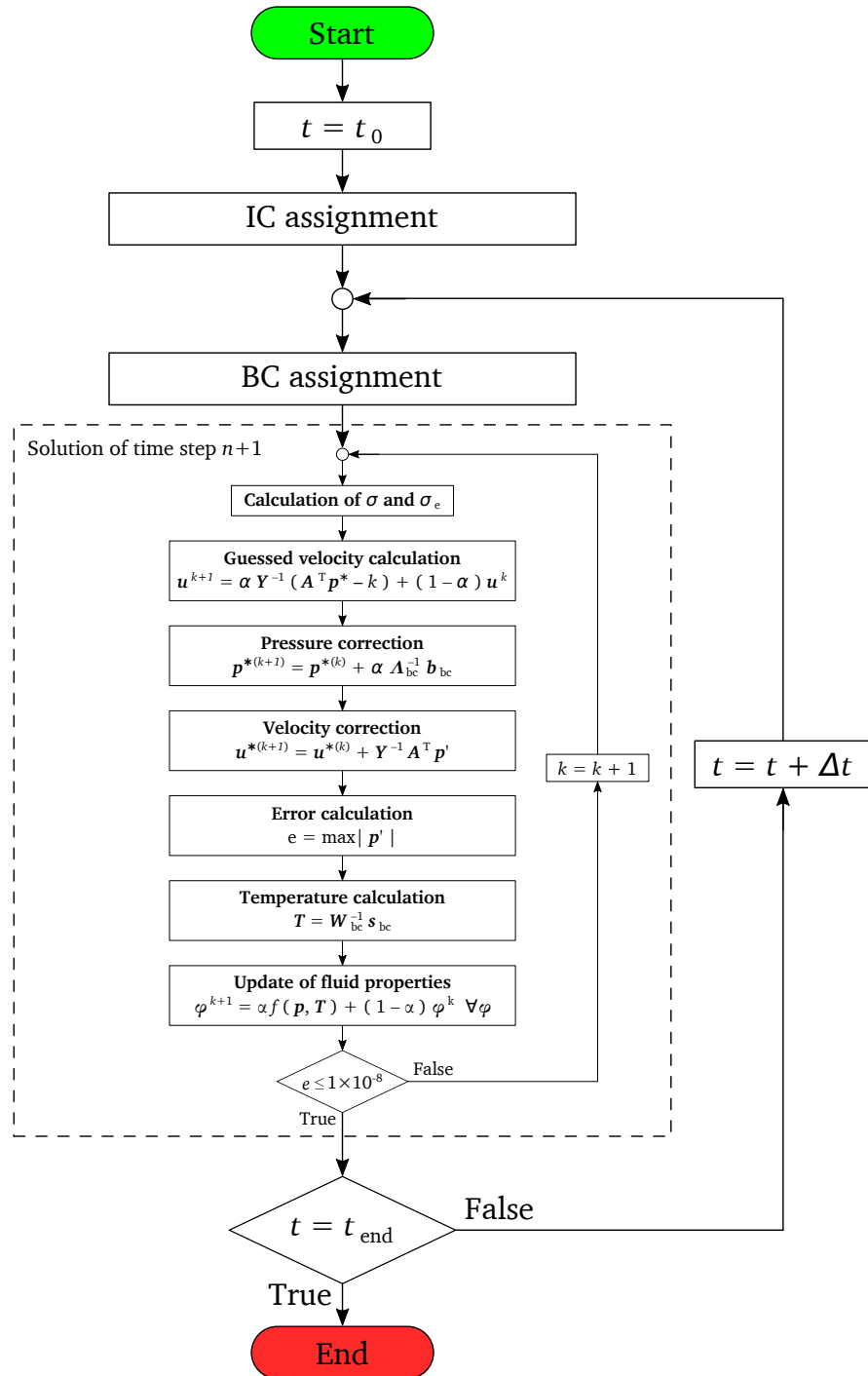


Figure 2.6. Flow chart describing the main steps involved in the iterative procedure used for the solution of transient one-dimensional flow problems.

2.6. Implementation of a thermal network solver

Thermal network models are widely employed for the thermal analysis of complex systems, which can be accurately represented using a lumped parameter approach, such as electric motors [153], semiconductor packages [154], battery packs [155], and many others. LPTNs are developed by subdividing the physical domain under examination in a set of regions, depending on the desired levels of accuracy and spatial resolution of the model. Each of these regions is identified by a single temperature node, placed at the centre of the subdomain, representing its average temperature. The temperature nodes are connected by thermal resistances that model heat transfer processes along the main heat paths between nodes. In the following, the graph-based domain discretisation approach presented in Section 2.2 and the generalised formulation for source terms defined in Section 2.4 are employed to develop a numerical algorithm for the analysis of networks of thermal resistances.

2.6.1. Governing equations

In thermal networks, the temperature field is obtained by solving the heat equation at each node i :

$$C_i \frac{dT_i}{dt} + \sum_r \frac{T_i - T_r}{R_{ir}} = \dot{Q}_i, \quad (2.144)$$

where the index r is used to iterate over the nodes connected to node i through the thermal resistances R_{ir} . C_i and $\dot{Q}_i$ are the thermal capacity and the heat source associated with node i , respectively. Thermal resistances are modelled by suitably defining a thermal energy source and applying it to branches. The heat flowing through a given branch j is modelled as a source term by employing the generalised formulation reported in Equation 2.20:

$$\dot{Q}_j = \frac{\Delta T_j}{R_j} = G_j \Delta T_j = \sigma_j = m_j \Delta T_j, \quad (2.145)$$

where the source coefficient m_j is used to model the particular physical phenomenon associated with R_j . The term G_j stands for the thermal conductance of branch j . The same approach is used to model heat generation in each node of the system:

$$\dot{Q}_i = \sigma_i = k_i. \quad (2.146)$$

In Equation (2.144) the sum over the r nodes connected to node i is replaced by the sum of the heat fluxes associated with the branches of the network:

$$C_i \frac{dT_i}{dt} + \sum_j^{n_b} A_{ud,ij} m_j \Delta T_j = k_i, \quad (2.147)$$

where $A_{ud,ij}$ are the elements of the $n_n \times n_b$ unidirectional incidence matrix $\mathbf{A}_{ud}$. The use of the terms A_{ij} introduces the following sign convention: heat fluxes entering node i assume a

positive sign, while exiting ones are assumed negative. The values of temperature difference related to branches are derived from nodal temperatures by introducing the transpose of $\mathbf{A}$:

$$C_i \frac{dT_i}{dt} + \sum_j^{n_b} A_{ud,ij} m_j \sum_l^{n_n} A_{lj} T_l = k_i. \quad (2.148)$$

Time integration of Equation (2.148) is performed using the implicit Euler method of first order accuracy (see Appendix A.3):

$$\frac{C_i}{\Delta t} T_i^{n+1} + \sum_j^{n_b} A_{ud,ij} m_j \sum_l^{n_n} A_{lj} T_l^{n+1} = k_i + \frac{C_i}{\Delta t} T_i^n, \quad (2.149)$$

where Δt is the time interval between the time steps t^n and t^{n+1} . By applying Equation (2.149) to all the nodes of the network the final system of equations is obtained:

$$\mathbf{L} \mathbf{T} = \mathbf{r}, \quad (2.150)$$

where the $n_n \times n_n$ system matrix $\mathbf{L}$ is given by:

$$\mathbf{L} = \mathbf{L}_u + \mathbf{A}_{ud} \mathbf{M}_s \mathbf{A}^T. \quad (2.151)$$

In Equation (2.151) $\mathbf{L}_u$ is the $n_n \times n_n$ diagonal matrix of elements:

$$L_{u,ii} = \frac{C_i}{\Delta t}, \quad (2.152)$$

while $\mathbf{M}_s$ is the $n_b \times n_b$ diagonal matrix of elements:

$$M_{s,jj} = m_j. \quad (2.153)$$

The vector of known terms $\mathbf{r}$ presents the elements:

$$r_i = k_i + \frac{C_i}{\Delta t} T_i^n. \quad (2.154)$$

2.6.2. Modelling schemes for thermal resistances

As previously anticipated, thermal resistances are modelled by specifying their equivalent thermal conductance values using the generalised formulation for source terms, as in Equation (2.145). Conductive resistances are expressed in terms of the conduction shape factor and thermal conductivity, leading to the following source term:

$$R_j = \frac{1}{S_{f,j} \lambda} \Rightarrow G_j = m_j = S_{f,j} \lambda . \quad (2.155)$$

Thermal resistances modelling combined phenomena of convective and radiation heat transfer are similarity defined:

$$R_j = \frac{1}{(U_c + U_r) A_{\text{ref}}} \Rightarrow G_j = m_j = (U_c + U_r) A_{\text{ref}} \quad (2.156)$$

where U_c and U_r are the convective and radiative heat transfer coefficients, respectively. These coefficients are obtained from user-defined functions of branch mean temperature and temperature difference.

2.6.3. Boundary conditions

Boundary conditions are applied by defining masks for internal and boundary nodes using a similar approach as the one described in Section 2.5.4.2. The masks for boundary and internal nodes are $n_n \times n_n$ diagonal matrices whose elements are defined as follows.

$$\begin{cases} I_{\text{bou},ii} = 1, I_{\text{int},ii} = 0 & \forall i \in \mathbb{N}_b \\ I_{\text{bou},ii} = 0, I_{\text{int},ii} = 1 & \forall i \notin \mathbb{N}_b \end{cases} \quad (2.157)$$

In Equation (2.157) $\mathbb{N}_b$ denotes the set of nodes where a fixed temperature boundary condition is applied. In this instance, the identification of boundary nodes is not done automatically as in the case of the one-dimensional flow solver developed in Section 2.5 for two main reasons. For one, fixed temperature boundary conditions may need to be applied to internal nodes depending on the topology of the studied system, and secondly, nodes that are of boundary from the topological point of view, i.e. those connected to a single branch, can be easily managed by tuning the value of the heat source $\dot{Q}_i$. Setting $\dot{Q}_i \neq 0$ allows for an entering/exiting heat flux at the boundary, while the condition $\dot{Q}_i = 0$ automatically translates into a zero-gradient condition applied at the boundary, according to Equation (2.144).

Fixed temperature boundary conditions are applied by zeroing each row of the matrix of coefficients corresponding to a boundary node and by adding a unitary coefficient at the position associated with boundary nodes:

$$\mathbf{L}_{bc} = \mathbf{I}_{int} \mathbf{L} + \mathbf{I}_{bou} . \quad (2.158)$$

The vector of known terms is also modified accordingly:

$$\mathbf{r}_{bc} = \mathbf{I}_{int} \mathbf{r} + \mathbf{I}_{bou} \mathbf{T}_{bc} , \quad (2.159)$$

where $\mathbf{T}_{bc}$ is the vector holding the boundary temperature values.

2.6.4. Solution algorithm

The governing equations are solved by means of an iterative procedure. At each time step the temperature field is solved iteratively while updating the values of temperature-dependant thermal resistances and heat sources until convergence is reached. Time advancement is handled by an external loop. Figure 2.7 reports the flow chart of the algorithm used for the solution of transient problems. Steady-state cases are solved by removing unsteady terms from the governing equations, and using an algorithm constituted by the inner loop highlighted in Figure 2.7 by the dashed box.

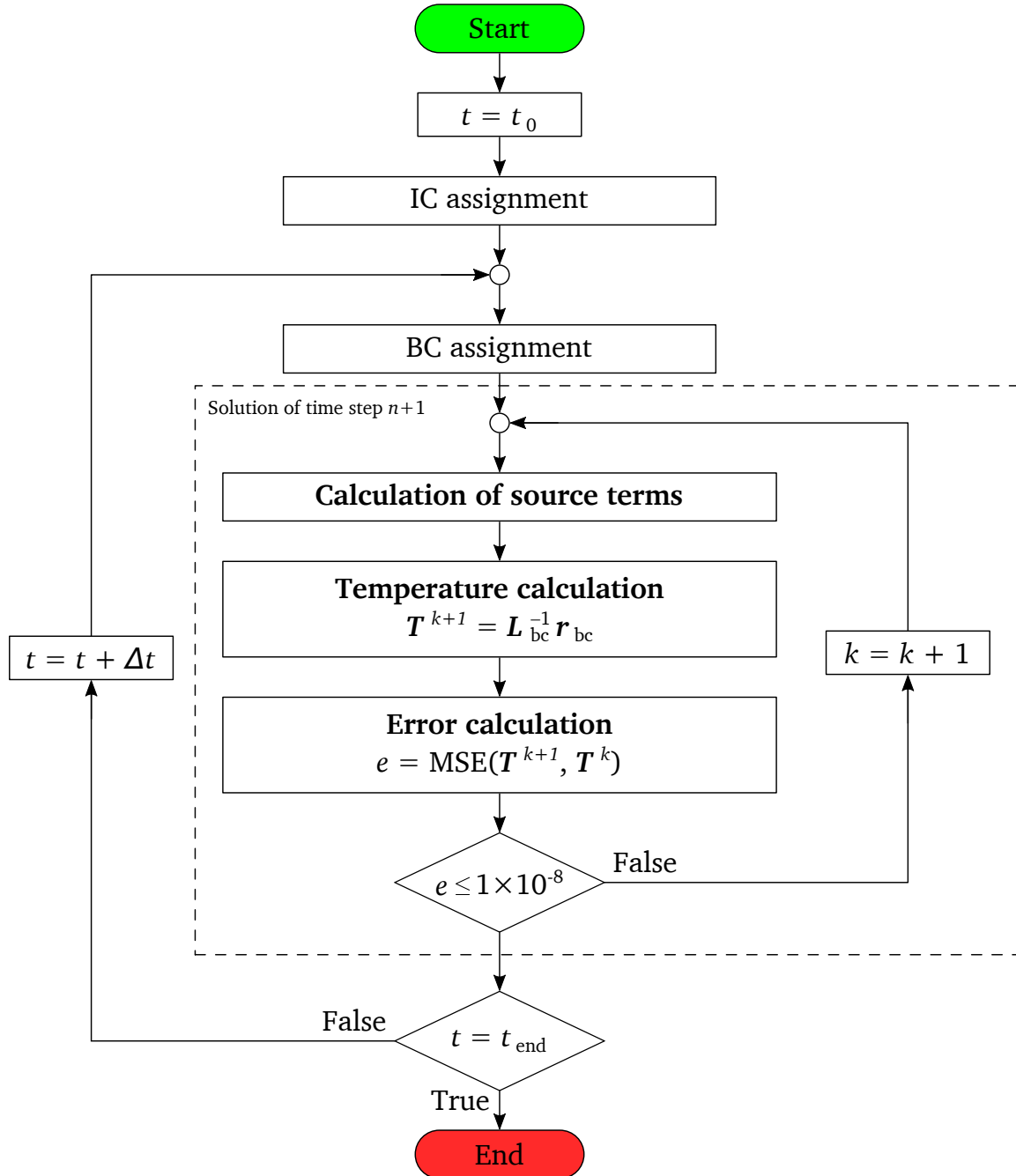


Figure 2.7. Flow chart reporting the main steps involved in the solution of transient Lumped Parameters Thermal Networks.

2.7. Concluding remarks

This Chapter presented the development of a computational toolbox for the development and analysis of reduced-order models of physical systems. At the core, this methodology was based on three main pillars: the use of the graph theory for the representation of the computational domain, an advanced management system for units of measurement to check the dimensional consistency of operations, and a generalised semi-implicit formulation for the source and sink terms used to model linear and non-linear physical phenomena.

These basic building blocks were used as a basis for the development of solvers for networks of pipes and thermal resistances. The first of these two solvers was based on a one-dimensional staggered Finite Volume method. The components typically present in pipe networks were modelled by suitably defined source and sink terms that were assigned to branches. Steady-state and transient flow solvers were developed on the basis of the simple algorithm.

The solver for networks of thermal resistances was obtained by modelling thermal resistances through branches with suitable thermal properties. The temperature dependence of thermal resistances introduced by varying material properties or temperature dependent heat transfer mechanisms was managed by iterative procedures.

Most of the developed tools are used extensively in Chapter 3 for the thermal analysis of an integrated motor drive for efficient and sustainable vehicles. Future research efforts specific to this computational toolbox will involve the development of multi-physics solvers and the compilation of part of the library to reduce the computational effort required for very large models.

CHAPTER 3

Thermal analysis of an integrated motor drive

3.1. Introduction

3.1.1. Background and aims

In the field of electric vehicle propulsion, a significant research and development effort is aimed at improving the degree of integration between electric motors and their power converters, as this allows for the reduction of the size and weight of the powertrains while improving their efficiency, thanks to the elimination of superfluous electric components. A critical aspect of the integration between motors and their inverters is thermal management, as the two systems share the same casing and cooling system, which must be designed to effectively remove heat from the motor and power electronics components, while ensuring that their respective thermal limits are never reached.

This chapter presents the thermal analysis of an integrated motor drive for efficient and sustainable vehicles composed by a six-phase synchronous motor using permanent magnets for the excitation and its power converter, which presents a switching cell array architecture and silicon carbide semiconductors. The developed thermal models are mainly employed to verify the feasibility of the proposed architecture from the thermal point of view. This study was conducted as part of the European project SCAPE, which is funded by the European Union's Horizon Europe research and innovation program under grant agreement No. 101056781. SCAPE aims to improve the standardisation and performance of power electronics components for next generation electric vehicles, while also reducing their cost. The views and opinions expressed in this work are those of the author only and do not necessarily reflect those of the European Union or CINEA. Neither the European Union nor the granting authority can be held responsible for them.

NOMENCLATURE*Latin letters*

A	area [m^2]
c_p	specific heat [$\text{J kg}^{-1} \text{K}^{-1}$]
D_h	hydraulic diameter [m]
f	frequency [Hz]
h	height [m]
I	current intensity [A]
K_{rk}	AC/DC loss ratio [—]
l	length [m]
$\dot{m}$	mass flow rate [kg s^{-1}]
Nu	Nusselt number [—]
p	fluid pressure [Pa]
Pr	Prandtl number [—]
q	heat flux [W m^{-2}]
$\dot{Q}$	heat [W]
r	radius [m]
R	thermal resistance [K W^{-1}]
R_{el}	electrical resistance [V A^{-1}]
Re	Reynolds number [—]
S	conduction shape factor [m]
t	thickness [m]
T	temperature [K]
Ta	Taylor number [—]
U	overall heat transfer coefficient [$\text{W m}^{-2} \text{K}^{-1}$]
$\vec{v}$	fluid velocity [m s^{-1}]
V	volume [m^3]
w	width [m]

Greek letters

γ	electrical resistivity [$\Omega \text{ m}$]
λ	thermal conductivity [$\text{W m}^{-1} \text{K}^{-1}$]
μ	dynamic viscosity [Pa s]
ν	kinematic viscosity [$\text{m}^2 \text{s}^{-1}$]
ρ	density [kg m^{-3}]
θ	dimensionless temperature difference [—]
φ	angle [rad]
ω	angular velocity [rad s^{-1}]

Subscripts

adv	advective
ag	air gap
b	bulk
bsl	baseline
c	conductor
cmp	component
conv	convective
cr	core
e	enamel
ext	external
fp	flat plate
i	inlet
int	internal
m	magnet
o	outlet
p	paper
rs	resin
s	stator
sh	short
sl	slot
ss	stator section
t	turbulent
ta	tall
w	winding

3.1.2. Literature survey

The thermal analysis of electric motors is often performed by means of Lumped Parameters Thermal Networks (LPTNs), thanks to their flexibility, ease of implementation, and low computational cost when compared with higher order methods such as Finite Element Analysis (FEA). In LPTNs the physical domain is divided in regions, each defined by a temperature node. These points are connected by thermal resistances that model the heat transfer mechanisms and paths between regions. This technique can be used to obtain simple models formed by a handful of nodes and resistances or more complex implementations based on large and intricate networks of thermal resistances.

The work by Kotnik [156] represents one of the first attempts at the thermal modelling of electric machines using lumped parameter models. The author developed a thermal network, by considering one-dimensional steady-state conduction through plane walls. Despite the simplicity of the model, numerical results were in good agreement with the available experimental data for National Electrical Manufacturers Association (NEMA) non-ventilated machines. Contemporary modelling schemes involving reduced-order thermal models are based on the pioneering work by Mellor, Roberts, and Turner [153], who proposed a generalised approach for the representation of the internals of electric motors using equivalent hollow cylindrical segments. Thermal resistances were computed by considering nominal geometric parameters and constant heat transfer coefficients, and the overall methodology was validated by considering experimental data on three air-cooled induction motors. Several authors built upon the base approach presented by Mellor *et al.* by proposing thermal models developed for electric motors of various architectures. Chowdhury *et al.* [157] developed a LPTN-based model for a single phase induction machine, implemented in a MATLAB environment. Kral *et al.* [158, 159] implemented an analogous model in the Modelica language to study an induction motor with a water-cooled housing. Aglén and Andersson [160] studied the thermal management of a high-speed synchronous generator, evaluating the effects on the temperature distribution given by a copper shield for the rotor. An induction motor with a copper rotor was also studied by Ahmed *et al.* [161]. El-Refaie *et al.* [162] developed LPTNs for two multi-barrier interior synchronous motors, validating their accuracy by comparing results from the reduced-order models with the outcomes of detailed Finite Element (FE) analyses. The good agreement between the two sets of numerical results highlighted the strengths of the LPTN models as accurate and fast predictive tools that can be very useful in evaluating initial machine designs. Similarly, Ponomarev *et al.* [163] compared LPTN, FE, and Computational Fluid Dynamics (CFD) approaches in terms of the resolution and fidelity of results and the required development and computational efforts. The LPTN approach was the best suited for obtaining fast estimates about hot-spot temperatures, while the CFD models offered more accurate results at a much higher computational cost.

Some authors have attempted to further simplify LPTN models by reducing the number of temperature nodes, in order to reduce development and implementation time, without sacrificing accuracy. Boglietti *et al.* [164] exploited radial and axial symmetry to reduce the number of nodes included in the network. The values of heat transfer coefficients were derived from parameter identification procedures based on experimental data, allowing for great overall accuracy at a reduced node count. Henneberger *et al.* [165] applied similar techniques on a water-cooled induction motor. Wallscheid *et al.* [166] developed a LPTN composed of just four temperature nodes to model the most critical components of a Permanent Magnet Synchronous Motor (PMSM): the stator yoke and teeth, the windings, and the magnets. Then, the authors applied a global identification technique, based on particle swarm optimisation, to obtain the parameters of the thermal network. Similarly, Sciascera *et al.* [167] developed a seven-node thermal network for a permanent magnet synchronous machine. The authors employed the SQP iterative method to tune the values of the most relevant resistances, to enhance the accuracy of the model.

While simplified thermal models may seem very attractive due to their straightforward implementation, their possible reliance upon parameter identification methods based on experimental data may severely limit their generality, and the extensive experimental campaigns involved may considerably drive up development costs.

Lumped parameter thermal networks are built upon a large set of parameters, namely geometric data, material properties, heat transfer coefficients, contact resistances, and many more. In most cases, geometric parameters are well-defined and associated with reasonably high accuracy levels, while the other main constitutive parameters are linked to far greater uncertainties, which could be mitigated through the aforementioned parameter identification procedures. Nevertheless, it is important to evaluate the effects of uncertainties on temperature results to make sure that the design considered is not too sensitive with respect to manufacturing tolerances, and the choice of constitutive materials. The effects of uncertainties are usually assessed by means of sensitivity analyses, where multiple numerical analyses are performed varying the value of one parameter each time, and evaluating the differences with respect to the baseline results. Boglietti *et al.* [61, 168] carried out a sensitivity analysis, to identify the most influential parameters on the accuracy of LPTNs modelling five induction motors. The authors performed experimental tests and numerical analyses on five induction motors to obtain sensitivity curves. In a natural convection setting (still rotor), the winding temperature was mostly sensitive to the housing convective heat transfer coefficient, the surface emissivity of the case, and the thermal conductivity of the impregnation resin. A minor role was played by the interface gap between the housing and stator laminations. In case of moving rotor, the winding temperature was mostly dependent on the value of air velocity in the channels formed by the case fins, which is usually difficult to determine due to local leakage and blockage effects. Likewise, the sensitivity analysis performed in [153] identified the convective resistance between the case

and the ambient as the most influential parameter on heat transfer performance. Similar results were obtained by Bennion and Cousineau [169] who analysed two permanent magnet synchronous motors. Assaad *et al.* [170] performed a sensitivity analysis by means of a Monte-Carlo approach on a PMSM. The most influential parameter was the heat transfer coefficient associated with the housing, while a minor contribution was given by the stator-casing interface gap. The authors also performed a sensitivity analysis to simplify a LPTN model by removing minor heat transfer paths, thus removing less relevant thermal resistances [171]. In [172] Cavazzuti *et al.* proposed an extensive sensitivity analysis including dimensional parameters, along with material properties and heat transfer coefficients.

Although these studies were conducted using different techniques and considering different motor architectures, they all come to similar conclusions: in most of the works, the two parameters that most influence winding temperatures are the heat transfer coefficient between the housing and the cooling fluid and the contact resistance between the housing and the stator. This is expected as these two resistances are connected directly in series with the thermal energy sink, furthermore, the conductive resistances in the stator represent the major contribution to the equivalent thermal resistance between the windings and the cooling system, as electrical steel [173] and insulating materials [174–176] are generally poor thermal conductors.

The accuracy and fidelity of numerical results provided by LPTNs can be improved by coupling reduced-order models with higher order methods as CFD and FEA. The former is usually employed to derive heat transfer coefficients in the end winding region and in the cooling jacket, while the latter is used to obtain accurate estimates about the losses generated in the windings, in the iron laminations, and in the magnets if present. Nategh *et al.* [177] developed a reduced-order model of an oil-cooled motor based on a set of angular segments as in [153]. Simple CFD analyses were performed to compute coolant flow rate within the housing channels, and on the end winding surfaces. The resulting velocity values were then used to derive heat transfer coefficients from empirical relationships. This approach allowed for an accurate prediction of the temperature distribution along the circumferential direction and in the end windings. Ahmed *et al.* [178] presented a combined LPTN and CFD approach to determine heat transfer coefficients in the end winding region of an induction motor with a smooth copper rotor. In [179] Guo *et al.* used a similar approach for the thermal analysis of an electric motor with a conical rotor, where CFD methods were used to determine heat transfer coefficients in the air gap. In [180] and [163] lumped parameter thermal networks coupled with CFD models were developed for electric motors directly cooled by dielectric oil.

As previously anticipated, FE approaches are mainly employed for an accurate evaluation of the losses. In [181] and [182] thermal networks were coupled with a Finite Element models of the magnetic field, allowing for an accurate calculation of the temperature dependant distributed losses. Fan *et al.* [183] applied this approach to a PMSM intended for

traction applications, considering a realistic driving duty cycle for the computation of the losses. Alberti and Bianchi [184] performed finite element analyses to compute the parameters of the equivalent electric circuit of an induction machine. The equivalent circuit was coupled with a thermal network model to perform coupled thermal and electromagnetic calculations, with a low computational effort.

Over the years, interesting variants of the original LPTN approach proposed by Mellor *et al.* [153] have been proposed. Dajaku and Gerling [185] proposed an alternative thermal network model for a permanent magnet machine, with the inclusion of compensation thermal elements. These network entities allow for the correct evaluation of the temperature rise in regions with distributed heat generation, without having to perform a fitting procedure. These concepts are discussed extensively in [186] and [187]. A comparison with numerical results obtained from a finite element analysis showed that the use of this improved modelling approach leads to a better temperature prediction in the windings and teeth.

The survey of the above-referenced works paints LPTNs as the best approach to study the thermal management of electric motors and generators. Almost all the methodologies considered in the literature review are variants of the original method proposed by Mellor *et al.* [153], while higher order methods are often deemed unfeasible to carry out complete studies on electric motors due to their high computational effort. These techniques are more often employed to carry out simple analyses with limited scope to improve the computation of the losses and constitutive coefficients of LPTN-based models.

3.1.3. Modelling approach

The modelling methodology developed in this Chapter for the thermal analysis of an integrated motor drive is based on lumped parameters thermal networks developed on the basis of nominal geometric parameters, tabled material properties, established empirical correlations, and numerical results obtained from CFD and FE models. Specifically, three LPTN models with varying complexity and number of degrees of freedom are developed, by using the computational toolbox developed in Chapter 2. The first and simplest of these consists in a two-dimensional thermal network including a periodic section of the electric machine, a single converter Printed Circuit Board (PCB), and a sixth of the water cooled case. The periodic domain modelling the electric motor is composed by a single stator slot along with its conductors and insulation, half of the two adjacent teeth, and the slice of the rotor associated with the considered angular span of the stator. The second thermal model is formed by expanding the initial 2D network along the axial direction by repeating the in-plane arrangement of nodes and resistances related to the motor, and connecting all interior nodes by suitably defined axial resistances. In this model the end windings are modelled by dedicated temperature nodes to account for the losses generated in these regions of the motor. This basic three-dimensional model is used to model the whole motor by mean of

the parallel rule, to allow for the estimation of the coolant temperature rise along the axial direction. The thermal resistances modelling the advection of heat include thermal diodes, which are purposely conceived to account for the direction of coolant flow within the cooling jacket. The third and final thermal model consists in an extensive 3D thermal network modelling the electric motor in its entirety, all the six converter legs, and the whole cooling jacket. The portion of the LPTN modelling the motor is obtained by repeating the arrangement of nodes and resistances forming the 2D model along the axial and circumferential directions, connecting interior nodes by circumferential and axial conductive resistances. The coolant temperature nodes, each associated with an axial portion of a slot of the motor, are connected by advective thermal resistances that follow the same path as coolant in the aluminium housing.

In all developed thermal networks the losses generated in the motor are retrieved from previous electromagnetic Finite Element analyses performed on the motor, considering a uniform temperature distribution. The effects of temperature on winding losses are introduced by using the scaling laws proposed by Wrobel, Griffo, and Mellor [188]. The influence of temperature on the losses in the iron laminations and magnets is not taken into consideration as minor deviations are expected in this case. Conduction and switching losses in the power switches forming the inverter are obtained from preliminary estimates. In addition, conduction losses are scaled with temperature considering a simple linear law.

The convective heat transfer coefficients associated with coolant flow in the casing are evaluated by developing a dedicated CFD model including a sixth of one of the cooling channels under periodic flow conditions. The momentum, continuity, and energy equations are solved by means of a validated conjugate heat transfer solver based on the Finite Volume method. Fluid and solid properties are assumed constant with respect to temperature and the pressure-velocity coupling is addressed by a variant of the SIMPLE algorithm. The heat transfer coefficients associated with the internal and external walls of the case are computed by performing multiple runs, considering different sets of boundary conditions for temperature, where an entering heat flux is set to the wall under investigation, while adiabatic conditions are assigned to all other wall boundaries. In all cases periodic boundary conditions are applied along the streamwise direction, while the continuity of temperature and heat flux is enforced at all fluid-solid interfaces.

The basic 2D LPTN model of the traction motor is validated by comparing temperature results with previous numerical outcomes from a Finite Element model of the same machine. Subsequently, a comprehensive sensitivity analysis is performed to identify the most critical parameters to the thermal performance of the integrated assembly, and to assess the effects of changes in manufacturing tolerances and constitutive materials on temperature results. Finally, numerical results from the three LPTNs are used to evaluate the feasibility of the proposed integrated architecture, and they are compared with each other to evaluate the effect of spatial resolution on the temperature results.

3.2. Case study

The integrated motor drive under consideration consists of a six-phase permanent magnet synchronous motor and the associated power converter. The stator of the electric machine is made out of NO20-1200H laminated electrical steel, its external and internal radii are $r_{s,ext} = 132.15$ mm and $r_{s,int} = 96.0$ mm, respectively. The motor presents an active length of $l_s = 140$ mm. The stator houses 96 rectangular slots, which have a width of $w_{sl} = 3.6$ mm and a height of $h_{sl} = 17.2$ mm. Each slot is lined with Kapton[®]-based insulating paper with a thickness of $t_p = 0.22$ mm.

At the base of the slots the paper is folded, forming a layer of insulation with a thickness of 0.44 mm. The copper windings present a hairpin arrangement, including six conductor layers with varying cross-section. This design allows for a greater ratio of the area of copper to the total area of the slot, compared to conventional windings. In addition, the use of conductors with a smaller cross-sectional area near the rotor allows for the mitigation of AC losses caused by proximity and skin effects. All the copper busbars have a rectangular cross-section with a width of $w_c = 2.8$ mm, while they present different height-values depending on their position: the four conductors closest to the rotor have a height of $h_{c,sh} = 1.8$ mm, while the two furthest away have a height of $h_{c,ta} = 4.0$ mm. The vertical spacing between two conductors is 0.245 mm. The busbars are enamelled with a layer of Voltatex[®] insulating compound that has a thickness of $t_e = 0.0725$ mm, and the space between them is filled with SbTCM resin. All the materials used for the insulation of the windings are H-class rated, meaning that the maximum working temperature of the windings is 180 °C.

The rotor features two types of neodymium-iron-boron magnets having the same width of $w_m = 11$ mm, and heights of $h_{m,ta} = 5.5$ mm and $h_{m,sh} = 4.5$ mm. The material constituting the magnets presents a Curie temperature of 310 °C. Rotor laminations are made out of NO20-1200H electrical steel. Figure 3.1 shows a cross-section of the motor and a detailed view of one slot highlighting the arrangement of the conductors and insulation. In addition, Figure 3.1 reports all the most relevant geometric parameters of the examined electric motor.

The power converter features a multilevel switching cell array architecture [94], including six main boards or legs, one for each phase. Each converter leg is composed by 8 Switching Cells (SCs) and 12 capacitors. The switching cell is the basic building block of the power converter, and it is made up of a main power switch and its driver, along with the iFuse, an additional switch used to deactivate the switching cell in case of a fault. These power devices are directly embedded in the PCBs of the legs to improve efficiency and reduce the overall size of the converter unit. Figure 3.2 shows a simple schematic depicting the layout of a single converter leg, where each square block roughly occupies a surface area of 30×30 mm on the PCB.

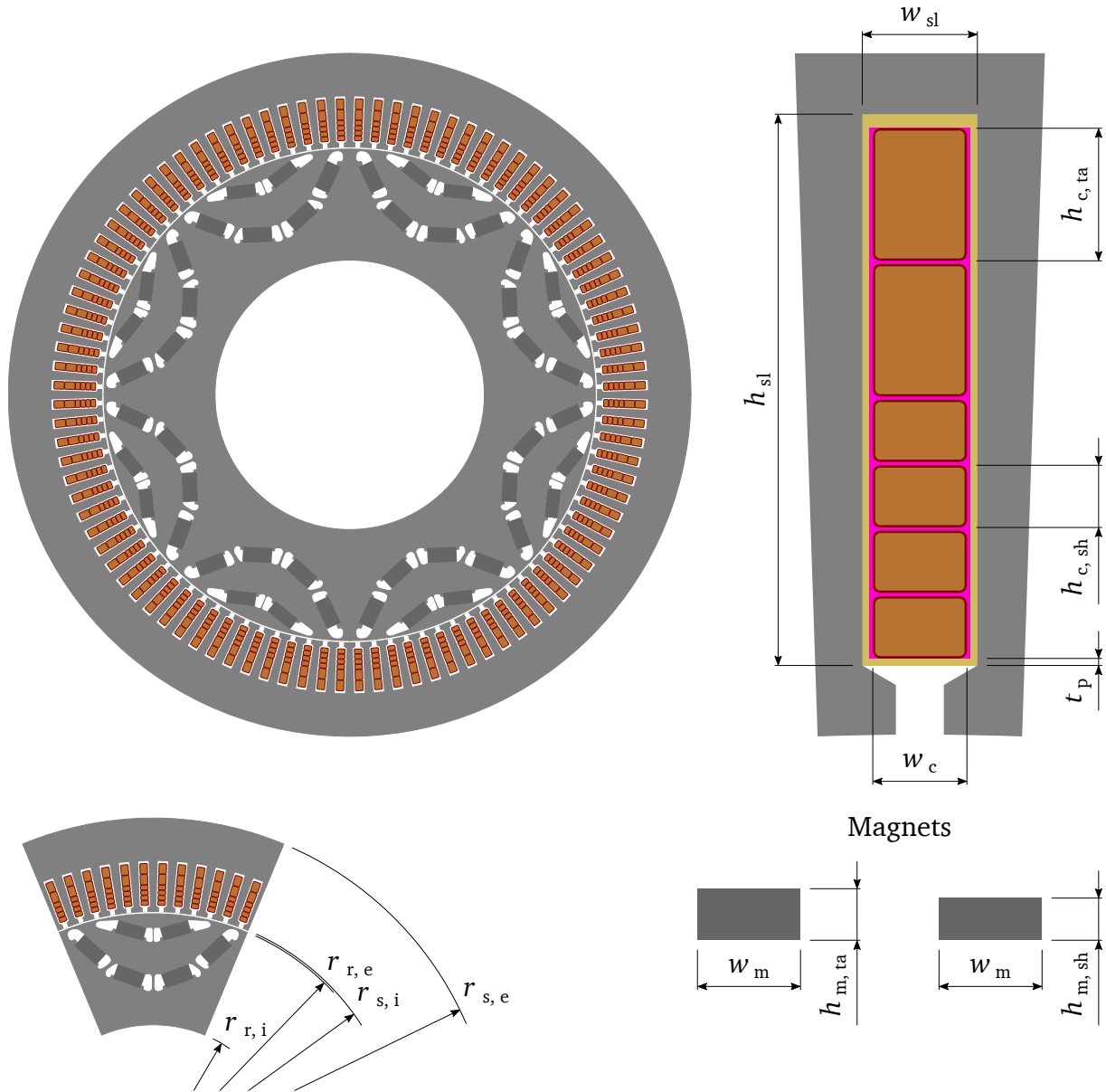


Figure 3.1. Cross-section of the electric machine [189], with the arrangement of magnets and windings.

The electric motor and converter legs are integrated by a shared water-cooled case that features a hexagonal shape with the external walls serving as cold plates for the converter legs. The cooling channels present a simple helical design with a height of 5 mm and a width of 40 mm. Figure 3.3 shows a simple schematic of the proposed cooling system where the six converter legs are mounted on the external walls of the cooling jacket.

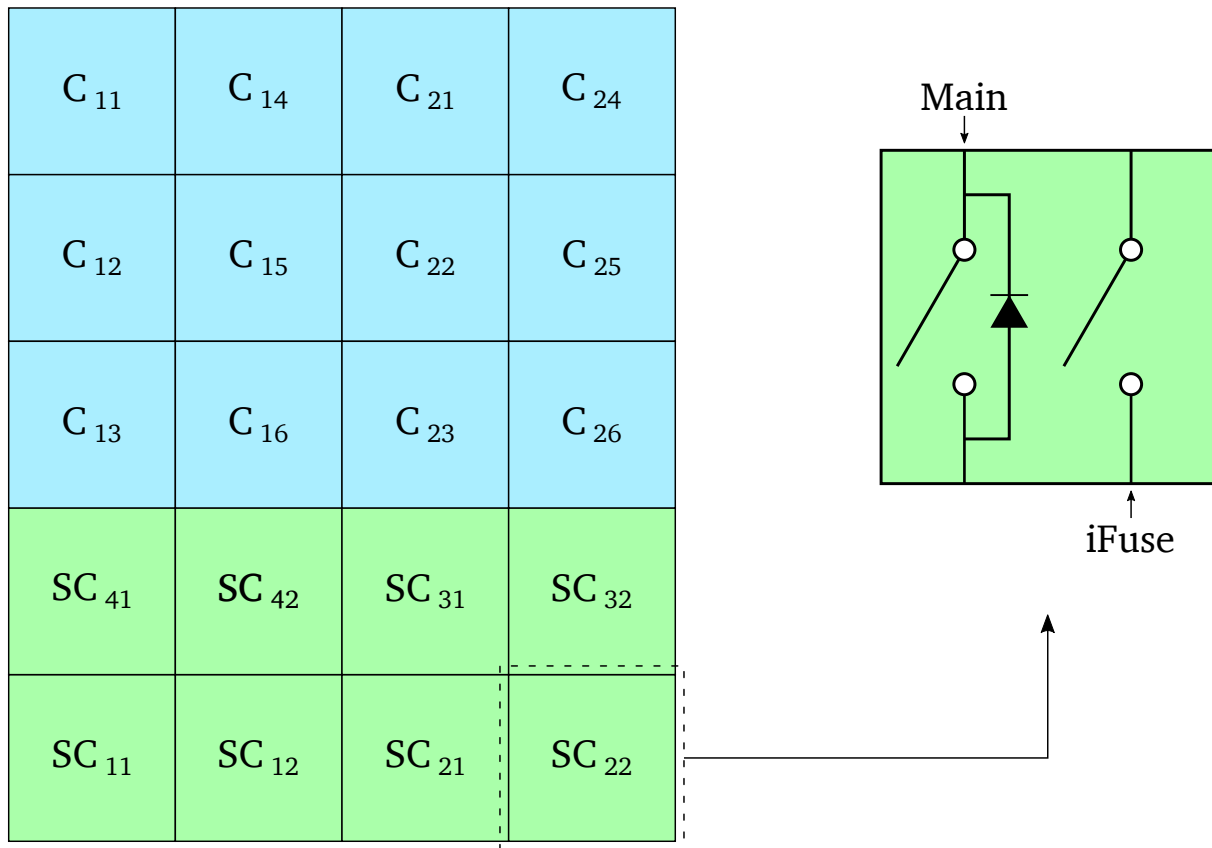


Figure 3.2. Arrangement of Switching Cells (SCs) and capacitors (Cs) constituting one of the six converter legs.

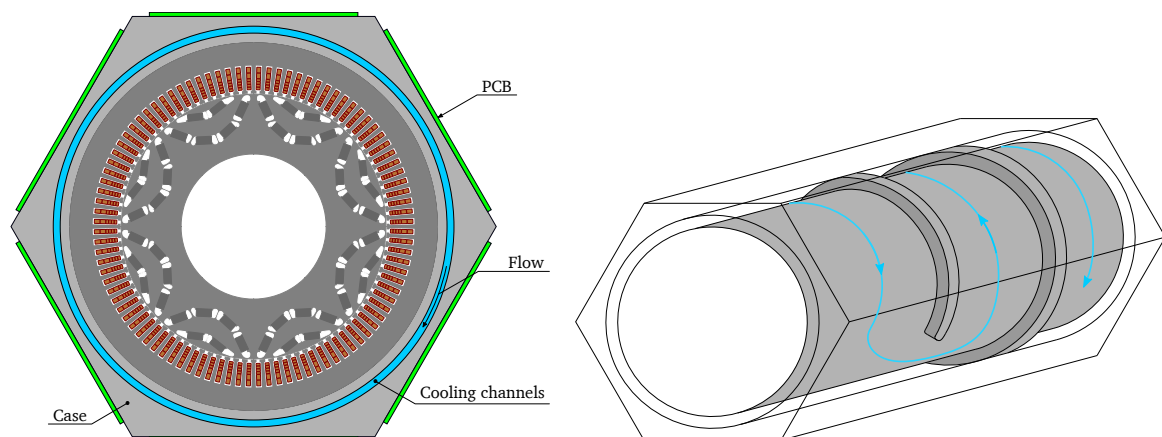


Figure 3.3. Schematic representing the integrated assembly comprising the electric motor, the PCBs of the power converter, and the water cooled hexagonal case.

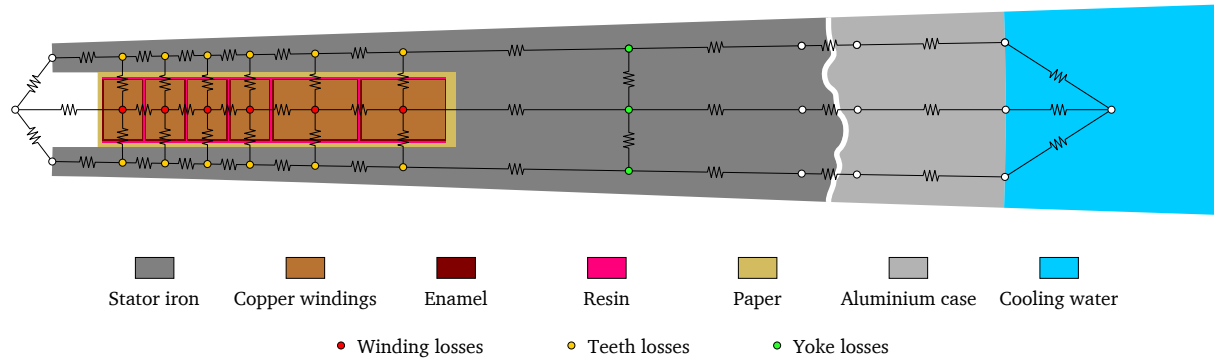


Figure 3.4. Schematic of the network of thermal resistances used to model the stator of the electric motor, with the main winding and insulation components highlighted.

3.3. Numerical methods

The integrated assembly composed by the electric motor, the power converter, and their shared cooling system is modelled by means of Lumped Parameter Thermal Networks, which are often referred to as LPTNs. Three LPTNs with varying number of degrees of freedom are developed with the aim of evaluating the effect of modelling complexity on numerical results, specifically on the estimation of hot-spot temperatures. The simplest of these models is a two-dimensional thermal network encompassing a single slot of the electric motor, a single converter leg, and a sixth of a channel of the cooling jacket. The other two thermal models are developed by extending the 2D LPTN along the axial and circumferential directions, resulting in a 3D thermal model of a slice of the assembly including a single slot of the motor and a single leg, and a 3D LPTN modelling the complete assembly. The development of the aforementioned lumped parameter thermal networks is presented in the following.

3.3.1. Two-dimensional thermal model

The physical domain of the electric motor is reduced down to a basic geometry consisting of a single slot, along with the associated conductors and insulation, and half of the two adjacent teeth. This allows for a vast reduction in the number of nodes included in the thermal model, in addition, this basic thermal network can be repeated along the circumferential and axial directions to easily extend the computational domain. The thermal model of the stator is constructed by dividing the domain in regions or zones and placing a temperature node in the middle of each heat generating component, namely the six copper bars, the corresponding teeth sections, and the yoke of the stator, as shown in the drawing of Figure 3.4. The thermal network is completed by the nodes standing for the temperatures of the air gap, the surface of the teeth, the surface of the case, and the mean bulk temperature of the cooling fluid. The conductive thermal resistance between a pair of nodes is evaluated by summing the contributions given by each layer of material:

$$R_{ij} = \sum_l \frac{1}{S_l \lambda_l}, \quad (3.1)$$

where S_l and λ_l are the conduction shape factor and thermal conductivity of a single layer, respectively. The shape factors are computed on the basis of geometric data. In the windings, flat-plate shape factors are used for the copper busbars and the insulation layers:

$$S_{fp} = \frac{A}{t}, \quad (3.2)$$

where A and t are the cross-sectional area and length associated with the path taken by the heat flux, respectively. In the magnetic circuit of the stator, shape factors for radial and circumferential geometries are used. These are computed by means of the general approach proposed by Yovanovich [190]:

$$S_r = \frac{\Delta z \Delta \varphi}{\ln(r_{ext}/r_{int})}, \quad S_\varphi = \frac{\Delta z \ln(r_{ext}/r_{int})}{\Delta \varphi}. \quad (3.3)$$

The values of thermal conductivity considered for all the materials making up the electric motor are listed in Table 3.1. The conductive resistances between the node of one conductor and the adjacent node of a tooth are obtained from Equation (3.1), considering half a busbar, the enamel, the resin, the paper, and half of the section of tooth associated with the node under consideration:

$$R_{ct,k} = \frac{0.5 w_c}{h_{c,k} l_{ss} \lambda_c} + \frac{t_e}{h_{c,k} l_{ss} \lambda_e} + \frac{t_{rs}}{h_{c,k} l_{ss} \lambda_{rs}} + \frac{t_p}{h_{c,k} l_{ss} \lambda_p} + \frac{0.25 \Delta \varphi_{tooth}}{\lambda_{cr} l_{ss} \ln(r_{ext,k}/r_{int,k})}, \quad (3.4)$$

where the index k refers to the considered conductor layer, starting from 0 at the layer closest to the rotor. l_{ss} stands for the portion of the active length of the stator included in the model. In this instance, it is important to underline that the value of l_{ss} does not affect the temperature results, as the losses are proportional to l_{ss} , while the thermal resistances

Table 3.1. Values of thermal conductivity of the materials associated with the main components of the electric motor.

Component	Material	λ [W m ⁻¹ K ⁻¹]
Stator and rotor laminations	NO20-1200H electrical steel	21.0
Winding conductors	Copper	385.0
Wire enamel	Voltatex®	0.23
Impregnation resin	SbTCM resin	3.20
Insulating paper	Kapton®	0.12
Magnets	N40UH NdFeB	6.45
Shaft	Steel	45.0
Case	Aluminium	237.0
Thermal pad	Carbon fibre	20.0

are inversely proportional to it. The thermal resistance between two adjacent conductors is obtained using a similar approach:

$$R_{cc,k} = \frac{0.5 h_{c,k}}{w_c l_{ss} \lambda_c} + \frac{2 t_e}{w_c l_{ss} \lambda_e} + \frac{t_{rs}}{w_c l_{ss} \lambda_{rs}} + \frac{0.5 h_{c,k+1}}{w_c l_{ss} \lambda_c}. \quad (3.5)$$

The other conduction resistances of the LPTN of the stator include a single layer of material and are computed according to Equation (3.1), by selecting the suited values of S and λ . The contact resistances between the stator and the internal wall of the cooling jacket are estimated by modelling heat transfer between the two uneven surfaces as heat conduction through an equivalent layer of still air. The associated thermal resistances are computed by considering a flat-plate shape factor:

$$R_{co} = \frac{t_{eq}}{\Delta \varphi r_{s,ext} l_{ss} \lambda_{air}} \quad (3.6)$$

where t_{eq} is the thickness of the equivalent layer of air between the stator and case. t_{eq} is set to 0.006 mm, as it representative of a coupling between iron and aluminium components with loose tolerances [31], which is a conservative assumption from the point of view of thermal management.

The convective resistances modelling heat transfer in the air gap and cooling jacket are defined as follows:

$$R_{conv} = \frac{1}{UA}, \quad (3.7)$$

where U is the convective heat transfer coefficient. In the air gap, U -values are obtained from the correlation by Bjorklund and Kays [191] for Taylor-Couette flows:

$$Nu = \frac{U D_h}{\lambda} = 0.35 Ta^{0.25}, \quad (3.8)$$

where Ta is the Taylor number, which is computed on the basis of the angular velocity of the rotor ω , its radius $r_{r,ext}$ and the height of the air gap h_{ag} :

$$Ta = \frac{\omega^2 r_{r,ext} h_{ag}^3}{\nu^2}. \quad (3.9)$$

The heat transfer coefficients associated with the cooling jacket are derived from the dedicated CFD model presented in Section 3.3.6.

The heat generated in the conductors is modelled by means of a modified Joule heating law including the temperature scaling laws proposed by Wrobel *et al.* [188]:

$$\dot{Q}_{c,k} = \gamma_{cu} \frac{l_{ss}}{A_{c,k}} I^2 \left[\frac{K_{rk}|_{T_{ref,cu}} - 1}{\sqrt{1 + \alpha_{cu}(T - T_{ref,cu})}} + 1 + \alpha_{cu}(T - T_{ref,cu}) \right], \quad (3.10)$$

where γ_{cu} is the electrical resistivity of copper, which is evaluated at the reference temperature $T_{\text{ref, cu}} = 20^\circ\text{C}$: $\gamma_{\text{cu}} = 1.724 \times 10^{-8} \Omega\text{m}$. α_{cu} is the temperature coefficient of copper, which equates to $4.29 \times 10^{-3} ^\circ\text{C}^{-1}$. The K_{rk} coefficient represents the ratio between AC and DC losses of the k -th conductor. In general, the AC/DC loss ratio depends on the frequency of the phase current, the position and geometry of the conductor, and the resistivity of its material, which in turn is influenced by temperature. In this case, the K_{rk} coefficients are derived from 2D Finite Element Analyses analogous to those presented in [189, 192], which are conducted considering a uniform winding temperature of 120°C . The values of K_{rk} stemming from these analyses are reported in the graph of Figure 3.5, for all the conductor layers and for Low Saturation (LS) and High Saturation (HS) conditions. The values of the AC/DC loss ratio are scaled from 120°C to 20°C using the approach in [188] so that they can be scaled consistently with temperature once they are inserted in Equation (3.10).

The losses associated with the laminations of the yoke, the teeth, the rotor, and the magnets are derived from the aforementioned FE models, and are listed in Table 3.2. In this case, temperature effects are not considered as they are much less relevant compared to those in the copper windings. These losses are assigned to the nodes related to their respective components (see Figure 3.4) by assuming uniform heat generation across the volume of the examined component:

$$\dot{Q}_i = \frac{\dot{Q}_{\text{cmp}}}{V_{\text{cmp}}} V_i, \quad (3.11)$$

where V_i is the volume of the region identified by node i , while the subscript cmp stands for component of the motor under examination, among those in Table 3.2.

The thermal model of the rotor is developed by simplifying the actual geometry of the rotor to represent it in a cylindrical reference frame at a reduced node count. The discrete magnets shown in Figure 3.1 are substituted by two equivalent magnet layers, defined by uniformly distributing the volume of the discrete magnets across the whole rotor. The conduction thermal resistances are obtained from Equation (3.1) using the radial and circumferential shape factors in Equation (3.3). The simplified geometric arrangement of the rotor is shown in the drawing of Figure 3.6, along with its thermal network model.

The thermal model of the power converter and external wall of the cooling jacket features a much simpler layout compared to the LPTNs of the stator and rotor. The nodes standing for the main and iFuse switches are connected in parallel to the last copper layer of the

Table 3.2. Values of thermal conductivity of the materials associated with the main components of the electric motor.

Component	Yoke	Teeth	Rotor	Magnets
Losses [W]	1028.5	848.7	290.0	7.952

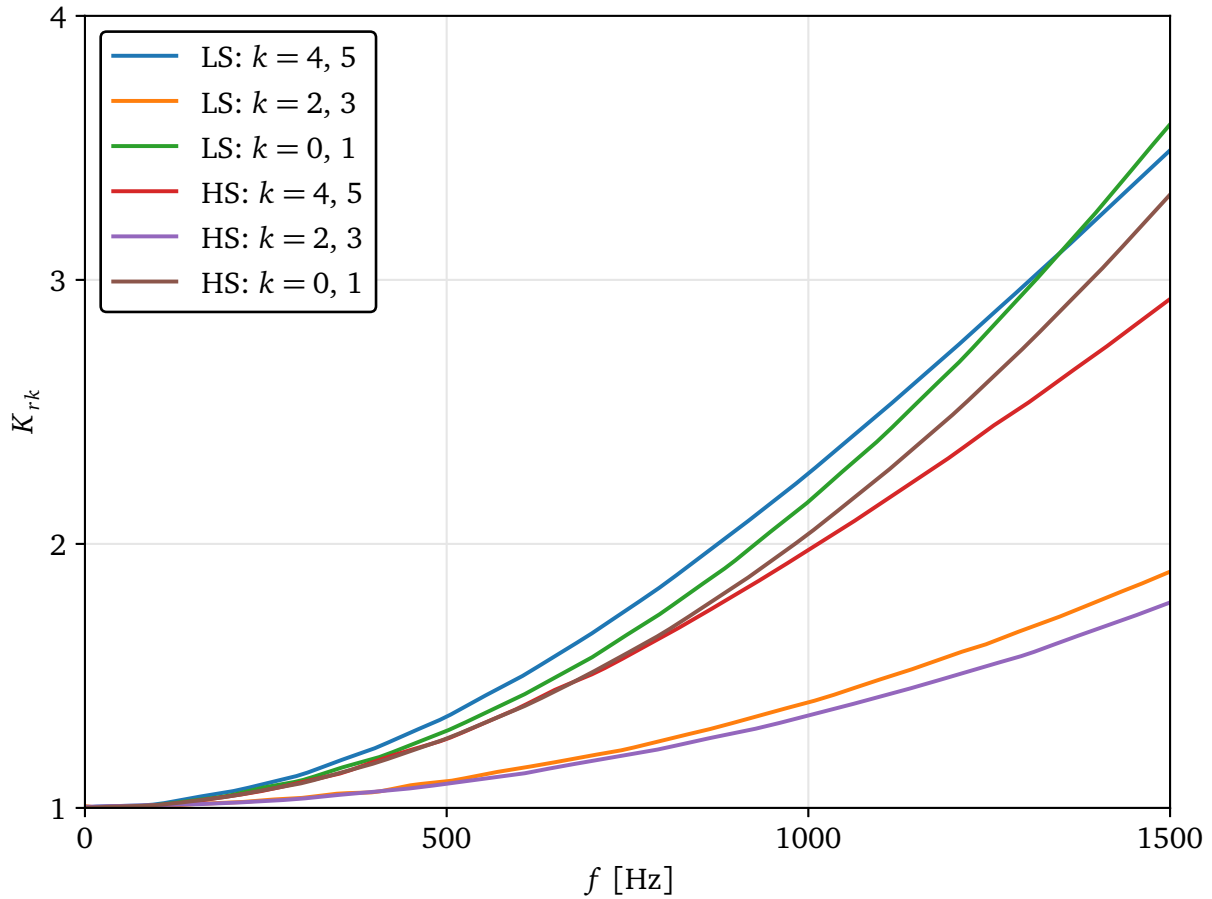


Figure 3.5. Values of the K_{rk} coefficient as a function of frequency for all conductor layers.

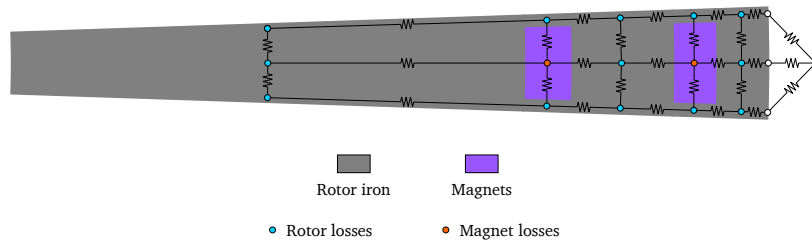


Figure 3.6. Arrangement of nodes and thermal resistances making up the rotor thermal model.

PCB through $R_j = 0.62 \text{ } ^\circ\text{C/W}$ [193], which models heat conduction between the junction of the semiconductors and the last copper layer of the PCB. The latter is connected to the case by means of a conductive carbon fibre thermal pad that has a thickness of $t_{\text{pad}} = 0.4 \text{ mm}$. In this case, conductive thermal interface materials can be used without causing short circuits as the semiconductors are internally insulated. The convective resistance between the cooling fluid and the external walls of the case is obtained from the CFD model presented in Section 3.3.6. The switching and conduction losses of each power device are derived from initial estimates obtained by using the approach presented in [194], and by considering the reference temperature $T_{\text{ref},j} = 100 \text{ } ^\circ\text{C}$. These losses are listed in Table 3.3, where values in red refer to conduction losses in iFuses, occurring when these devices are made out of the

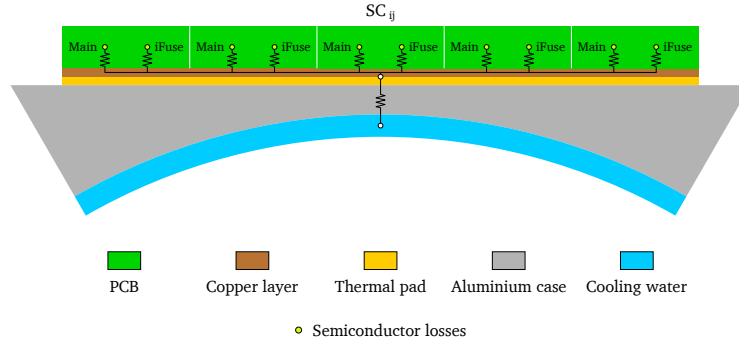


Figure 3.7. Thermal network of one of the six converter legs.

same semiconductors used for the main switches, which are optimised for switching performance rather than pure conduction. On the other hand, values in green are referred to a power switch optimised for continuous conduction. In this case, the losses of each switching cell are computed considering the non-optimal configuration as it is more demanding from the thermal point of view. In addition, using the same power switch for the main and iFuse allows for a simpler chip embedding procedure. Conduction losses in Table 3.3 are scaled with the junction temperature of the semiconductors according to the following relation:

$$\dot{Q}_{\text{cond}}|_T = \dot{Q}_{\text{cond}}|_{T_{\text{ref},j}} \left[1 + 1.429 \times 10^{-3} (T - T_{\text{ref},j}) \right]. \quad (3.12)$$

The thermal networks modelling the power converter, stator, and rotor are connected at the nodes standing for the temperatures of the coolant and the air gap as shown in Figure 3.8, forming the thermal model of the overall system.

Table 3.3. Conduction, switching and total losses for each switching cell.

Conditions: $P = 100 \text{ kW}$, $m = 0.75$, $\varphi = 30^\circ$, $P_{\text{cond}} = 0.5 P_{\text{sw}} @ I_{\text{nom}}$

Component	Sub-component	P_{cond} [W]	P_{sw} [W]	P_{tot} [W]	
$SC_{11}, SC_{12}, SC_{41}, SC_{42}$	Q1 (main)	9	43	52	$\rightarrow 61/54$
	Q2 (iFuse)	9/2	0	9/2	
SC_{21}, SC_{31}	Q1	7	21	28	$\rightarrow 35/29$
	Q2	7/1	0	7/1	
SC_{22}, SC_{32}	Q1	41	0	41	$\rightarrow 82/49$
	Q2	41/8	0	41/8	
C_{11}, C_{26}	–	–	–	0.085	

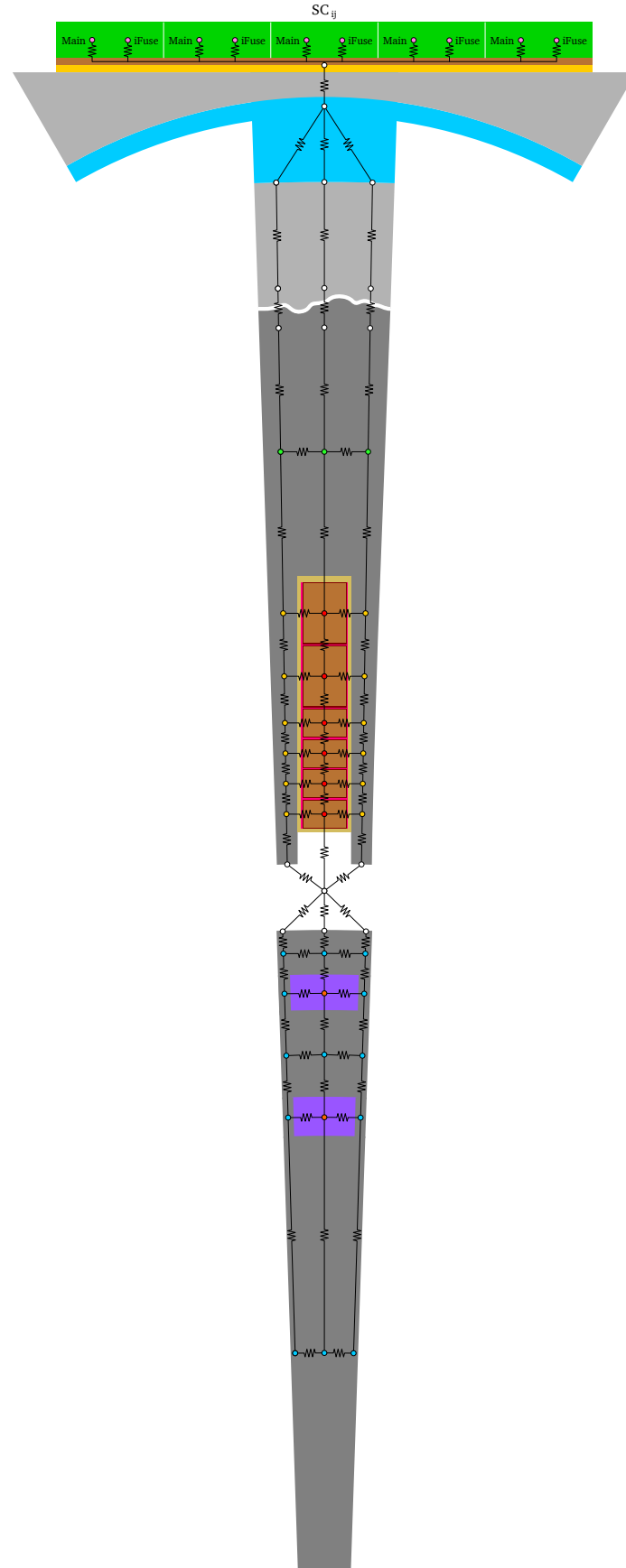


Figure 3.8. Two-dimensional lumped parameter thermal network modelling the examined integrated motor drive.

3.3.2. Three-dimensional thermal model of a single slot

The basic two-dimensional thermal model developed in Section 3.3.1 allows for quick evaluations of the hot-spot temperatures and overall thermal management of the IMD assembly. However, its intrinsic simplicity introduces a couple of limitations that could potentially affect the accuracy of the resulting estimates. For one, the adopted two-dimensional framework does not allow for the introduction of the losses generated in the end windings, and the computation of the temperatures in this region. Although the losses in the end windings are much lower than those observed in the active region due to reduced proximity effects, it is important to consider the thermal management of the exposed conductors on both sides of the machine, as these are not in contact with the rest of the motor and cooling system, which may result in elevated temperatures during the operation of the machine. A further significant limitation of the 2D LPTN is the assumption of a uniform coolant bulk temperature, which of course is not verified in the real system.

The drawbacks and limitations brought by the two-dimensional thermal model are addressed by extending it along the axis of the motor to derive a primitive 3D LPTN encompassing a single slot of the electric machine and a single converter leg. The arrangement of nodes and resistances constituting the thermal models of the stator and rotor is replicated three times, subdividing the domain into three axial slices, as shown in Figure 3.9. The nodes that identify a subregion of the model, namely all centre-point nodes, are connected by axial resistances that are defined on the basis of the shape factor for flat plates in Equation (3.2). The end windings are modelled by considering six open-ended conductors on either side of the active section of the motor. A temperature node is positioned at the centre of each section of conductor extending from the stator. These nodes are connected to the corresponding nodes in the active section of the machine by conductive thermal resistances. The losses generated in the exposed conductors are evaluated by means of Equation (3.10), using the K_{rk} coefficients derived from a 3D Finite Element model that includes the complete end winding arrangement [26]. These values are reported in the plot of Figure 3.10, as a function of frequency. In this case, the K_{rk} -values are significantly lower when compared to those related to the active section presented in Figure 3.5, as proximity effects are almost negligible in the end windings, and AC losses are mostly influenced by the skin effect, which is produced by the conductor's own flux linkage [25]. The lateral surfaces of the end windings are assumed adiabatic, as this represents the most unfavourable scenario for thermal performance. The advection of thermal energy due to coolant flow in the case is modelled by connecting the nodes standing for the bulk temperature of the fluid with advective or transport thermal resistances. These are defined by considering the thermal energy balance applied to the inlet and outlet sections of a simple duct:

$$R_{\text{adv}} = \frac{\Delta T}{\dot{Q}} = \frac{T_{b,o} - T_{b,i}}{\dot{m} c_p (T_{b,o} - T_{b,i})} = \frac{1}{\dot{m} c_p}. \quad (3.13)$$

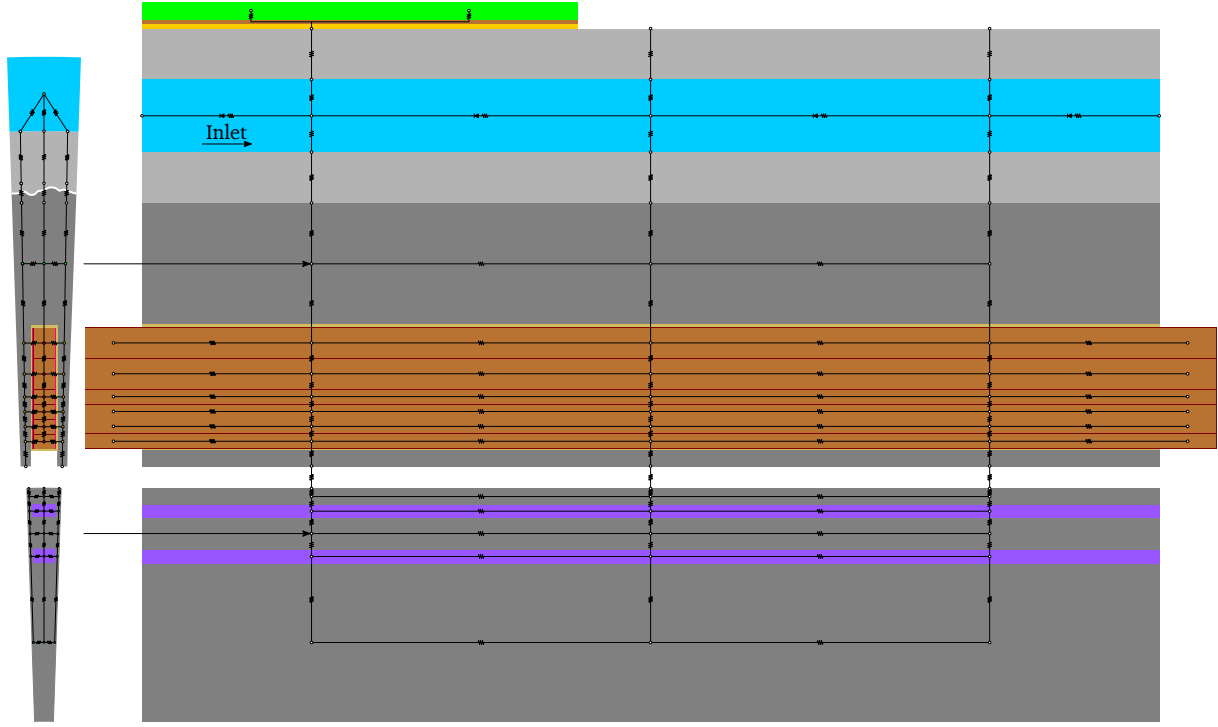


Figure 3.9. Longitudinal section of the 3D LPTN of a single slot positioned at the centreline of the windings.

R_{adv} , unlike the other thermal resistances, is not bidirectional. Specifically, it allows for heat to flow only in one direction. A close analysis of the third term of Equation (3.13) reveals that R_{adv} is oriented along the direction opposite to that of the velocity vector or, in other words, it is oriented from $T_{b,o}$ (outlet) to $T_{b,i}$ (inlet). The directionality of R_{adv} is modelled by an in-line thermal diode, which is represented by a one-sided connection at the topology level, as discussed in Section 2.2.

The LPTN presented in Figure 3.9 is used to model the complete integrated motor drive by considering several of these axial slices encompassing a single slot connected in parallel. To this end, all thermal resistances apart from advective ones are divided by the number of parallel connections n_{par} , while all losses are multiplied by it:

$$R_{ij,full} = \frac{R_{ij}}{n_{par}}, \quad \dot{Q}_{i,full} = \dot{Q}_i n_{par}, \quad (3.14)$$

where n_{par} is equal to the number of slots (96) for all resistances and losses associated with the motor, while it is equal to 6 for the resistances and losses associated with the power converter, as the latter presents six converter legs. These assumptions allow for a correct evaluation of the coolant temperature increase at a reduced node count, as the losses of the complete assembly are injected into a simplified model comprising nodes and resistances associated to a single slot of the motor and a single converter leg.

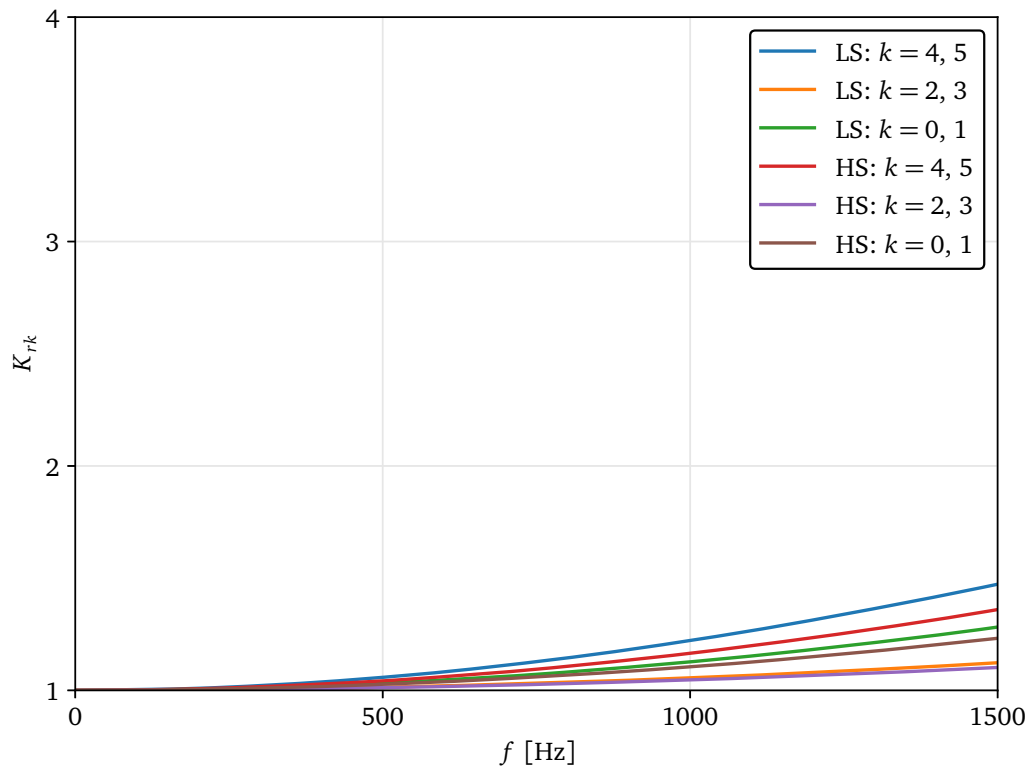


Figure 3.10. Values of the K_{rk} coefficient obtained in the end winding region, as a function of frequency for all conductor layers [26].

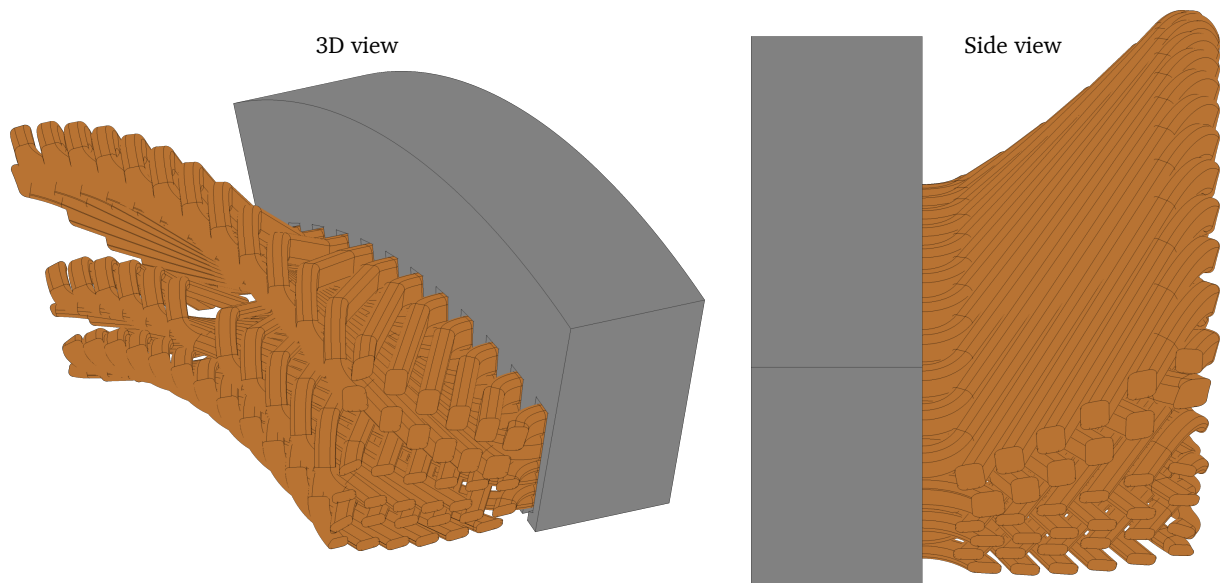


Figure 3.11. Selected views of the end windings associated with a 45° slice of one end of the electric motor.

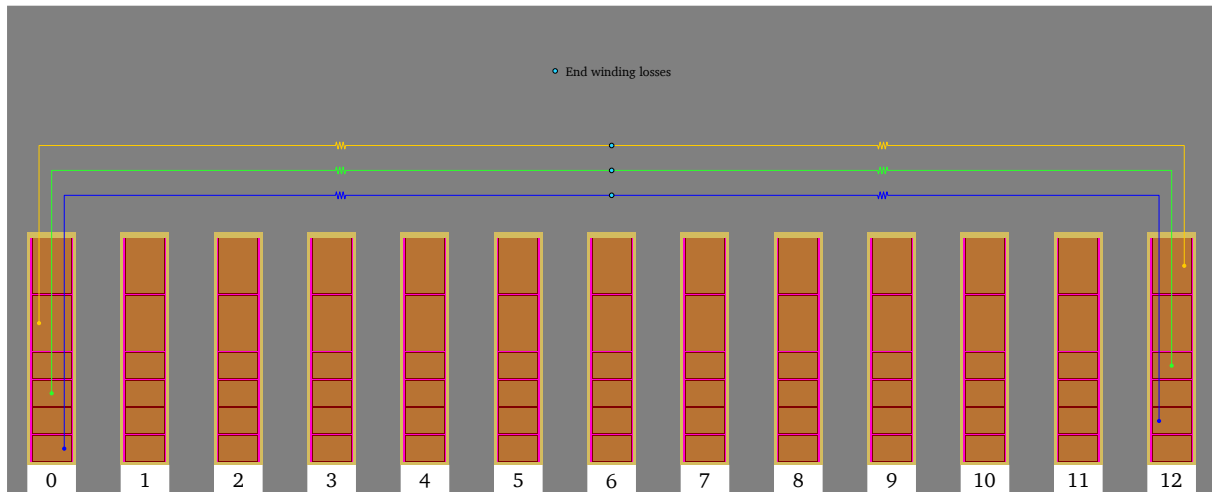


Figure 3.12. Example of thermal connections between winding layers.

3.3.3. Complete 3D thermal model

The basic three-dimensional model developed in Section 3.3.2 allows for an estimation of the average thermal behaviour of the integrated motor drive. However, this simplified 3D approach, much like the 2D LPTN, features some limitations when compared to a complete three-dimensional model. The most relevant of these drawbacks is the lack of an important path of heat flow associated with the motor, which consists in the connections between the various conductor layers that are formed in the end windings, as shown in Figure 3.11. This heat path is particularly relevant as the thermal conductivity of copper is several orders of magnitude greater than those of the other materials making up the motor.

The complete three-dimensional model of the integrated motor drive is obtained by starting from the 2D LPTN, and reproducing its arrangement of nodes and resistances 96 times along the circumferential direction and three times along the axial one. All centre-point nodes are connected by circumferential and axial resistances defined on the basis of their respective conduction shape factors. By modelling the complete motor, now the connections between the winding layers can be accurately represented. The thermal connections are arranged as the electrical connections, as shown in Figure 3.12. Starting from the conductor closest to the rotor, each busbar is connected to the one positioned one layer above it, and associated with a slot found at a distance of 12 slots from the starting one. The coolant temperature nodes associated with the slots are connected by advective resistances and thermal diodes following the path of coolant within the case, as shown in Figure 3.13. The sub-models for the converter boards are connected to the coolant temperature nodes of slots 0, 16, 32, 48, 64, and 80. Figure 3.14 reports a schematic of the front face of the complete 3D thermal model, where the connections between the converter sub-models and the coolant temperature nodes are clearly shown.

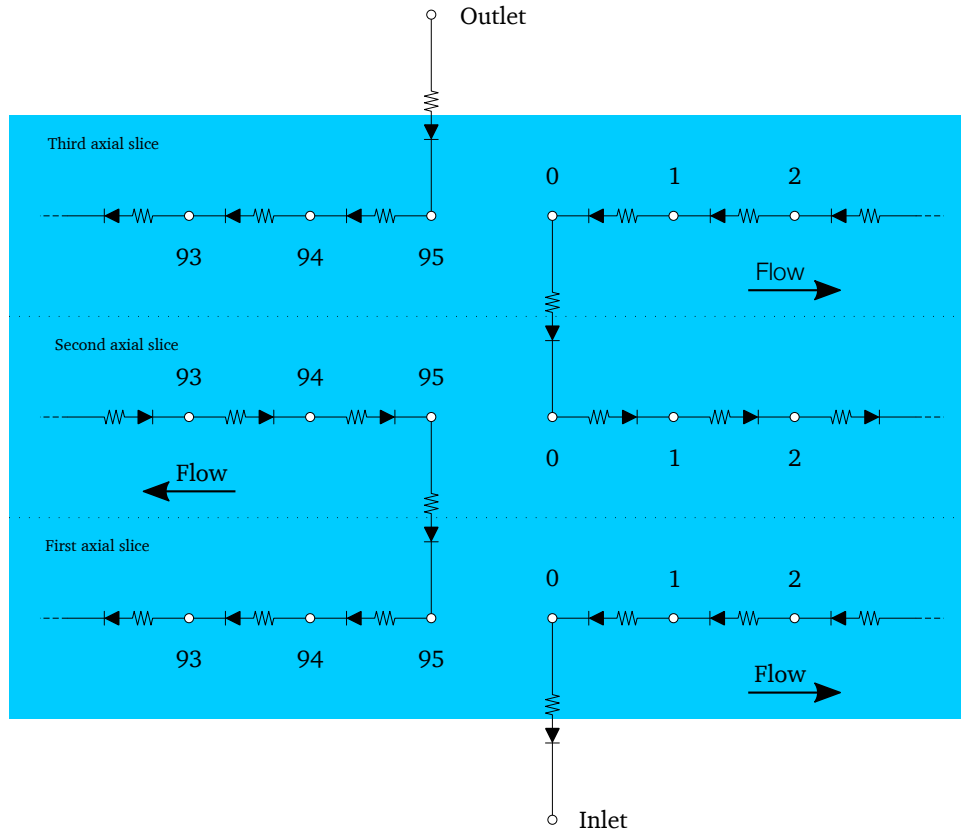


Figure 3.13. Example of thermal connections between the nodes modelling the coolant bulk temperature values.

3.3.4. Boundary conditions and implementation

The boundary conditions applied to the thermal models consists of those defining the operating point of the motor, and the temperature boundary condition considered for the cooling system. The root-mean-square value of the phase current is set to 80 A, which corresponds to an overall electric power of 100 kW. The frequency of the phase current is set to 600 Hz leading to a shaft rotational speed of 9000 rpm, which is suited for the desired vehicular application. In the 3D models, the inlet temperature of the coolant is arbitrarily set to 60 °C, while in the 2D LPTN the coolant temperature is set to 68.6 °C, as it corresponds to the mean bulk temperature associated with the whole system. The coolant flow rate is set to 5 l/min. The boundary conditions described herein are summarised in Table 3.4. The lumped parameter thermal networks are implemented in a Python 3 environment using the approach for the solution of networks of thermal resistances developed in Section 2.6. The LPTNs are implemented using a modular approach so that the basic network forming the 2D LPTN can be simply extended to form the other two networks using routines based on for cycles.

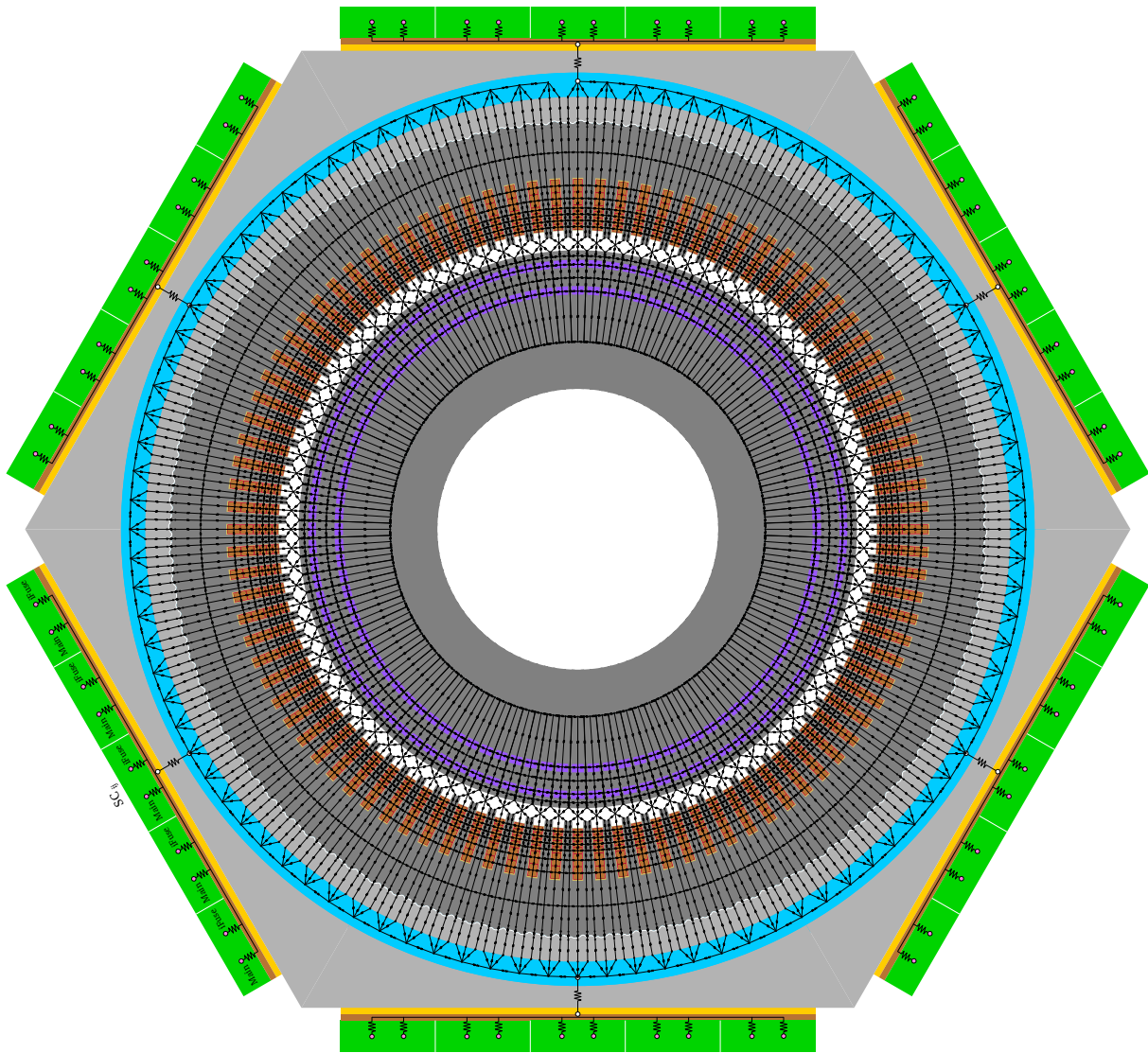


Figure 3.14. Schematic showing the front face of the complete three-dimensional thermal model.

3.3.5. Validation of advective resistances and thermal diodes

The correct implementation of the advective resistances included in both the 3D LPTNs presented in Section 3.3.2 and Section 3.3.3 is verified by comparing numerical results on

Table 3.4. Boundary conditions applied to the thermal models developed for the integrated motor drive under examination.

Parameter	Value	Units
Phase current	80	A
Rotor speed	9000	rpm
Frequency	600	Hz
Coolant flow rate	5	l/min
Mean coolant temperature (2D model)	68.6	°C
Inlet coolant temperature (3D model)	60	°C

the coolant bulk temperature increase, obtained by considering the boundary conditions in Table 3.4, with the expected ΔT_b -value retrieved from a simple thermal energy balance applied between the inlet and outlet of the cooling jacket:

$$\Delta T_b = \frac{\dot{Q}_{\text{tot}}}{\dot{m} c_p}, \quad (3.15)$$

where $\dot{Q}_{\text{tot}}$ is the sum of all losses introduced in the model. Table 3.5 reports the expected values of ΔT_b , their numerical counterparts, and the difference between them. Both the 3D LPTNs are able to correctly model the coolant temperature increase. The deviations noted between the expected and numerical values result from round-off errors typical of floating point arithmetic operations and are negligible with respect to the temperature values.

3.3.6. Numerical model of the cooling jacket

The heat transfer coefficients associated with the internal and external walls of the cooling jacket are obtained by developing a dedicated CFD model. In an effort to reduce the required computational effort, a reduced domain comprising a sixth of a single channel is analysed under the assumptions of periodic and incompressible flow. The flow is governed by the RANS and energy equations:

$$\begin{cases} \nabla \cdot (\vec{v} \otimes \vec{v}) = -\frac{1}{\rho} \nabla p + \nabla \cdot [(\nu + \nu_t) \nabla \vec{v}] + \sigma \\ \nabla \cdot \vec{v} = 0 \\ \nabla \cdot (\rho \vec{v} h) + \nabla \cdot \left(\rho \vec{v} \frac{|\vec{v}|^2}{2} \right) = -\nabla \cdot \vec{q} + \nabla \cdot (\vec{\tau} \cdot \vec{v}) \end{cases} \quad (3.16)$$

Temperature in the aluminium walls is determined as follows:

$$\nabla^2 T = 0. \quad (3.17)$$

The $k - \omega$ SST turbulence model [195] is employed to compute eddy viscosity. The momentum source term σ in Equations (3.16) stands for the external forcing needed to maintain the desired flow rate at the channel vents. The channel height y is chosen as the reference length of the problem, and it is used to scale down the computational domain. The flow

Table 3.5. Expected and numerical values of bulk temperature difference obtained from the two three-dimensional thermal models.

	Expected ΔT_b [°C]	Model ΔT_b [°C]	Deviation [°C]
3D single-slot model	17.25	17.25	1.7×10^{-13}
Complete 3D model	17.25	17.25	4.7×10^{-11}

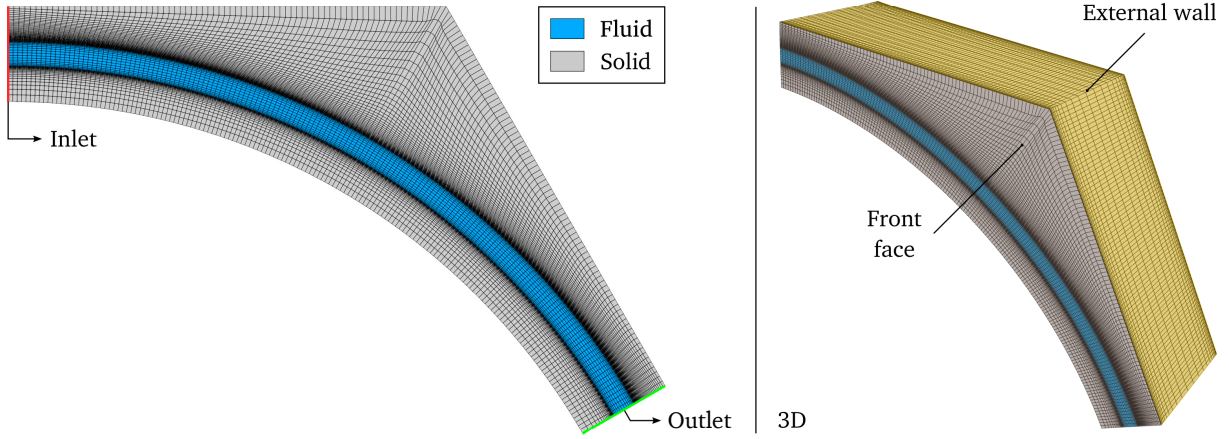


Figure 3.15. Side and 3D views of the cooling jacket computational mesh, where the inlet, outlet and fluid and solid regions are clearly identified.

is defined on the channel-height-based Reynolds and Prandtl numbers. Mean velocity over inlet and outlet sections, fluid density, and thermal conductivity are set to unitary reference values. Under the selected dimensionless framework, the dimensionless kinematic viscosity and specific heat of the fluid are:

$$\mu^* = \nu^* = \frac{1}{\text{Re}} \quad c_p^* = \text{Re Pr} . \quad (3.18)$$

Dimensionless properties of solid regions are derived by scaling the thermophysical properties of aluminium by the ones of the adjacent fluid.

$$\lambda_s^* = \frac{\lambda_{\text{alu}}}{\lambda_{\text{fluid}}} \quad (3.19)$$

A fully developed flow solution is attained by enforcing periodic boundary conditions between the inlet and outlet patches for all fields. No slip conditions are applied at the front and back faces of the domain, as well as at the fluid-solid interface, where the continuity of temperature and heat fluxes is also enforced. The two heat transfer coefficients are obtained by imposing and entering heat flux to the considered wall and setting the other as adiabatic. This procedure is repeated for the internal and the external walls, the latter serving as the cold plate for the converter legs.

The numerical solution of the governing equations is obtained by means of a Finite Volume conjugate heat transfer solver based on the SIMPLE algorithm [143], and implemented in the OpenFOAM computational suite [196]. Advective terms are discretised by means of second-order upwind schemes. Instead, central schemes featuring explicit non-orthogonality correction are used for diffusive terms. Figure 3.15 shows the selected views of the smoothed structured mesh employed for the discretisation of the computational domain. The channel's inlet and outlet sections, and the fluid and solid regions are clearly highlighted.

The influence of the grid on numerical results is assessed by developing three computational meshes with different cell sizes, and comparing the numerical results obtained. The coarse, medium, and fine computational meshes present analogous topology, with the wall normal spacing and mean size of the cells decreasing each time by one half from the coarse to the fine grid. Numerical results from the three computational grids are listed in Table 3.6, along with the average size of the cells $\bar{\Delta}^*$, the integral mean values of the dimensionless wall distance $\bar{y}^+$, and the value of the Nusselt number associated with the internal wall Nu_{int} obtained at zero grid spacing by means of Richardson's extrapolation [197], which is applied on the results from the medium and fine grids. Numerical results do not show a convergent behaviour with respect to the mean element size, as the coarse grid produces the maximum among the Nusselt number values observed, while the intermediate mesh accounts for the minimum. However, numerical results from the coarse grid may not be reliable as the mean $\bar{y}^+$ value is well above unity, which is considered the upper limit for the use of low-Reynolds turbulence models. The fine grid produces results close to the extrapolated continuum value, while the numerical outcomes obtained from the medium and coarse meshes present substantial deviations with respect to each other and the fine grid. For these reasons, the results from the fine grid are considered for all subsequent thermal analyses.

Table 3.6. Numerical results obtained from the coarse, medium, and fine computational grids, along with the continuum value obtained by means of Richardson's extrapolation.

	$\bar{\Delta}^*$	$\bar{y}^+$	Nu_{int}	Difference to previous grid
Coarse grid	0.1	4.7	24.9	
Medium grid	0.05	0.72	21.2	−14.9%
Fine grid	0.025	0.37	22.4	5.6%
Richardson's extrapolation	-	-	22.8	1.7%

3.3.7. Parameter sensitivity analysis

When dealing with complex physical or mathematical models it is often difficult to understand the relationship between the parameters upon which the model is built and its outputs. This aspect is often investigated by performing sensitivity analyses, that is, by varying the values of the parameters constituting the model and assessing the effects on the results. When developing thermal models for electric machines, sensitivity analyses become a useful tool to evaluate the effects of manufacturing tolerances and the choice of materials that make up the machine on its overall thermal performance. To this end, the single-slot three-dimensional thermal model is taken into consideration and the values of thermal conductivity and heat transfer coefficients are varied to assess their effect on the resulting temperature values. The deviations associated to each parameter are selected according to the following criteria. The thermal properties of the materials constituting the integrated assembly are varied by considering commercially available variants performing the same functions. In case of generic and well known materials like copper, the λ -value computed at 25 °C is varied by considering two temperature values: $T = 0$ °C that could be associated with a cold start in winter time, and $T = 180$ °C which is the thermal limit posed by the insulation class of the windings. The heat transfer coefficients and the junction thermal resistance are varied by an arbitrary $\pm 25\%$, which represents a reasonable estimation error that is usually associated with the use of correlations, CFD models, and basic estimates. The different materials considered for each subcomponent, their associated values of λ , and the resulting deviations with respect to the baseline values are reported in Table 3.7.

A special case is the one of the contact resistances between the stator and the case. These depend on two parameters: the thickness of the equivalent gap between the stator and case which is mostly influenced by manufacturing tolerances, and the thermal conductivity of air, which in turn depends on temperature. The baseline values of the contact resistances are computed by considering an equivalent gap of 0.006 mm, which is the upper limit for interface gaps usually observed for iron-aluminium interface couplings [31], and a value of λ_{air} computed at $T = 25$ °C. The contact resistances are varied by considering the gap-value of 0.0006 mm, which is the average minimum gap that can be achieved in these couplings [31], and by considering a value of λ_{air} computed at $T = 0$ °C, corresponding to a possible cold start condition. The effect of an increased air temperature in the interface gaps is not considered because even at $T = 180$ °C, the reduction of the contact resistances is negligible when compared to the effect given by the reduction of the equivalent gap. The percentage deviations related to the contact resistances are listed in Table 3.7.

The results of the sensitivity analysis are presented by contour plots of dimensionless temperature difference ϑ :

$$\vartheta = \frac{T - T_b}{(T - T_b)_{\text{bsl}}} , \quad (3.20)$$

where the subscript bsl denotes the temperature difference related to the baseline set of

Table 3.7. Values of thermal conductivity of the materials associated with the main components of the electric motor.

Component	Value	Materials	Limits	Deviations
Conductors	λ_c	Copper $T = 0\text{ }^\circ\text{C}$	404	+0.6%
		Copper $T = 180\text{ }^\circ\text{C}$	389	-3.0%
Enamel	λ_e	Kapton CR + NP1025 [198]	0.419	+82.2%
		Magnetemp CA 200 [198]	0.114	-50.4%
Resin	λ_{rs}	EpoxyLite [199]	0.85	-73.4%
		Varnish [199]	0.25	-92.2%
Paper	λ_p	DMD [175]	0.2	+66.7%
		TVAR [175]	0.116	-3.3%
Iron core	λ_{cr}	50A1300 [200]	63.1	+200%
		50A230 [200]	17.9	-14.8%
Magnets	λ_m	SmCo [201]	12.0	+85.9%
		NdFeB [201]	2.0	-69.0%
Thermal pad	λ_{pad}	Dielectric pad	6.5	-67.5%
		Dielectric pad	2.0	-90.0%
Air gap	U_{ag}		137	+25.0%
			87.9	-25.0%
Power converter	R_j		0.78	+25.0%
			0.5	-25.0%
Cooling jacket	U_{int}		2385	+25.0%
			1526	-25.0%
	U_{ext}		1849	+25.0%
			1184	-25.0%
Stator-case interface	R_{co}		0.426	+7.17%
			0.04	-90.0%

parameters. Thanks to the above formulation, the results become independent of the unit of measurement used for temperature, which is an improvement over existing sensitivity analyses available in the literature. ϑ directly represents the variation of the temperature increase of a given node with respect to the baseline. For example, $\vartheta = 1.2$ translates to a 20% larger temperature difference with respect to the baseline. Figure 3.16 shows contour plots of ϑ from the sensitivity analyses performed on the parameters associated with the stator of the motor, namely the λ -values of the insulation materials, the contact resistances, and the heat transfer coefficient associated with the cooling jacket. These results are associated with the front of the machine, where the coolant inlet is located.

Despite the broad temperature range considered for the computation of λ_{cr} , negligible effects are observed on the temperature outcomes, as the thermal conductivity of copper is orders of magnitude greater than the that of the other materials. Thus, it is not a major contributor to the equivalent thermal resistance of the stator. The different choice of insulating material for the lining of the conductors causes moderate ϑ deviations of +4%/-3%, which are concentrated in the vicinity of the first winding layer, which is also the one presenting the highest temperature values, since it features the largest equivalent thermal resistance to the cooling fluid with respect to the other layers. Thus, selecting a wire coating with improved thermal properties could provide slight performance benefits in terms of power output. The change of the impregnation resin appears to have similar effects to those observed for the wire enamel. An insulating paper with a greater λ -value seems to provide a moderate improvement to the thermal performance of the stator and winding assembly. However, these are concentrated at the last winding layer, while noticeably higher ϑ -values are noted in correspondence of the first layer, which represents the hot-spot of the motor. Thus, a paper with better thermal performance will provide smaller gains in terms of power output, than those that would be observed by improving the thermal conductivity of the enamel or the resin. The thermal model shows a marked sensitivity to the thermal conductivity of the iron, as it constitutes most of the stator and rotor of the machine. However, it is important to stress that any temperature change observed due to the change of λ_{cr} may be offset by the core losses, which might vary substantially with different choices of materials for the core. The trade-off between the electrical and thermal performance of electrical steel is strongly influenced by its silicon content. A steel with greater silicon content may present a low thermal conductivity, but at the same time it would reduce the losses due to eddy currents. Thus, identifying the optimum trade-off between electrical and thermal performance is a complicated task, that would require further numerical studies.

As expected, both the convective resistances associated with the cooling jacket and the contact resistances have a marked effect on the temperature field as these resistances are the first ones connected in series with the nodes standing for the coolant temperature. In both cases, the greatest deviations are observed near surface of the stator, while less marked effects are noted with increasing distance from the surface, as the relative contributions

given by the convective and contact resistances to the equivalent resistance between a given point and the cooling fluid decrease in favour of those given by conductive resistances.

Figure 3.17 shows contour plots of ϑ on a longitudinal cross-section positioned at the symmetry plane of the considered computational domain, and associated with $\pm 25\%$ deviations of the overall heat transfer coefficient associated with the cooling jacket. While noticeable axial gradients of ϑ can be observed close to the external radius of the motor, the temperature variations associated with the windings and insulation, which are the ones of interest for thermal management, remain practically constant along the axial development of the machine. This behaviour was noted for all the considered parameters.

Figure 3.18 shows the results obtained for the rotor. Compared to the stator, the rotor presents more uniform ϑ distributions. The thermal conductivity of the core and the heat transfer coefficient in the air gap are clearly the most influential parameters for the thermal performance of the rotor. On the other hand, the considered changes in λ_m lead to deviations in temperature difference that amount to a few percentage points. The ϑ -values obtained for the power converter are shown in Figure 3.19. All the parameters have substantial effects on the temperature field, as the thermal model of the power converter includes a lower amount of parameters compared to the other two LPTNs. The λ -value of the thermal pad has a marked influence on the temperature field, and the obtained ϑ -values underline the benefits given by the use of a conductive pad instead of an insulating one.

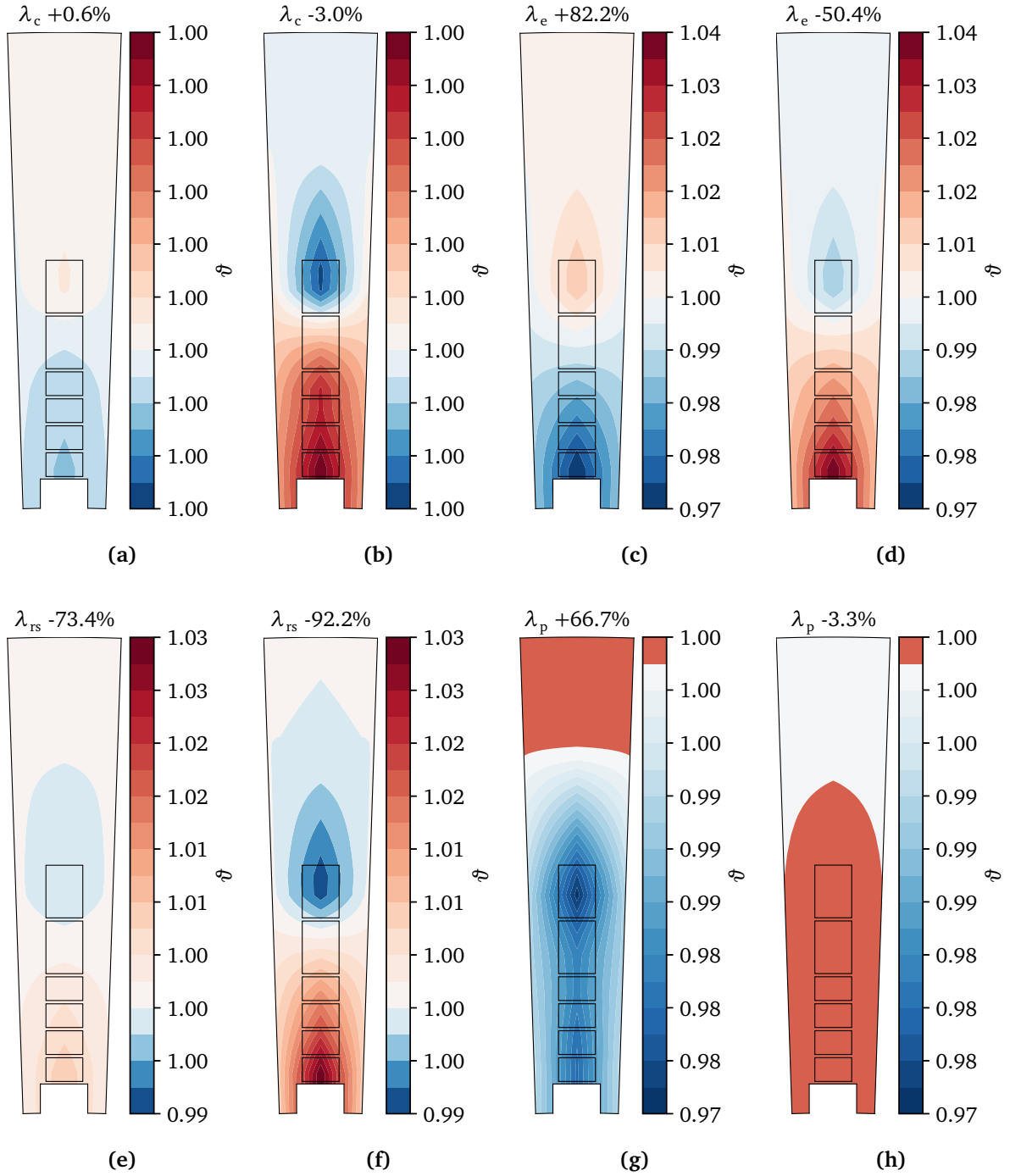


Figure 3.16. Contour plots of dimensionless temperature difference on the stator, obtained from the sensitivity analyses performed on λ_c (a, b), λ_e (c, d), λ_{rs} (e, f), and λ_p (g, h).

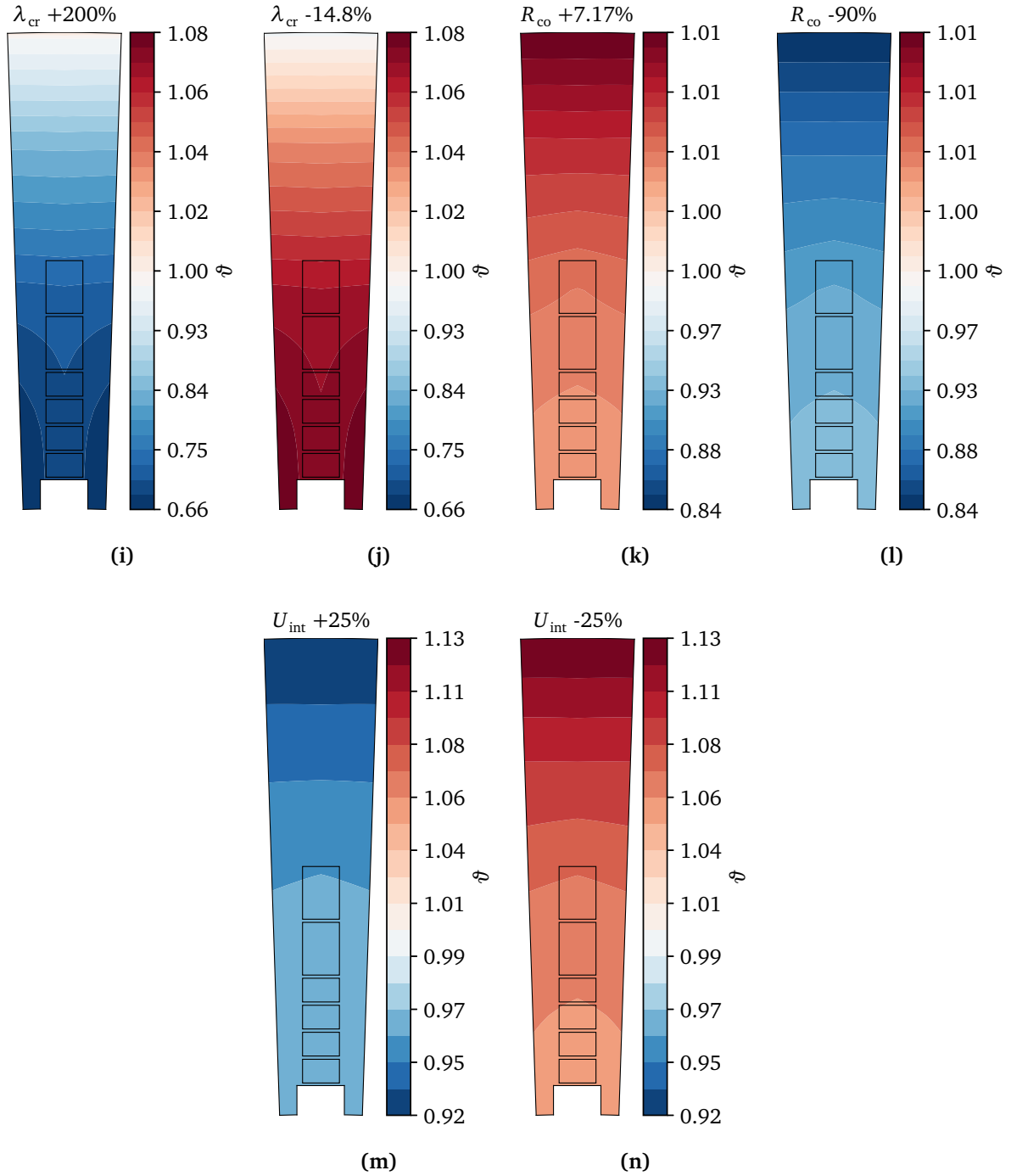
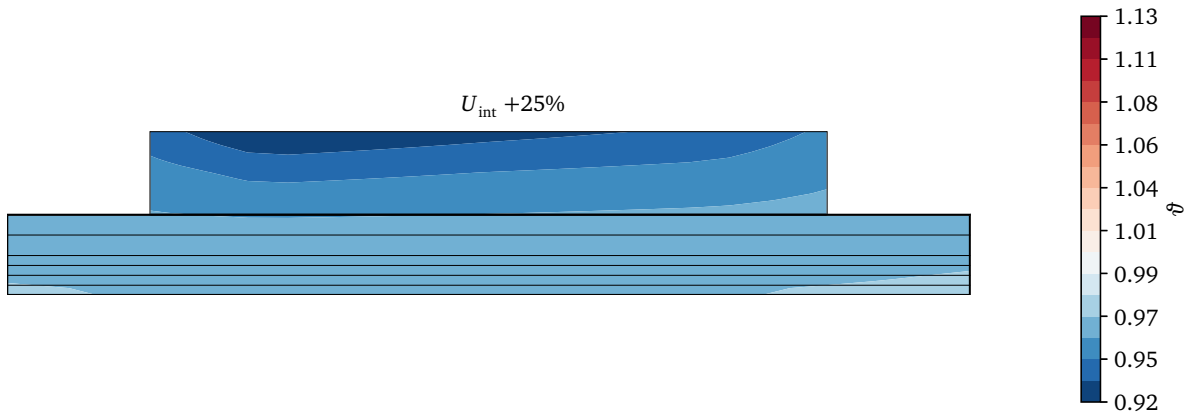
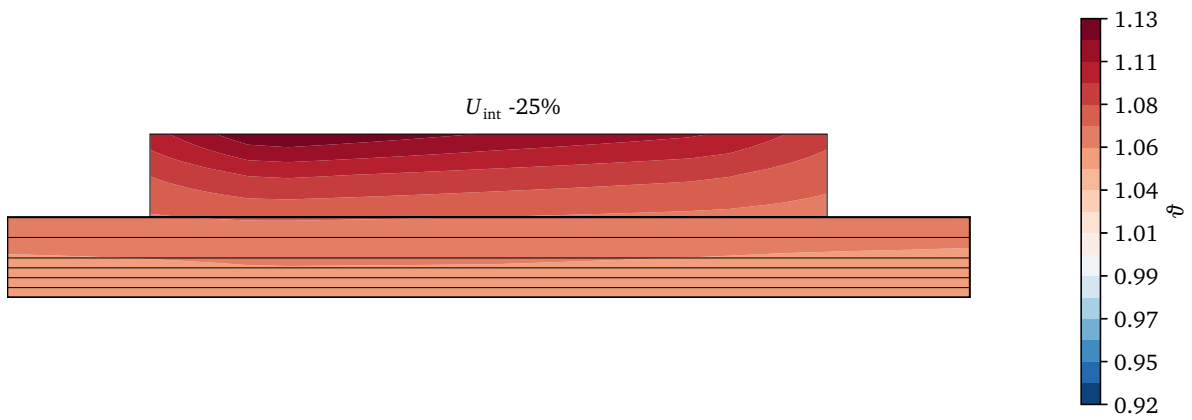


Figure 3.16. Contour plots of dimensionless temperature difference on the stator, obtained from the sensitivity analyses performed on λ_{cr} (i, j), R_{co} (k, l), and U_{int} (m, n).



(a)



(b)

Figure 3.17. Contour plots of the axial distribution of ϑ associated with $\pm 25\%$ variations of U_{int} .

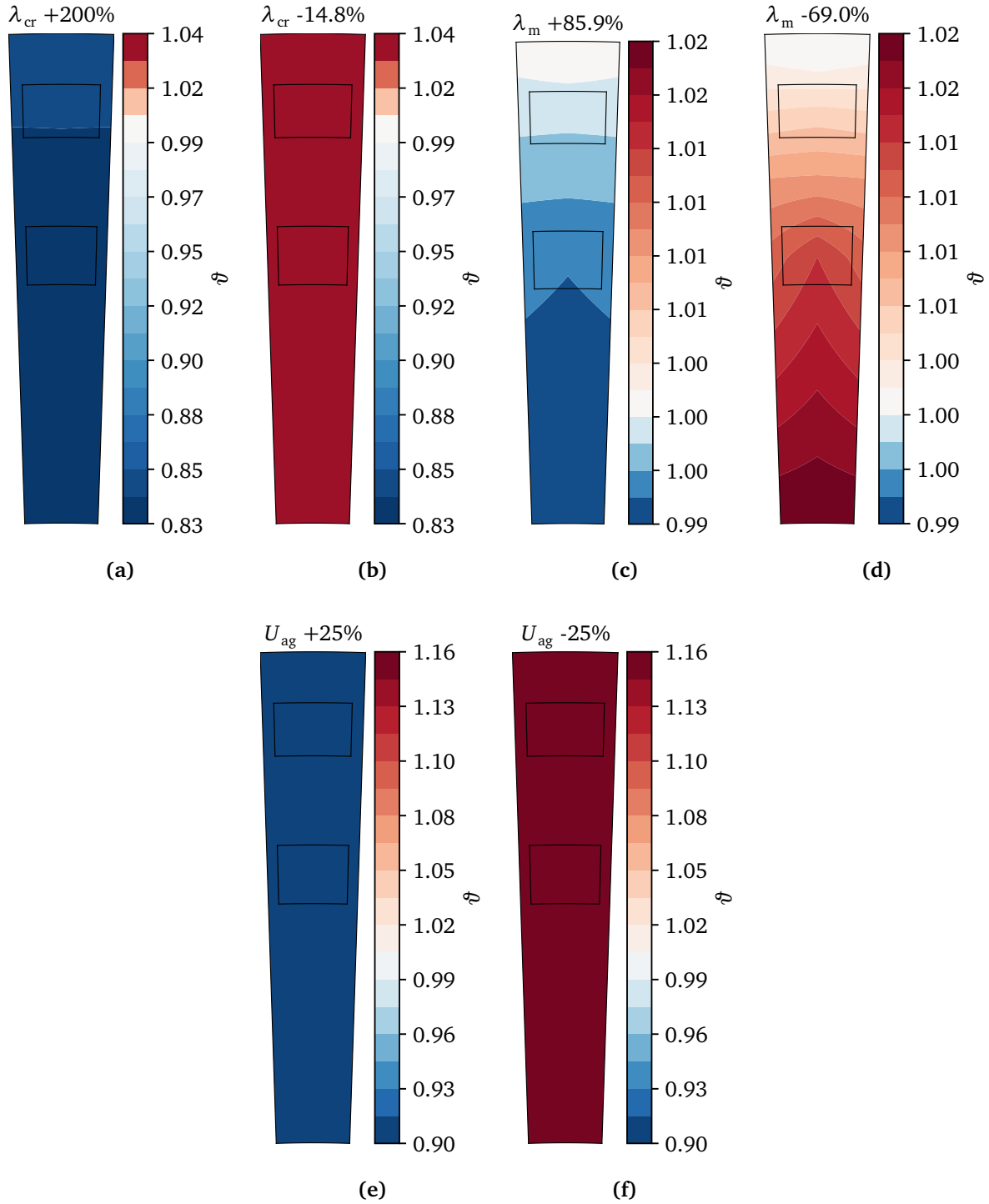


Figure 3.18. Contour plots of dimensionless temperature difference on the rotor, obtained from the sensitivity analyses performed on λ_c (a, b), λ_m (c, d), and U_{int} (e, f).

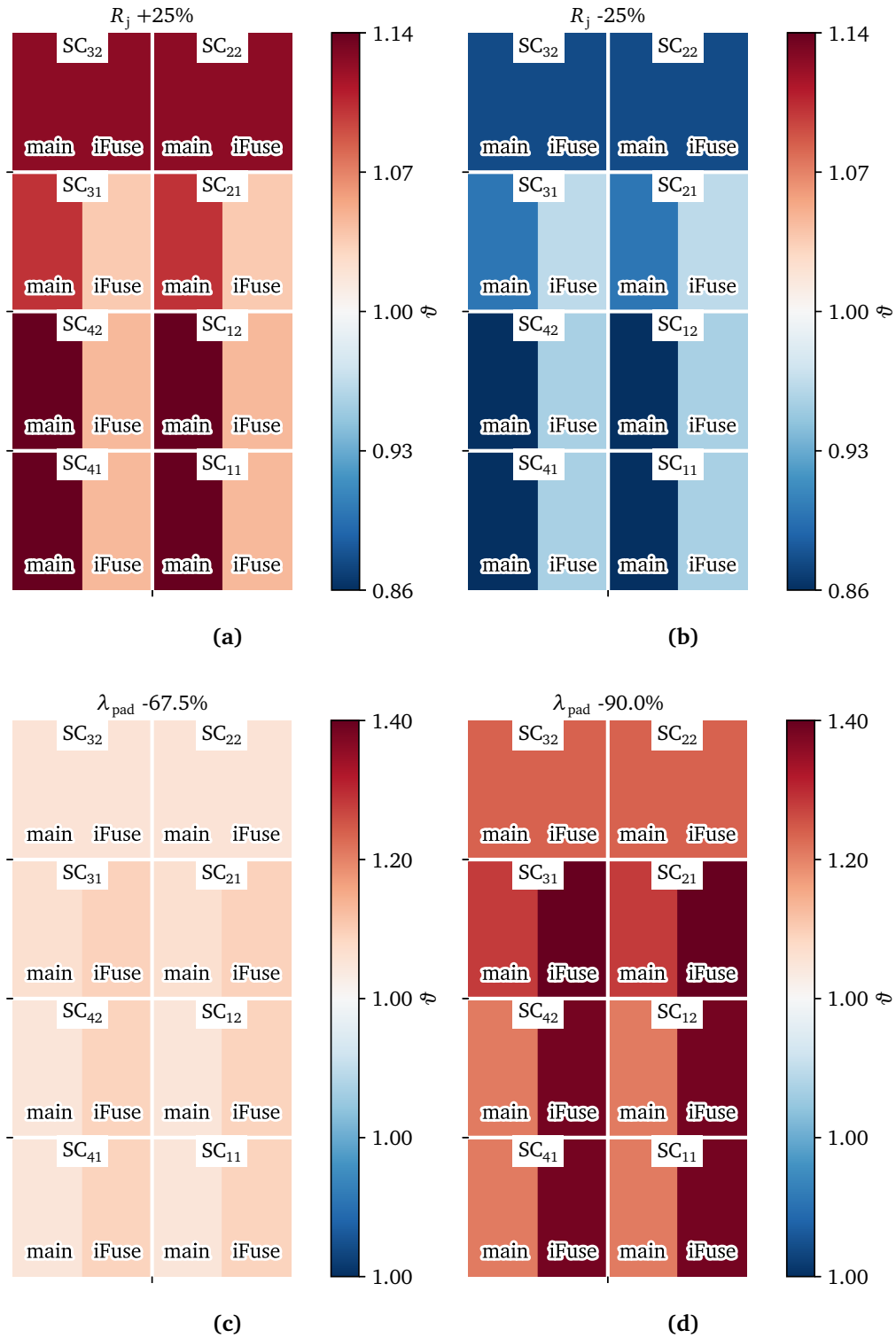


Figure 3.19. Values of dimensionless temperature difference in the power converter, obtained from the sensitivity analyses performed on R_j (a, b) and λ_{pad} (c, d).

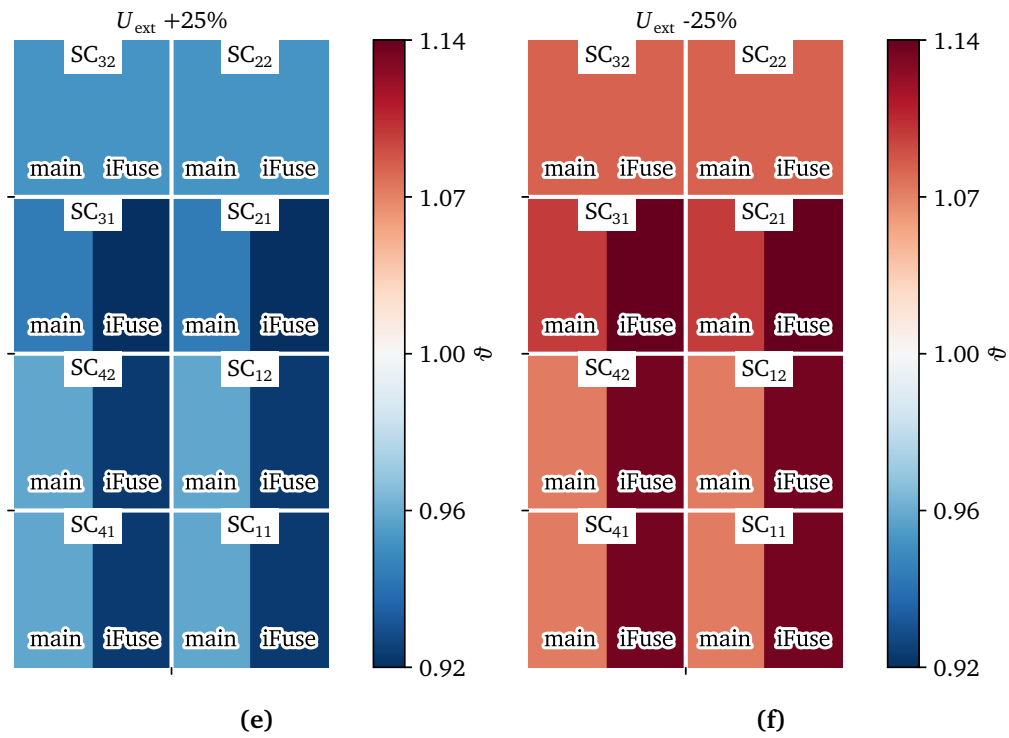


Figure 3.19. Values of dimensionless temperature difference in the power converter, obtained from the sensitivity analyses performed on U_{ext} (e, f).

3.4. Results and discussion

3.4.1. Validation of the 2D motor thermal model

The thermal modelling approach developed in this work is validated by comparing numerical results from the two-dimensional thermal model with those derived from a Finite Element Analysis performed on exactly the same motor geometry, which is presented in [192]. In this instance, validation is performed with respect to numerical data as a prototype of this motor and power converter assembly does not exist yet. The boundary conditions related to the reference FEA are reproduced by considering a coolant temperature of 50 °C, a phase current of 355 A, and an equivalent heat transfer coefficient of 3000 W/(m²°C) in the cooling jacket. In this case, due to some limitations of the computational tool used for the FE analysis, the stator and rotor are thermally uncoupled. Figure 3.20 shows the comparison between the temperature values obtained from the 2D LPTN and FE models in the windings and in the stator magnetic circuit, showing a very good agreement between the two models. This confirms the validity of the modelling approach selected for the electric motor. Furthermore, it demonstrates that a model consisting of a few thermal resistances can produce the same results as models with a much larger number of degrees of freedom, for this particular motor. The accurate results observed for the 2D LPTN can be used to partially validate both of the 3D approaches as they share the same basic arrangement of nodes and thermal resistances.

3.4.2. Cooling jacket

Figure 3.21 shows colourmaps of dimensionless velocity magnitude and temperature difference $\Delta T^* = (\bar{T}_{\text{wall}} - T)/(\bar{T}_{\text{wall}} - \bar{T}_b)$, that are obtained on centred transversal sections of the cooling jacket. As expected, the velocity profile is not symmetric with respect to the channel centreline due to the effect of centrifugal forces. The heat transfer effectiveness related to the internal and external walls is evaluated by means of an equivalent Nusselt number, which is computed as:

$$\text{Nu} = \frac{q'' y}{\lambda(\bar{T}_{\text{wall}} - \bar{T}_b)}, \quad (3.21)$$

where $\bar{T}_{\text{wall}}$ and $\bar{T}_b$ are the mean wall and fluid bulk temperatures, respectively. y is the channel height, which is chosen as the reference length of the problem. In the case of a flow of a 50% by mass solution of water and ethylene glycol at 5 l/min, the Nusselt number values related to the inner and outer walls are 22.4 and 17.4, respectively.

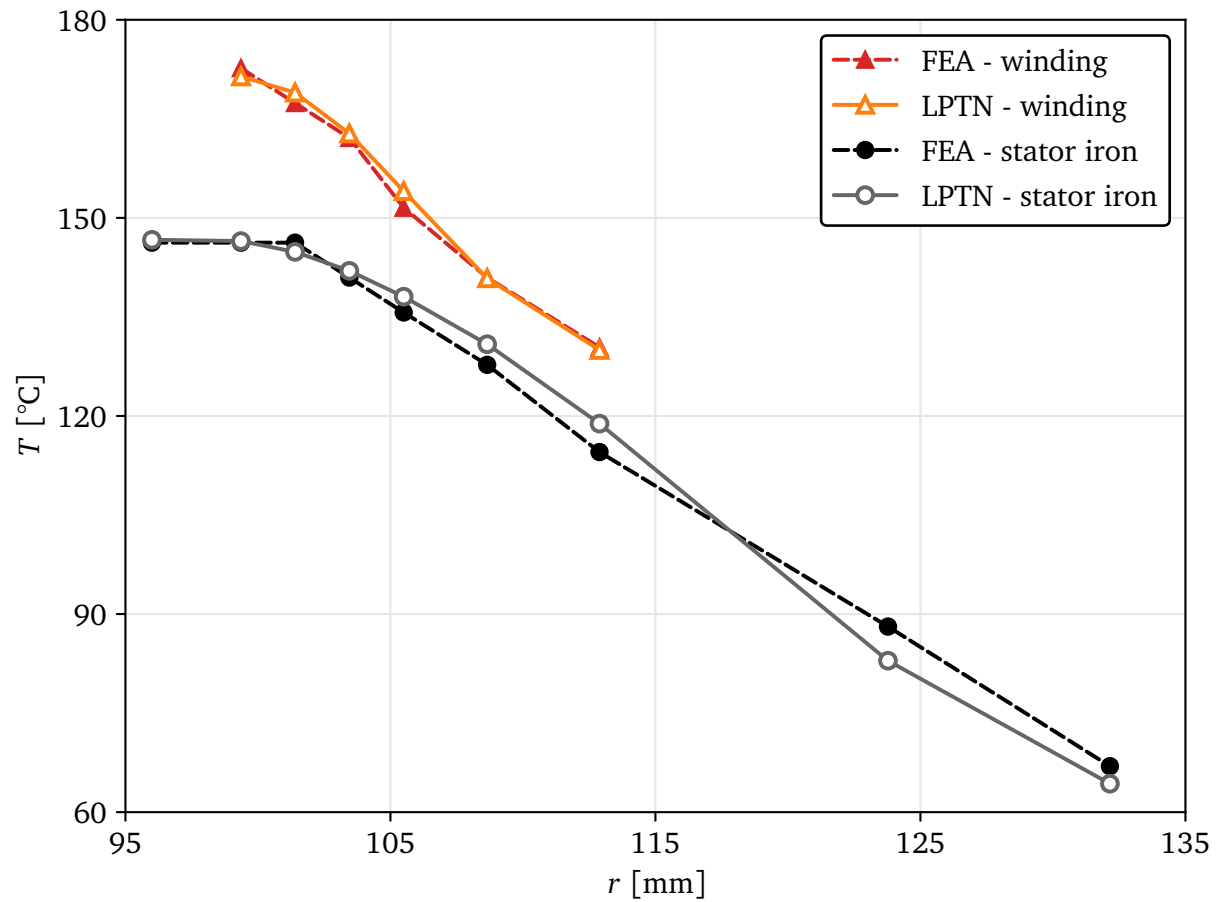


Figure 3.20. Comparison between the temperature values of the windings and stator iron obtained from the LPTN and FE models.

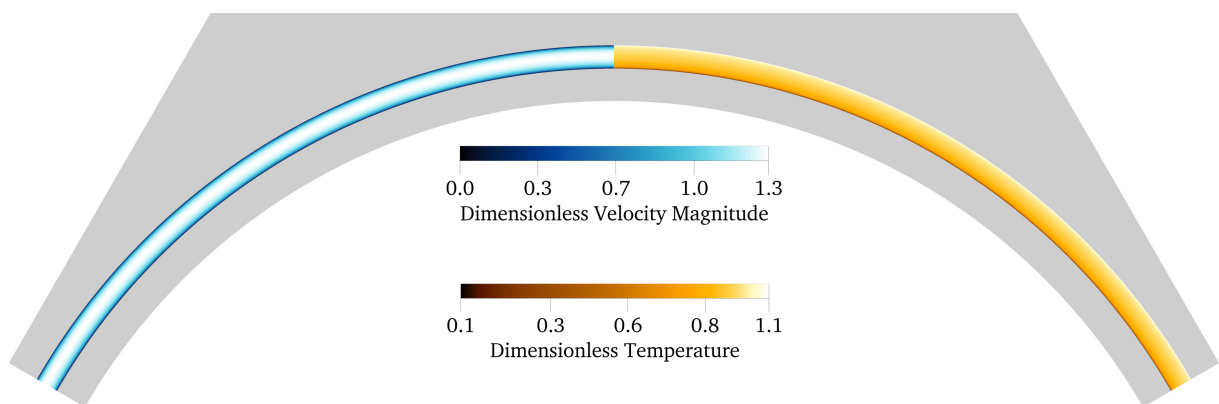


Figure 3.21. Colourmaps of dimensionless velocity magnitude and temperature difference on a transversal cross-section of the cooling jacket.

3.4.3. System level results

The system-level thermal models developed for the examined IMD, namely the 2D, 3D single-slot, and full 3D LPTNs are used to assess the feasibility of the proposed architecture from the thermal point of view. To this end, the maximum temperatures of each component, derived from the numerical results from the three models, are compared with their respective thermal limit, as shown in Table 3.8. The effectiveness of the proposed cooling system appears adequate to guarantee the correct and safe operation of the motor-converter assembly, according to the results from all three models. However, substantial differences in terms of hot-spot temperatures are noted between the three approaches, particularly between the 2D and both 3D LPTNs. On average, the $T_{\max}$ -values derived from the 2D LPTN are considerably lower than their counterparts from the 3D models. This is a somewhat expected behaviour since the 2D approach does not account for Joule effect losses in the end winding conductors. In addition, the assumption of a uniform coolant bulk temperatures appears to yield lower overall temperatures in the rotor. The temperature values associated with the power converter are the least sensitive to the type of approach, as the converter board are placed towards the front of the motor where the coolant inlet is located. As a result, the switching cells are less sensitive with respect to the coolant temperature increase along the axial direction.

The two 3D approaches show comparatively similar results as both models consider the full amount of losses associated with the assembly. These results also confirm the validity of the approach based on the parallel rule for the extension of the single-slot 3D model.

Figure 3.22 shows the temperature distributions of all six winding layers along the axial direction, which is denoted by z . In the case of the complete 3D LPTN this distribution is obtained by averaging winding temperatures along the circumferential direction, in others to make a fair comparison with the outcomes of the single-slot 3D approach. As previously stated, both models produce similar hot-spot winding temperatures in correspondence of the first end winding layer at the back of the motor.

Figure 3.23 reports a contour plot of the temperature distribution of stator and rotor on a transversal cross-section at the centreline of the motor. The effects of the increasing coolant temperature along the circumferential direction can be noticed, particularly in the rotor, where the region of high temperature near the surface tends to be more noticeable in the angle span $\pi/2 \leq \varphi \leq 3\pi/2$. On the other hand, in the stator, circumferential conduction allows for the redistribution of temperatures across the slots leading to a much more uniform temperature distribution.

In conclusion, the 3D single-slot thermal model represents the best compromise between complexity and accuracy for the computation of hot-spot temperatures as it constitutes a direct improvement over the 2D LPTN, while doing away with development and computational effort required by the complete 3D model. On the other hand, if a more faithful estimation of the temperature distribution is sought, the 3D LPTN should be used instead.

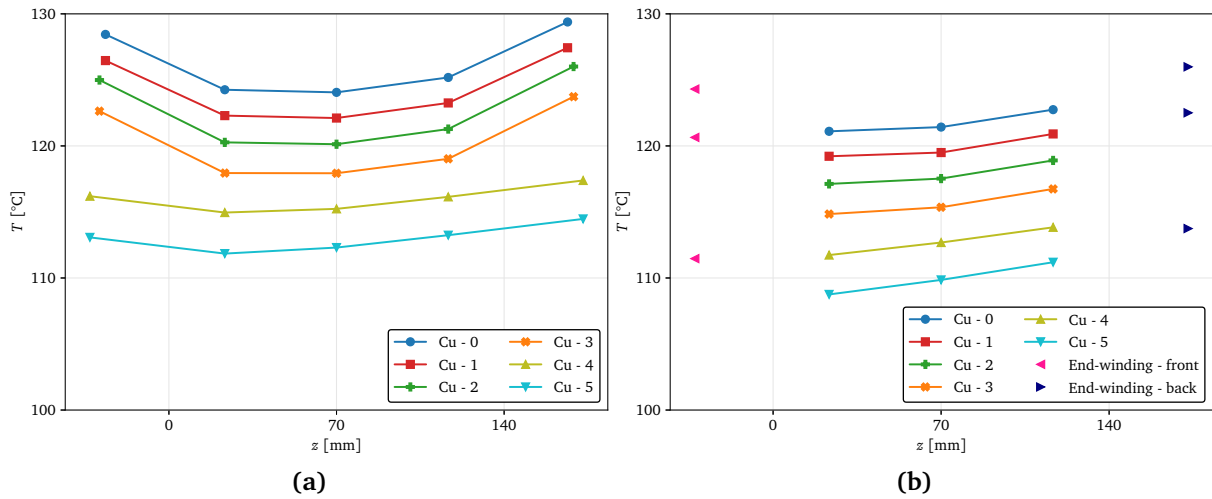


Figure 3.22. Distributions of winding temperatures along axis of the motor, obtained from the 3D single-slot model (a) and complete 3D LPTN (b)

Still, the 2D approach represents a very simple approach to obtain rapid and fairly accurate evaluations on the thermal performance of the machine, especially for the magnets and power converter, while moderate deviations should be expected for the windings.

3.5. Concluding remarks

This Chapter covered the thermal analysis of an integrated motor drive composed by a permanent magnet synchronous motor and a switching cell array power converter. This study was carried out by developing three LPTN thermal models: a basic two-dimensional thermal network, a simplified three-dimensional thermal model, and a complete 3D thermal network of the integrated assembly. These models were based on nominal geometric pa-

Table 3.8. Temperature limits and numerically obtained maximum temperature values of the main components of the integrated motor drive.

Model	Component	Temperature limit [°C]	$T_{\max}$ [°C]	$T_{\max} - T_{\text{ref}}$ [°C]
2D	Windings	180	115	46
	Magnets	310	183	114
	Converter	200	115	46
3D single-slot	Windings	180	129	69
	Magnets	310	192	132
	Converter	200	118	58
3D complete	Windings	180	127	67
	Magnets	310	189	129
	Converter	200	118	58

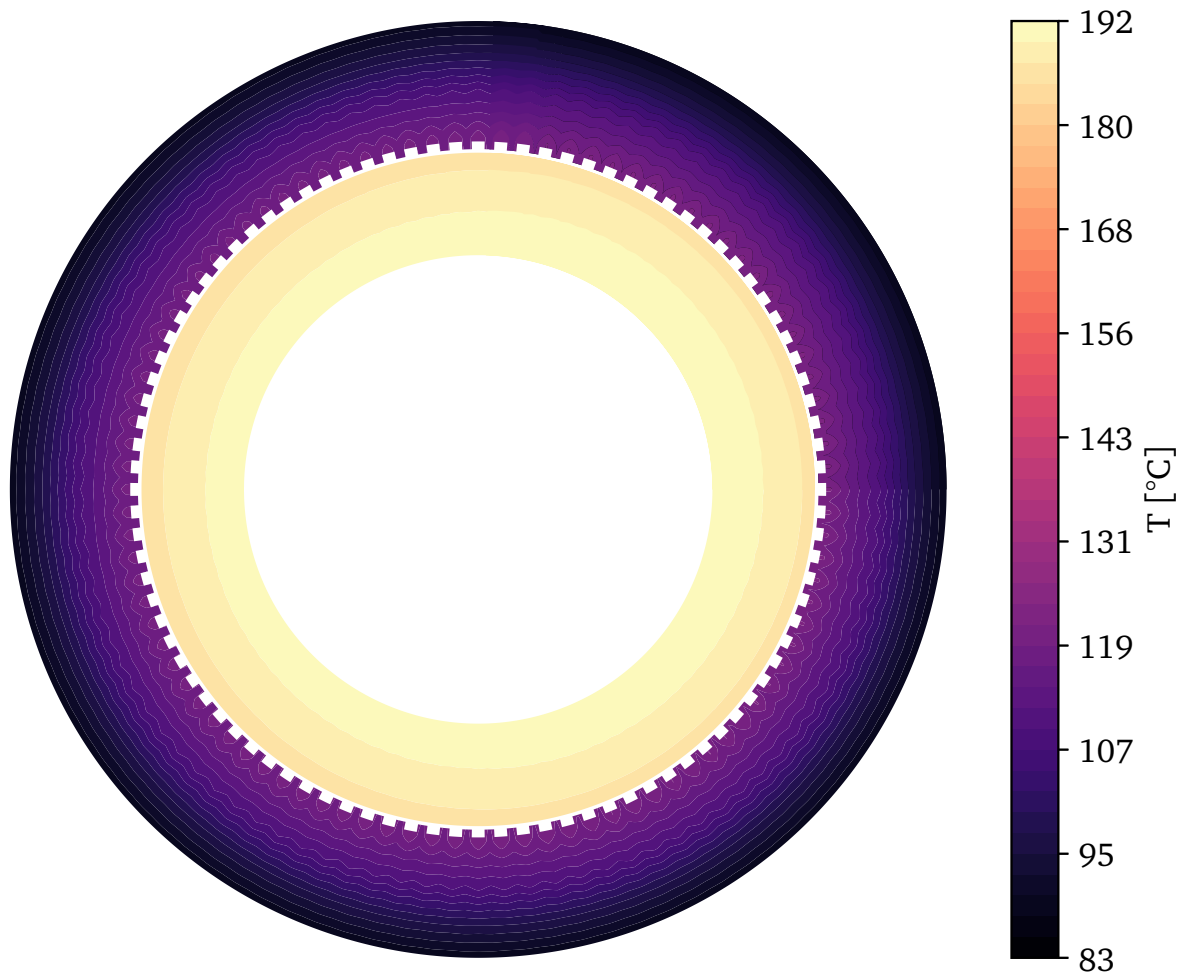


Figure 3.23. Temperature contour plot of the stator and rotor obtained on a cross-section located at the centre of the motor, considering numerical results from the complete 3D thermal model.

rameters, material properties and numerical results from CFD and FE models. Specifically, the heat transfer coefficients in the cooling jacket were obtained by developing a dedicated CFD model of a sixth of one of the cooling channels, while numerical results from previous FE analyses were used for an accurate description of the losses produced in the electric motor.

A comprehensive sensitivity analysis was performed to identify the most relevant parameters constituting the thermal models and to investigate the changes in thermal performance given by changing the constitutive materials of the integrated assembly. The most relevant parameters for the thermal management of the motor were the heat transfer coefficient associated with coolant flow, the thermal conductivity of the iron laminations, the interface gap between stator and case, and the heat transfer coefficients in the air gap. These findings are in agreement with existing sensitivity analyses found in the literature [170, 172]. Minor reductions of the hot-spot temperature in the windings were observed by considering insulating enamels and resins with improved thermal conductivity. Comparatively, smaller

improvements were found for insulating paper with better thermal properties. However, in this case, the larger temperature reductions were not observed at the hot-spot, but near the conductors close to the base of the slot, which are typically the coolest points in the winding assembly.

Finally, the developed thermal models were used to assess the feasibility of the proposed integrated architecture from the thermal point of view. As expected, the 2D LPTN provided the lowest hot-spot temperatures due to the absence of end winding losses and the assumption of a uniform coolant temperature throughout the whole cooling jacket. Both the 3D thermal models provided very close estimates for the hot-spot temperatures in the windings, rotor, and power converter. The simplified three-dimensional LPTN seems to represent the best trade-off between computational effort and accuracy regarding the computation of the hot-spots.

Physics-based modelling of plate-fin heat exchangers

4.1. Introduction

4.1.1. Motivations and aims

Aluminium Plate-Fin Heat Exchangers (PFHEs), also referred to as plate and bar heat exchangers, are widely used in the automotive, agricultural, and power generation industries, where reliability is key. The stacked assembly of finned plates and sidebars endow these heat exchangers with remarkable mechanical resistance and heat transfer properties. In spite of their widespread diffusion, both the design and verification processes of PFHEs still rely largely on rules of thumb and expensive testing campaigns. With the advent of the recent and repeated energy and financial crises, it is crucial for manufacturers to adopt suitable prediction tools in order to reduce the need for experimental tests and, in turn, contain design costs. Nevertheless, such tools must ensure a satisfactory accuracy in the estimation of both fluid dynamic and heat transfer performance, while maintaining a high degree of generality. However, most of the available models are based on parameter identification from experimental tests and, as such, suffer from two major drawbacks: (i) the generality of these models is quite limited; (ii) the number of tests required to achieve reliable predictions is still high.

In this work, a simple but accurate numerical methodology is presented, aimed at the prediction of overall head losses and heat transfer rates in PFHEs. To maintain a high degree of generality, the present methodology is developed considering only nominal geometric parameters, physical principles, and established models, without resorting to the identification of any empirical parameter.

NOMENCLATURE*Latin letters*

a	wave amplitude [m]
A	area [m^2]
b_i	coefficients in Equation (4.19)
c_i	coefficients in Equation (4.20)
c_p	specific heat [$\text{J kg}^{-1} \text{K}^{-1}$]
C	heat capacity rate [W K^{-1}]
d_{kr}	coefficients in Equation (4.32)
D_h	hydraulic diameter [m]
e_{kr}	coefficients in Equation (4.33)
f	equivalent Darcy friction factor [—]
h	enthalpy [$\text{kg m}^2 \text{s}^{-2}$]
H	heat exchanger height [m]
j	Colburn factor [—]
k_n	coefficients in Equation (4.35)
l	offset length and half the wavelength [m]
L	heat exchanger core length [m]
L_e	effective wavy fin length [m]
$\vec{n}$	normal unit vector [—]
N_{ch}	number of internal channels [—]
Nu	Nusselt number [—]
NTU	number of transfer units [—]
p	fluid pressure [Pa]
P	wetted perimeter [m]
Pr	Prandtl number [—]
q_{ij}	coefficients in Equation (4.36)
$\vec{q}$	heat flux [W m^{-2}]
$\dot{Q}$	heat [W]
R	thermal resistance [K W^{-1}]
Re	Reynolds number [—]
S	heat exchanger thickness [m]
t	fin thickness [m]
T	temperature [K]
U	overall heat transfer coefficient [$\text{W m}^{-2} \text{K}^{-1}$]

 $\vec{v}$ fluid velocity [m s^{-1}] W manifold width [m] x fin pitch [m] y fin height [m]*Greek letters*

β	concentrated loss coefficient [—]
ε	effectiveness [—]
ϑ	dimensionless temperature [—]
λ	thermal conductivity [$\text{W m}^{-1} \text{K}^{-1}$]
μ	dynamic viscosity [Pa s]
ν	kinematic viscosity [$\text{m}^2 \text{s}^{-1}$]
ρ	density [kg m^{-3}]
σ_m	momentum sink [m s^{-2}]
Φ	relative friction factor in the entrance region [—]
χ	single fin module number [—]
Ψ	relative Nusselt number in the entrance region [—]

Subscripts

b	bulk
ch	channel
e	entrance
exp	experimental
ext	external
F	Fanning
fd	fully developed
i	inlet
int	internal
m	model
o	outlet
s	solid
t	turbulent
w	wall

Superscripts

*	dimensionless
---	---------------

The study presented in the following was carried out in collaboration with the company VEMA industries S.r.l., which manufactures custom plate-fin heat exchangers for a variety of industrial applications including agricultural and construction vehicles.

4.1.2. Literature survey

Extended surfaces are a key component of any heat exchanger, as they allow for a substantial enhancement of heat transfer effectiveness compared to standard rectangular channels. The most commonly used extended surfaces in plate-fin exchangers often feature offset-strip and wavy or corrugated geometries. Offset-Strip Fins (OSFs) are characterized by high compactness, and their outstanding heat transfer performance is achieved through periodic interruptions of the boundary layers and the oscillating flow patterns present in the wakes of the fins. Over the years, OSFs have been the subject of several studies based on both experimental and numerical methods. Kays and London [202] and London and Shah [203] first provided a large set of experimental results on actual fin cores. Many authors, including Wieting [204], Mochizuki *et al.* [205], Manglik and Bergles [206], and Dubrovsky *et al.* [207], proposed empirical correlations for the evaluation of friction losses and heat transfer effectiveness in OSFs, which are based on the regression analysis of experimental results. Joshi and Webb [208] presented an analytical model for the laminar flow regime and a semi empirical approach for the turbulent one. All the aforementioned correlations were presented in terms of Colburn factor and Fanning friction factor, as a function of Reynolds and Prandtl numbers, and dimensionless geometric parameters. Numerical methods, improved by the rapid growth of computational capabilities over the last few decades, have been largely exploited to investigate the thermal performance of OSFs at a reduced cost. Recent studies employed 3D simulations with the standard Finite Volume approach and several turbulence models [209–211], while the Lattice Boltzmann method is employed for the study of extended surfaces in micro heat exchangers [212]. Kim *et al.* [213] investigated the effect of the turbulence model on numerical results, including the k - ω , the standard k - ϵ , the realizable k - ϵ , and the k - ω SST. The authors showed that numerical results obtained by means of the k - ω SST turbulence model are the ones in better agreement with the experimental results presented in [206].

Wavy Fins (WFs) are also used extensively in plate-fin heat exchangers, due to their superior resistance to clogging [121] in harsh environments, compared to offset-strip fins. The improvement in the thermal-hydraulic performance of wavy fins compared to plain ones is given by two main factors: at low Reynolds numbers ($Re < 100$) the dominant effect is that of an increased effective flow length, while at high Re -values the onset of streamwise and spanwise counter rotating lateral vortices in the troughs of the fins lead to higher momentum transfer, along with a thinning of the thermal boundary layer [214]. The geometry of wavy fins is defined on the basis of the fin pitch, x , fin height, y , wave length, $2l$, fin thickness, t , and wave amplitude, a . In most of the studies concerning wavy fins, the performances are

expressed in terms of isothermal Fanning friction factor and Colburn factor, as a function of the Reynolds number and of the three dimensionless parameters: aspect ratio, fin spacing ratio, and corrugation ratio. Several studies have addressed the thermal-hydraulic characterization of wavy fins either experimentally or numerically. The earliest experimental data can be found in [202] for three different geometries using air as the working fluid. Goldstein and Sparrow [215] studied corrugated channels with triangular waves by carrying out experiments based on the naphthalene sublimation method. Rush *et al.* [216] conducted flow visualization tests to investigate fluid behaviour in the laminar and transitional flow regions. More recent experimental studies are found in [121, 217]. Compared to experimental studies, numerical research on wavy fins is much sparser. Asako and Faghri [218] numerically investigated two-dimensional steady laminar flow in plate channels with triangular-profiled wall corrugations. Zhang *et al.* [219] and Metwally and Manglik [220] investigated two-dimensional laminar periodic flow in sinusoidal wavy channels with different corrugation angles and fin spacing values. Three-dimensional computational studies include those of Aliabadi *et al.* [221], Wen *et al.* [210], Tao *et al.* [222], and Pham *et al.* [223].

Global modelling approaches for PFHEs can be found in the literature. In [224–227], the authors carried out design optimizations of plate-fin heat exchangers by means of various algorithms. In all these works, the PFHE cores featured offset-strip extended surfaces in both the internal and external channels. The ε – NTU method was adopted to study the thermal performance. In the finned channels, head losses were computed by means of the Darcy-Weisbach formula. The values of friction factor and overall heat transfer coefficient were obtained by means of the correlations proposed by Manglik and Bergles [206]. Hao *et al.* [228] optimised the geometry of a compact heat exchanger equipped with offset-strip fins by means of the tree traversal method. The authors evaluated overall heat transfer performance with the heat current method, which allows one to obtain an equivalent thermal resistance based on the Inlet Temperature Difference (ITD) [229–231]. This approach does away with the tedious iterative procedure that comes with the LMTD-based resistance, and, with respect to the ε – NTU method, it is better suited for the analysis of the intricate networks of heat exchangers. Also in this case, the authors employed the correlations found in [206], obtaining a good agreement with experimental data.

With respect to the current approach, the models presented in the above referenced works certainly have similar advantages in terms of ease of implementation and use, but they also feature shortcomings that may affect their accuracy. As will be shown later, the use of correlations from the literature can lead to significant deviations in the estimation of the thermal-hydraulic performance for particular geometries and operating fluids. Furthermore, the head losses produced by both manifolds and the influence of entrance effects on the overall heat transfer rates are two relevant aspects that are mostly neglected in the previously cited works.

4.1.3. Modelling approach

In light of the above considerations, in the present work an original multiscale approach is proposed, compounding detailed 3D analyses at fin scale with global models at system scale, with the aim of obtaining accurate estimates of heat transfer rates and head losses of plate-fin heat exchangers, with reduced model complexity and at a reasonable overall computational cost. The proposed approach is specifically targeted towards engineering needs and is readily reproducible for a wide range of heat exchanger applications.

Figure 4.2a shows a simple diagram depicting the steps of the adopted modelling strategy, which are briefly introduced and discussed in the following:

- i. The analysis starts with the characterization of the finned surfaces. The periodicity of these surfaces along the streamwise and spanwise directions is exploited to reduce the analysis of the entire channels to the study of fundamental units, under fully developed flow conditions. A detailed 3D CFD model is constructed for each different fin type, and a parametric analysis is carried out for different working fluids and several flow regimes, with the aim of deriving more general correlations describing their heat transfer and friction characteristics in terms of an overall Nusselt number, Nu , and an equivalent Darcy friction factor, f , respectively.
- ii. Additional dedicated CFD analyses are also carried out to take into account entrance effects for both the Nusselt number and the equivalent friction factor. As previously discussed, these effects are seldom taken into account in common heat exchanger analysis, but they appear instead to be relevant, especially for the external channels of cross-flow plate-fin heat exchangers. In the present approach, the contribution of the developing flow in the entry region is accounted for by means of generalized correlations for suitable correction factors to both Nu and f , which are again derived from CFD results.
- iii. Another aspect which is often overlooked in the assessment of heat exchanger performance is the evaluation of the pressure drop in the inlet and outlet manifolds, which, as shown in the following, can account for more than 90% of the total head losses of the internal side. The proposed methodology encompasses the use of CFD, coupled with an equivalent porosity model, in order to calculate equivalent concentrated loss coefficients for the manifolds, and therefore provide a more accurate estimate of the head losses of the internal side.
- iv. On the basis of the results of the previous steps, global models are developed to compute the heat rejection and mechanical energy losses of the complete heat exchanger. The global model for heat transfer is based on the classical $\varepsilon - NTU$ method [202], where the overall heat transfer coefficients are obtained by combining the correlations for the Nusselt number in fully developed flow—step (i) above—with the cor-

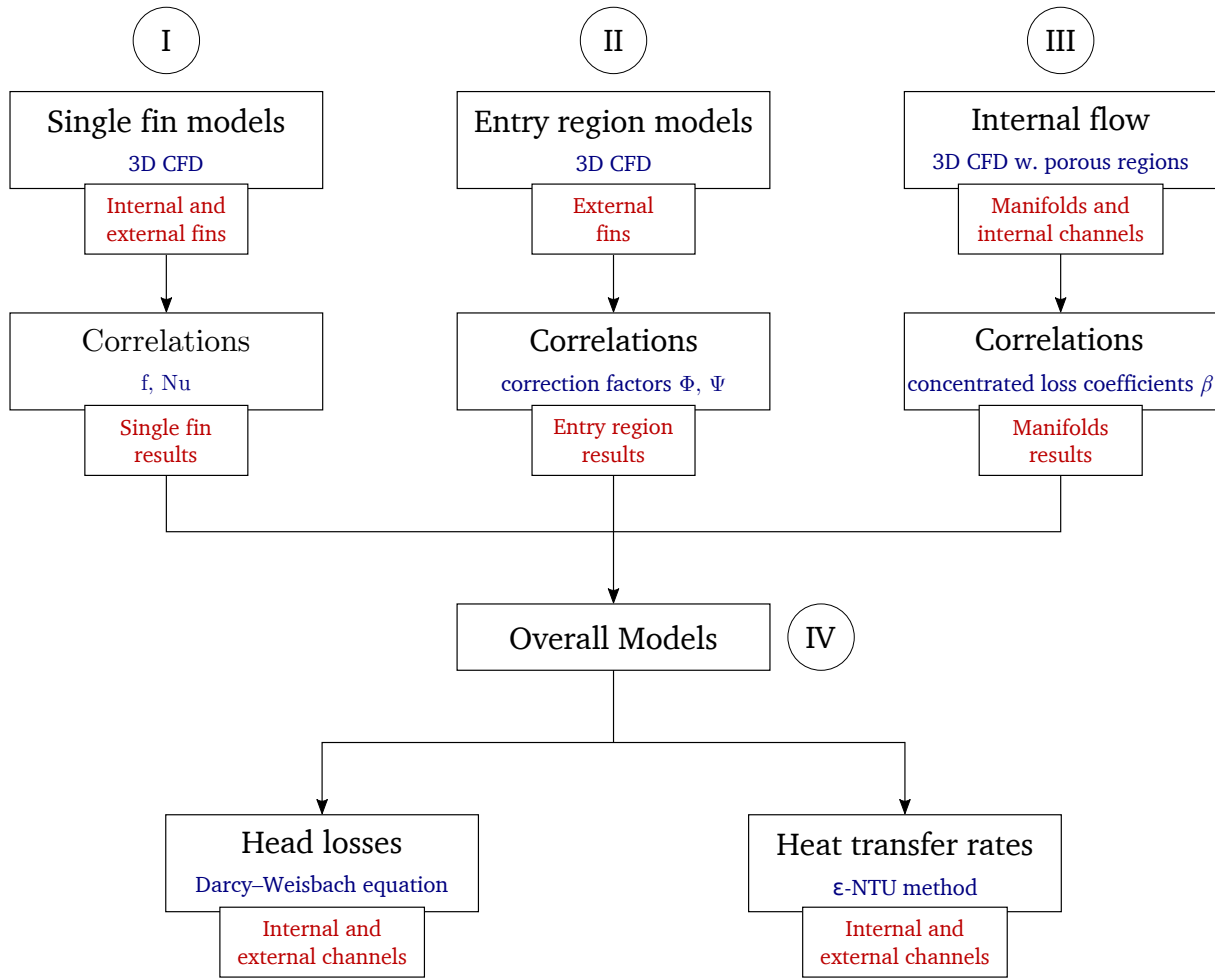


Figure 4.1. Simple schematic describing the modelling strategy adopted for the development of a high fidelity virtual model for PFHE heat exchangers.

rection factors for the entry region—step (ii)—where appropriate. Head losses are instead computed using the empirical relations for friction factor derived in step (i), the correction factors for the entry region in the internal channels—step (ii)—and the concentrated loss coefficients for the inlet and outlet manifolds—step (iii).

In the following, the above-described detailed computational models and global models for heat transfer and head losses are introduced and described in detail, using a selected set of PFHE configurations as the case study. Friction factors and Nusselt numbers obtained from the numerical analyses are also compared with the data presented by Joshi and Webb [208], Manglik and Bergles [206], Awad and Muzychka [232], and Dong *et al.* [217], to assess whether the correlations already available in the literature are suitable for predicting the performance of fins used in industrial applications. As will be shown in the following, the available correlations exhibit a fair agreement with most of the present results, but large uncertainties are present for several configurations of geometry and flow. Therefore, the present work, besides proposing a holistic approach being instrumental for a specific industrial use, provides additional insight on the research effort still needed to accurately

characterize a wide category of fin geometries of great applicative interest.

Finally, the proposed multiscale approach is validated by comparing global numerical results on head losses and heat transfer with available experimental data, for a considerable variety of heat exchanger configurations.

4.2. Case study

In the present work, a set of eight plate-fin heat exchangers is considered, consisting of three air-oil coolers, two water-glycol radiators, and three charge air coolers. All these heat exchangers feature manifolds with a simple box shape and main pipes with a round cross-section and a diameter of 35 mm. The height, width, and depth of the manifolds are denoted by H , W , and S , respectively. The core of each heat exchanger is composed of N_{ch} internal channels and $N_{\text{ch}} + 1$ external ducts. The length of the core is identified by L . Figure 4.2a depicts a schematic showing the geometric features of the heat exchangers examined, along with some relevant dimensions, while Figure 4.2b depicts a picture of one of the actual heat exchangers considered. Table 5.7 reports the geometric characteristics of each heat exchanger, along with an identification code for future reference. The letter stands for the type of internal fluid, specifically O for oil, G for a water-glycol solution, and A for compressed air, while the number indicates the thickness of the heat exchanger in millimetres.

The heat exchangers under consideration feature finned surfaces with offset-strip and wavy geometries. The oil and water coolers have internal channels equipped with offset-strip fins and present external channels with wavy fins. On the other hand, the air-to-air heat exchangers are equipped with wavy fins in both the internal and external channels. The OSF employed in the oil coolers, designated as OSF1, presents a centred offset, and its front and top views are represented in the drawing of Figure 4.3a. The fin OSF2 is used in water-glycol radiators and, in comparison to the OSF1, features a larger aspect ratio and offset length. Moreover, in OSF2 the offset is not symmetric. The geometric differences between OSF1 and OSF2 can be clearly seen by comparing Figures 4.3a and 4.3b. Oil coolers and air-to-air heat exchangers present external finned surfaces with a wavy geometry with a triangular cross-section. This fin is identified by the acronym WF1, and its front and top views are depicted in the drawing of Figure 4.3c, along with the key dimensions. The WF2 fin features the same triangular cross-section and axial development as WF1, but it has different values of pitch and height. The internal channels of the three charge air coolers present wavy fins with a rectangular cross-section, as illustrated in Figure 4.3d. This fin geometry is identified by the acronym WF3. WF4 is a wavy fin with a rectangular cross-section and sinusoidal development, as shown in Figure 4.3e. It is employed in the external channels of the two water-glycol radiators, designated as G94 and G113. The values of the geometric parameters of all the examined fins are presented in Table 4.2.

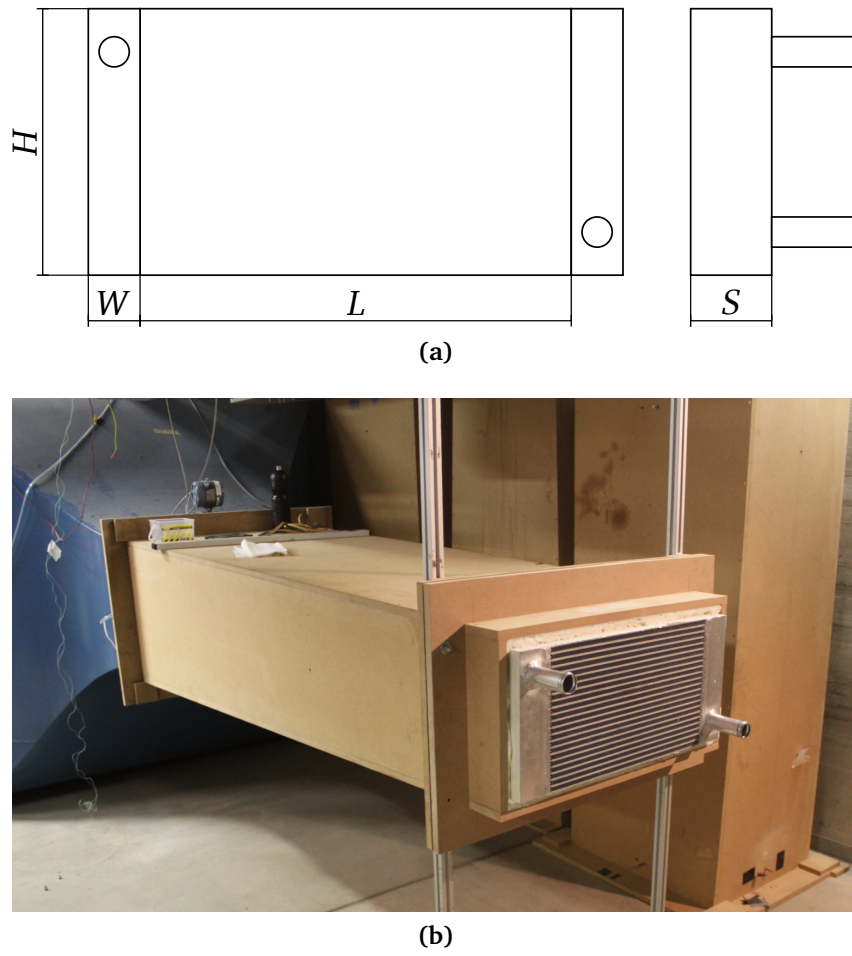


Figure 4.2. Sketch of the external geometry (a) and an example of plate-fin heat exchanger, provided by VEMA S.r.l. (b).

Table 4.1. Geometrical parameters of the plate-fin heat exchangers considered.

ID	S [mm]	H [mm]	W [mm]	L [mm]	N_{ch}	Int. fin	Ext. fin
O80	80	304	60	500	20	OSF1	WF1
O94	94	309	60	500	21	OSF1	WF1
O113	113	305	60	500	20	OSF1	WF1
G94	94	309	60	500	23	OSF2	WF4
G113	113	309	60	500	23	OSF2	WF4
A80	80	304	60	500	17	WF2	WF1
A94	94	304	60	500	17	WF3	WF1
A113	113	304	60	500	17	WF3	WF1

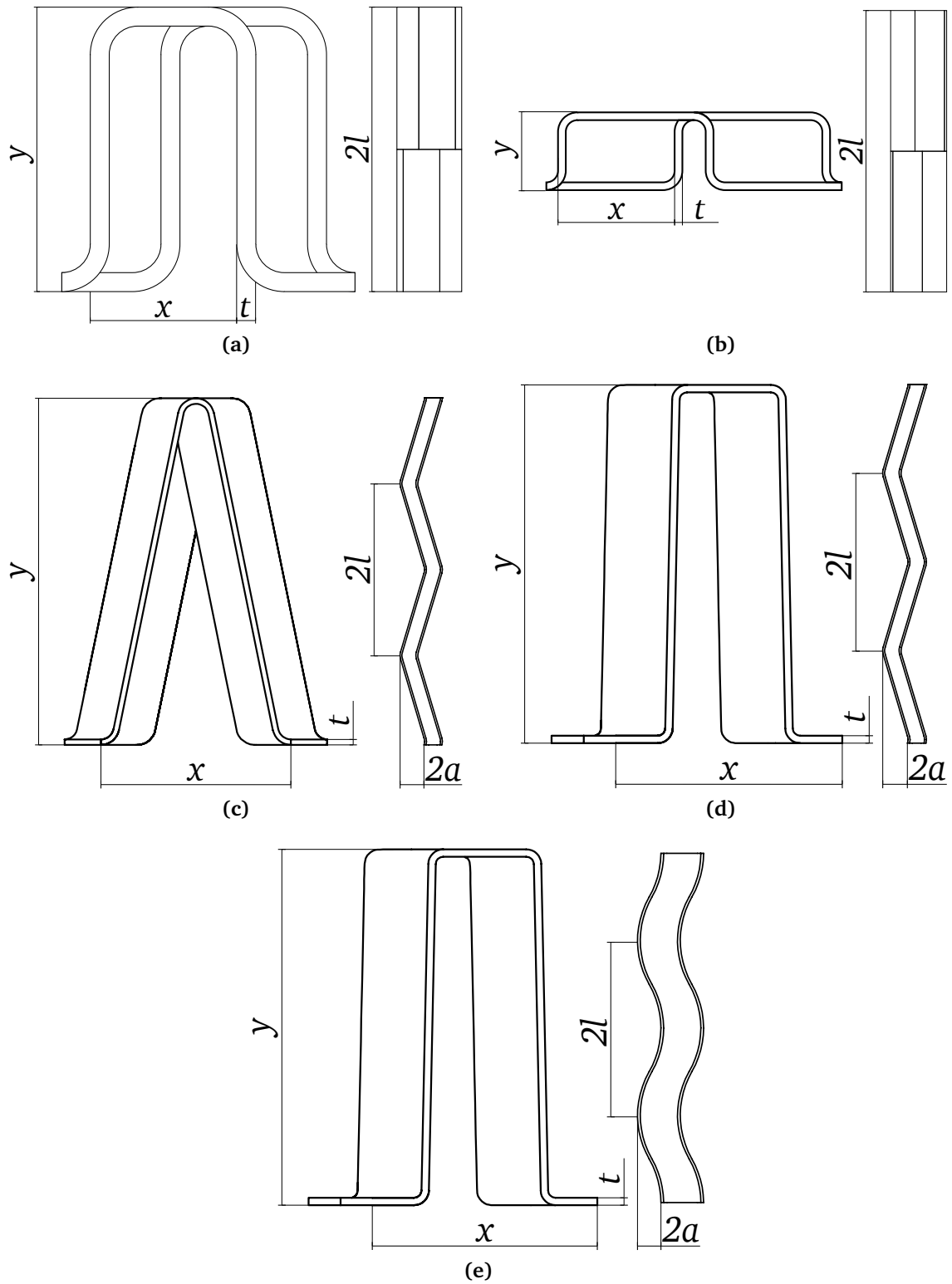


Figure 4.3. Front and top views of the fins: OSF1 (a), OSF2 (b), WF1 (c), WF2 (c), WF3 (d) and WF4 (e).

4.3. Numerical methods

4.3.1. Single fin model

4.3.1.1. Governing equations

Single fins are modelled in terms of their smallest unit geometry, which is repeated along the width and length of the channels. These basic modules are assumed to be located away from the inlet and outlet sections, so that the flow can be considered as fully developed. The flow is also assumed steady and incompressible for all the fluids and fins studied. In fluid regions, flow and temperature fields are determined by numerically solving momentum, continuity, and energy equations.

$$\begin{cases} \nabla \cdot (\vec{v} \otimes \vec{v}) = -\frac{1}{\rho} \nabla p + \nabla \cdot [(\nu + \nu_t) \nabla \vec{v}] \\ \nabla \cdot \vec{v} = 0 \\ \nabla \cdot (\rho \vec{v} h) + \nabla \cdot \left(\rho \vec{v} \frac{|\vec{v}|^2}{2} \right) = -\nabla \cdot \vec{q} + \nabla \cdot (\vec{\tau} \cdot \vec{v}) \end{cases} \quad (4.1)$$

Within the walls of the fins, temperature field is determined by solving the steady-state heat conduction equation.

$$\nabla \cdot (\lambda_s \nabla T) = 0 \quad (4.2)$$

Low Reynolds number flows are solved by performing laminar analyses, meaning that $\nu_t = 0$. This approach is maintained for increasing Re as long as a solution with a satisfactory degree of numerical convergence is reached. That is, when all residuals are of the order of 10^{-6} . The k - ω SST turbulence model is used for Reynolds number values that do not allow for a steady-state laminar solution. This turbulence model is selected among all the others as it is the one that leads to the best agreement between numerical and experimental results for offset-strip fins, according to Kim *et al.* [213].

To reduce the number of dimensional and flow parameters, the problem is reformulated in dimensionless form by scaling down the computational model by the reference length,

Table 4.2. Geometrical parameters of the fin geometries considered.

ID	x [mm]	y [mm]	l [mm]	a [mm]	t [mm]
OSF1	1.5	3.0	5.0	-	0.2
OSF2	4.0	2.0	30.0	-	0.2
WF1	5.1	9.5	7.0	1.0	0.2
WF2	5.0	6.5	7.0	1.0	0.2
WF3	5.0	6.5	7.0	0.6	0.2
WF4	6.0	9.5	6.35	0.85	0.2

chosen as the fin height y . The problem is then defined by the Reynolds and Prandtl numbers alone. The fluid average velocity along the streamwise direction is assumed unitary, and thermophysical properties in Equations (4.1) are scaled accordingly:

$$\mu^* = \nu^* = \frac{1}{\text{Re}_y}, \quad c_p^* = \text{Re}_y \text{Pr}. \quad (4.3)$$

Fins are made of 3003 aluminium alloy. The thermal conductivity of the solid walls is assumed constant with temperature and, in the adopted dimensionless framework, is obtained by scaling the thermal conductivity of aluminium by the one of the adjacent fluid.

$$\lambda_s^* = \frac{\lambda_{\text{alu}}}{\lambda_{\text{fluid}}} \quad (4.4)$$

4.3.1.2. Boundary conditions

To account for the periodicity of geometry and flow along the streamwise direction, field values at the inlet and outlet boundaries of the domain are linked by suitable mappings: At the inlet section, the velocity and temperature distributions are mapped from the outlet patch, keeping their integral average values fixed. At the inlet patch, a zero-gradient boundary condition is assigned to the pressure field. At the outlet patch, zero-gradient boundary conditions are applied to the velocity and temperature fields, while a uniform reference value is assigned to pressure. Periodic conditions are enforced along the spanwise direction. At the top and bottom boundaries, the temperature gradient is set to an arbitrary non-zero value, to impose an entering heat flux. At the walls, no-slip conditions are applied to the velocity field and a zero pressure gradient is imposed. Finally, continuity of temperature and heat flux is enforced at all fluid-solid interfaces. The boundary conditions described herein are summarised in Table 4.3.

4.3.1.3. Spatial discretisation

Numerical simulations are carried out by means of a Finite Volume conjugate heat transfer solver, based on the open source computational toolbox OpenFOAM 7 [196]. Convective

Table 4.3. Boundary conditions applied to the inlet, outlet, left and right boundaries, and top and bottom surfaces in fluid and solid regions of the single fin model.

Boundaries	inlet	outlet	left/right	top/bottom	fluid-solid interface
$\vec{v}$	mapped	zero-gradient	periodic	no-slip	no-slip
fluid p	zero-gradient	fixed mean	periodic	zero-gradient	zero-gradient
T	mapped	zero-gradient	periodic	fixed gradient	heat flux continuity
solid T	zero-gradient	zero-gradient	periodic	fixed gradient	heat flux continuity

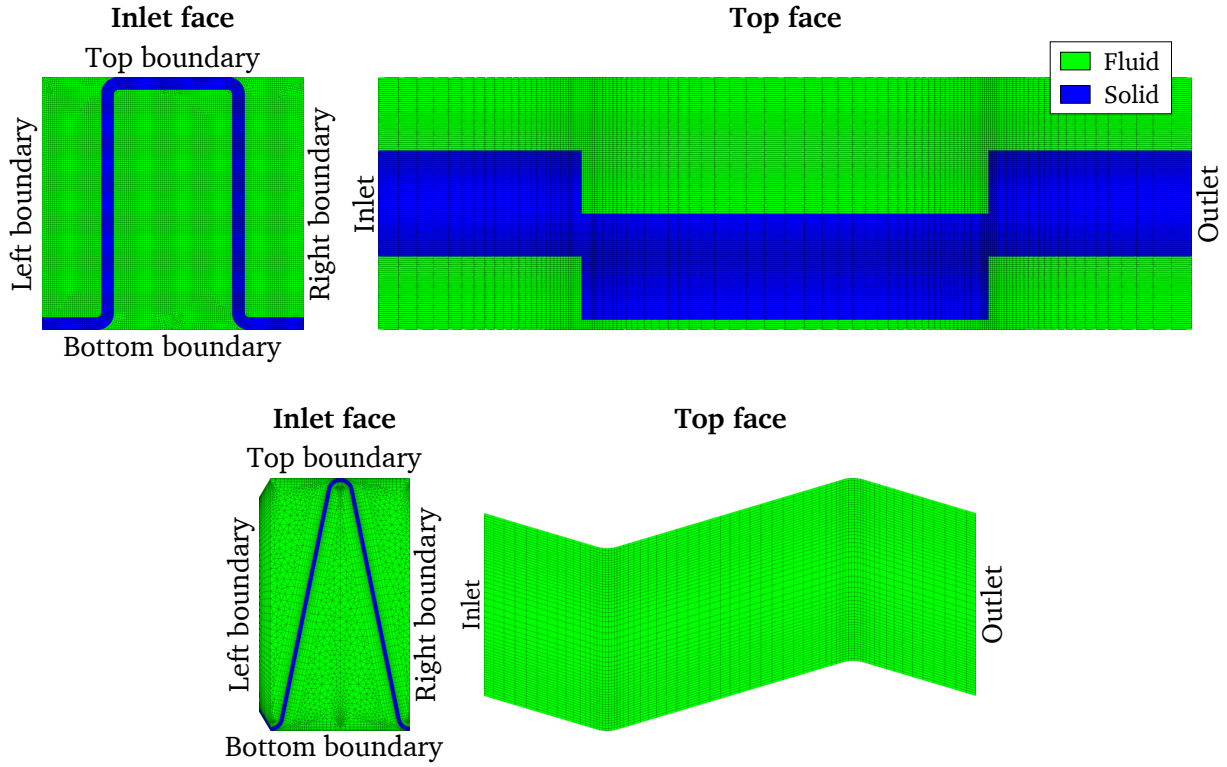


Figure 4.4. Selected cross-sections of the two computational meshes: OSF1 (top) and WF1 (bottom).

terms are discretised using upwind schemes of second-order accuracy, while diffusive terms are addressed with central schemes presenting explicit non-orthogonality corrections. A variant of the SIMPLE algorithm [233] is employed to decouple the pressure and velocity fields. Depending on the geometry, the dimensionless single-fin modules are discretised with block-structured and hybrid computational grids, using the commercial tool Pointwise. In hybrid meshes, boundary layers are obtained by means of an advancing front algorithm, starting from a 2D mesh produced by using Delaunay triangulation. In both hybrid and structured grids, fluid and solid regions are extruded as separate blocks to allow for the convenient set up of multi-region simulations using OpenFOAM's built-in tools. Figure 4.4 shows the top and front faces of the computational grids for two fin configurations, OSF1 and WF1.

4.3.1.4. Mesh sensitivity analysis

Grid independence of single-fin models is investigated by comparing the values of friction factor obtained from three computational meshes of the WF1 fin, which are developed by halving the number of the base mesh elements each time, maintaining the same topology. Table 4.4 lists the f -values derived from the three computational grids at $Re_y = 10000$, along with the average cell size in the far field Δ^* , the near-wall element size Δ_w^* , and the number of elements of each computational grid. The obtained results are very close, as the difference between the medium and the coarse values is 0.9%, while the one between fine

and medium values is 0.01%. The fine mesh is selected for all the following CFD analyses as it requires a lower number of iterations for the solution to converge, with respect to the other grids. An analogous procedure was adopted for all the fin geometries considered, leading to similar results.

4.3.2. Numerical model for entrance effects

4.3.2.1. Governing equations

The fully developed flow hypothesis can be considered valid for internal channels, for which entrance effects are negligible given the great ratio between the length of the channels and that of an elementary module. The external channels are much shorter; therefore, this hypothesis cannot be assumed valid *a priori*. The effects of flow development in the entry region on the thermal-hydraulic performance of the external finned surfaces are determined by means of dedicated CFD models comprising thirty single-fin modules in series, with the main aim of deriving the values of the Darcy friction factor and the Nusselt number of each module. The model also includes inlet and outlet tanks, and the analysis is performed on both of the external fins considered, WF1 and WF4. Similarly to the fully developed flow analyses, the flow is assumed steady and incompressible. Furthermore, the temperature and flow fields are obtained from the numerical solution of Equation (4.1), and by addressing the problem by the dimensionless framework described in Section 4.3.1. The governing equations are solved using the same algorithm and spatial discretisation schemes as those employed for single-fin models. In this case, as the analyses are performed considering air flows at medium to high Re-values, the k - ω SST turbulence model [195] is used for all the tested flow rates.

The main difference between the numerical models developed for fully developed flow and entrance region analyses, aside from the computational meshes, consists of the boundary conditions applied along the streamwise direction.

4.3.2.2. Boundary conditions

At the inlet section, fluid velocity is set to a uniform value, to impose a unitary mean streamwise velocity within the fin stack. The value of fluid velocity at the inlet is derived by ap-

Table 4.4. Friction factor values obtained from the coarse, medium, and fine meshes of WF1.

Grid	Δ^*	Δ_w^*	Number of cells	f	Relative deviation
Coarse	0.025	0.008	90×10^3	0.3982	-
Medium	0.020	0.004	180×10^3	0.4018	0.9%
Fine	0.016	0.002	360×10^3	0.4017	-0.01%

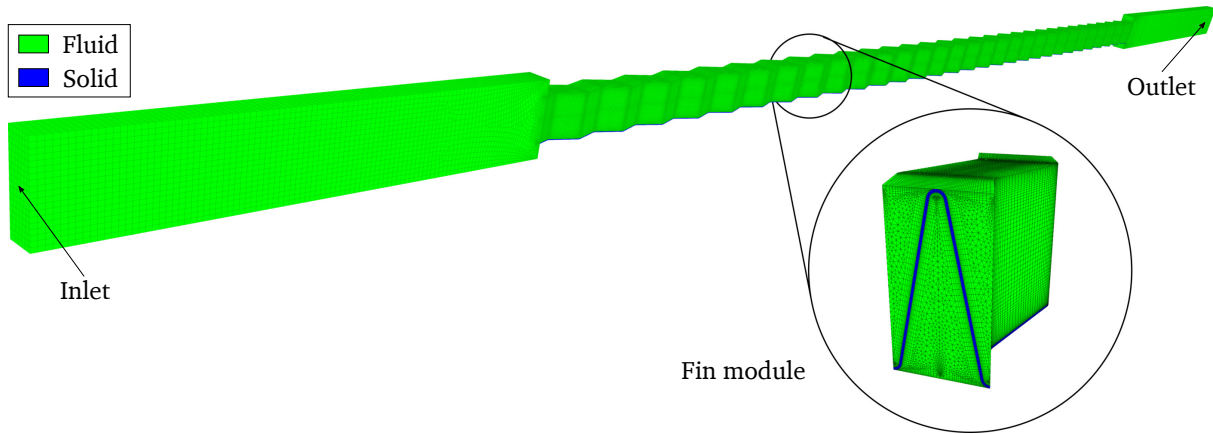


Figure 4.5. Entry region: computational mesh developed to study the entrance effect for the WF1 fin.

plying the mass conservation equation between the inlet section and the flow cross-section of the examined fin geometry. A uniform temperature value is set at the inlet, while a zero gradient boundary condition is assigned to pressure. At the outlet, pressure is constrained by setting a reference value for its integral mean, while a zero gradient boundary condition is applied to the velocity and temperature fields. Periodic conditions are enforced between the surfaces of the inlet and outlet domains. Table 4.5 summarises the boundary conditions considered for the entry region model.

4.3.2.3. Spatial discretisation

The computational domains are discretised by means of hybrid meshes. The fin stacks are formed by replicating the respective grid developed for the fully developed flow study. The techniques involved in the generation of the grids for fully developed flow are mentioned in Section 4.3.1.3. The lengths of the entrance and egress domains are equal to ten and twenty times the height of the fin, respectively. The computational grids feature a block-structured topology close to the inlet and outlet surfaces, whereas unstructured sections are present in the vicinity of the fin stack. Figure 4.5 shows the computational grid developed to evaluate the effects of entrance flows in the WF1 finned surface.

As stated previously, the computational grids for the entry region CFD models are devel-

Table 4.5. Boundary conditions applied to the inlet, outlet, left and right boundaries, and top and bottom surfaces in fluid and solid regions of entry region model.

Boundaries	inlet	outlet	left/right	top/bottom	fluid-solid interface
$\vec{v}$	uniform value	zero-gradient	periodic	no-slip	no-slip
fluid p	zero-gradient	fixed mean	periodic	zero-gradient	zero-gradient
T	uniform value	zero-gradient	periodic	uniform gradient	heat flux continuity
solid T	zero-gradient	zero-gradient	periodic	uniform gradient	heat flux continuity

oped on the basis of the meshes used for the fully developed flow analyses. Therefore, for the sensitivity analysis, the reader can refer to Section 4.3.1.

4.3.3. Head losses in inlet and outlet manifolds

4.3.3.1. Governing equations

The head losses associated with the inlet and outlet manifolds are determined by developing a CFD model comprising the actual geometries of both manifolds, along with porous regions modelling the internal finned channels. Fluid flow is supposed steady, isothermal, and incompressible. The velocity and pressure fields are derived by numerically solving the steady-state RANS equations using the $k-\omega$ SST turbulence model [195].

$$\begin{cases} \nabla \cdot (\vec{v} \otimes \vec{v}) = -\frac{1}{\rho} \nabla p + \nabla \cdot [(\nu + \nu_t) \nabla \vec{v}] - \sigma_m \\ \nabla \cdot \vec{v} = 0 \end{cases} \quad (4.5)$$

where σ_m is a momentum sink term that is non-zero only in porous regions, and it is defined on the basis of the Darcy-Forchheimer equation [234]:

$$\sigma_m = \left(\nu \vec{D} + \frac{1}{2} \text{tr}(\vec{v} \cdot I) \vec{F} \right) \vec{v}, \quad (4.6)$$

where the components of the vectors $\vec{D}$ and $\vec{F}$ determine the loss of momentum given by the porous regions. These vectors are obtained by calibrating the equivalent porosity model. The Darcy-Weisbach equation is used to determine the pressure gradient given by a single internal channel for different velocity values:

$$\frac{\nabla p_{\text{ch}}}{\rho} = \frac{f_{\text{fd}} L}{2 y L_m} \left(\frac{A_{\text{ch,m}}}{A_{\text{ch,r}}} v \right)^2, \quad (4.7)$$

where f_{fd} is the Darcy friction factor, related to fully developed flow conditions, of the specific fin considered. f_{fd} is an outcome of the fully developed flow analyses, whose results are presented in Section 4.5.1. L_m is the length of the porous regions in the computational model.

The porous regions within the CFD model feature larger flow cross-sections than those of the real heat exchanger channels, due to the absence of fin walls. For this reason, fluid velocity is corrected by term $A_{\text{ch,m}}/A_{\text{ch,r}}$, which is derived from the conservation equation in order to correctly evaluate the head losses in the internal channels. Finally, the streamwise components of $\vec{D}$ and $\vec{F}$ are determined by using a variant of the Levenberg-Marquardt algorithm [235] to fit Equation (4.6) to the $\nabla p_{\text{ch}}/\rho$ and corrected velocity values. Figure 4.6 shows an example of this procedure applied to the G94 heat exchanger. The difference between the corrected and unchanged channel losses is clearly spotted, underscoring the importance of the term $A_{\text{ch,m}}/A_{\text{ch,r}}$, especially for large velocity values. In addition, the

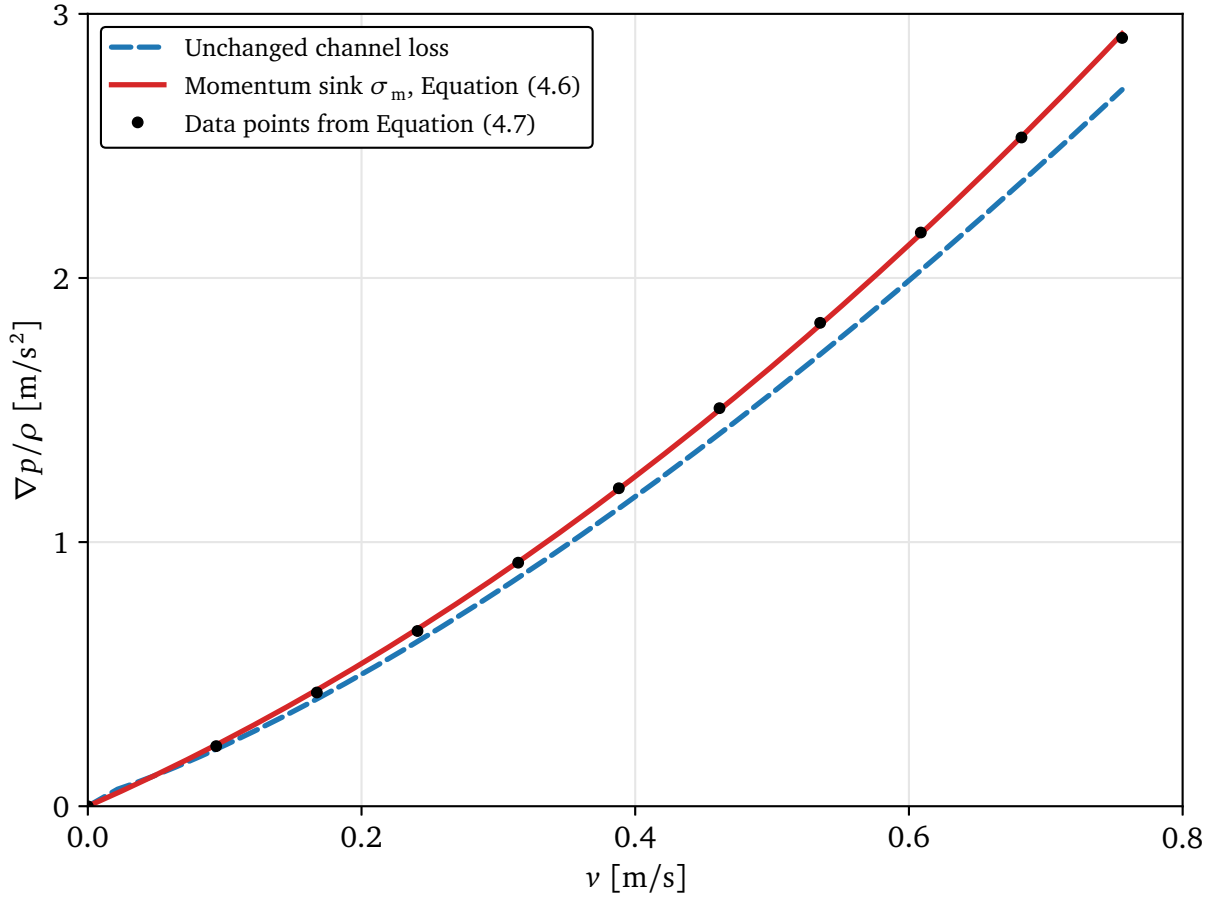


Figure 4.6. Comparison between the unchanged channel losses, the channel losses given by the calibrated porosity model, i.e. Equation (4.6), and the data points stemming from (4.7), which are employed for the curve fitting procedure.

selected optimisation algorithm allows for a near perfect match between the momentum sink term S_m and the corrected $\nabla p_{ch}/\rho$ values stemming from Equation (4.7). For this particular heat exchanger configuration the values of $\vec{D}$ and $\vec{F}$ along the streamwise direction amount to $D_z = 67.85 \times 10^6 \text{ m}^{-2}$ and $F_z = 105.2 \text{ m}^{-1}$. These are referred to porous regions with a length of $L_m = 40 \text{ mm}$. The components of $\vec{D}$ and $\vec{F}$ along the x and y directions are set to one thousand times the value of the streamwise component, to enforce one-dimensional flow inside porous regions.

4.3.3.2. Boundary conditions

At the inlet section, a uniform velocity value is imposed, and a zero gradient boundary condition is assigned to pressure. At the outlet, the surface mean pressure is set to a reference value, and a zero gradient boundary condition is applied to the velocity field. No-slip conditions are employed at solid walls. These boundary conditions are summarised in Table 4.6.

4.3.3.3. Spatial discretisation

The governing equations are solved numerically by means of an incompressible steady-state flow solver, which is based on the Finite Volume method and is implemented in OpenFOAM 7 [196]. The gradients are discretised through a linear Gauss scheme of second-order accuracy, whereas a first-order accurate upwind approach is employed for divergences. The convective terms and the pressure-velocity coupling are treated in the same way as in both the two types of models developed for the extended surfaces.

The computational domains are discretised with hybrid meshes using the same approach for all the configurations considered. The portions of the grid modelling the internal channels and the sections of the tanks away from inlet and outlet pipes are obtained using simple bottom-up block structured algorithms available in Pointwise. The sections of the manifolds close to the pipes are modelled using unstructured blocks extruded from 2D hybrid grids, where inflation layers are built using the same techniques described in Section 4.3.1.3. The simplified geometry resulting from the use of equivalent porous regions for the representation of internal channels allows for a substantial reduction of the number of elements included in the grid. The length of the inlet and outlet pipes is set to twenty times their diameter, to allow for correct flow development. The pipes are discretised with a butterfly-style grid, which is smoothed by means of a variant of the Thomas-Middlecoff algorithm [236] implemented in Pointwise. Figure 4.7 shows the selected views of the computational grid developed for the O94 heat exchanger.

4.3.3.4. Mesh sensitivity analysis

To evaluate the dependence of numerical results on the resolution of the mesh, three computational grids with the same topology are developed, with sizes of 6, 13, and 26 million elements, respectively. Table 4.7 shows the values of pressure difference obtained from the coarse, medium, and fine meshes of the A94 charge air cooler at $Re_{D_p} = 285 \times 10^3$, along with the number of elements of each grid, and the average size of cells in the far field, Δ , and near the walls, Δ_w . The Δp -values are evaluated between cross-sections located in the same positions as the pressure taps used during the experimental tests. The medium mesh is selected for all the following CFD analyses, as the deviation of 1.43% between the results obtained from the medium and fine meshes is deemed acceptable, particularly when considering the substantial savings in computational effort that can be achieved by using the medium grid. An analogous procedure is adopted for all the heat exchanger models, yielding similar results.

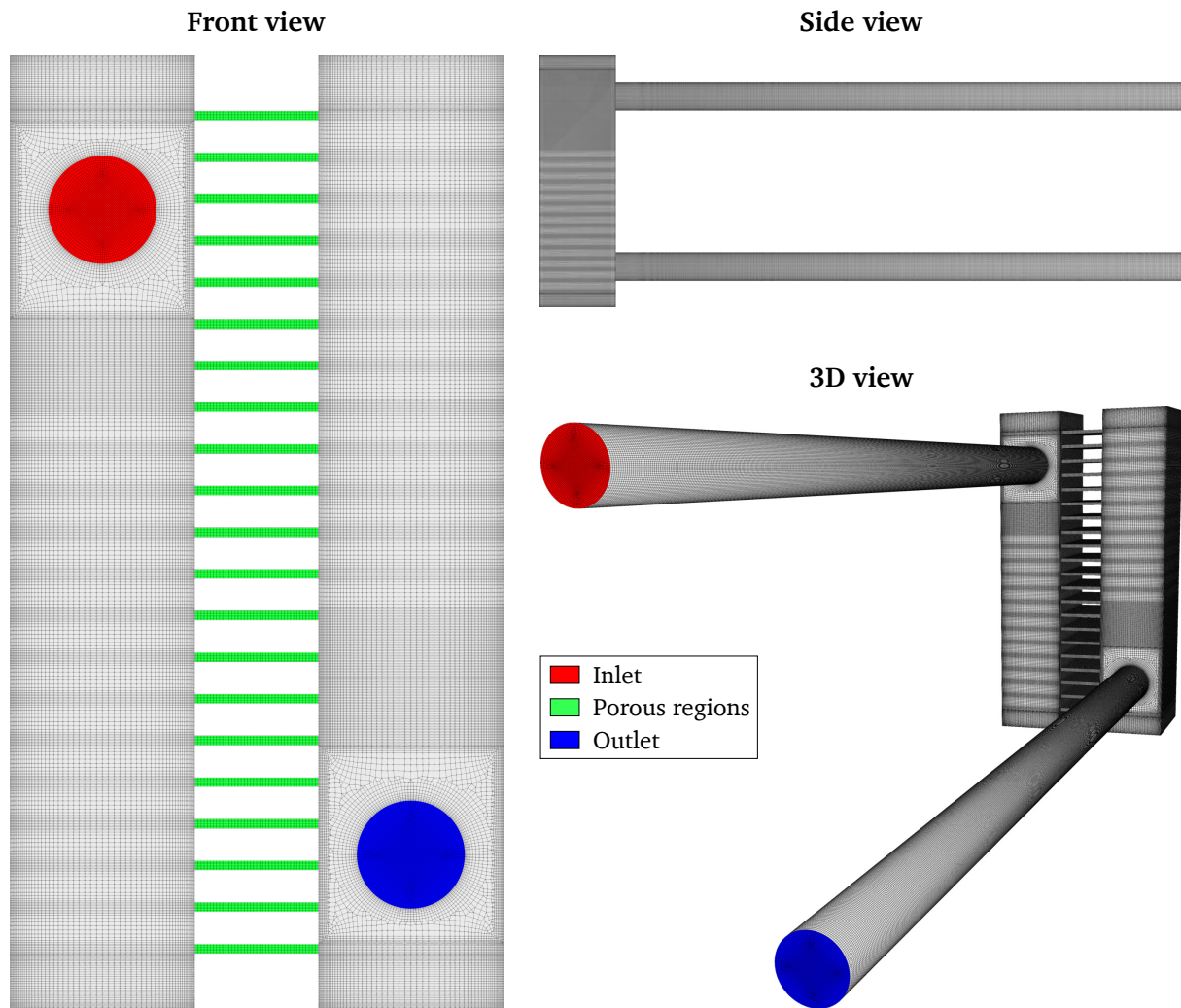


Figure 4.7. Sample mesh of the overall model for internal head losses, related to the O94 oil cooler.

Table 4.6. Boundary conditions applied to the inlet, outlet, and walls of the computational model of the internal flow of the heat exchangers.

Boundaries	Inlet	Outlet	Walls
$\vec{v}$	uniform value	zero-gradient	no-slip
p	zero-gradient	fixed mean	zero-gradient

Table 4.7. Pressure drop values obtained from the coarse, medium, and fine meshes of the A94 heat exchanger.

Grid	Δ [mm]	Δ_w [mm]	Number of cells	Δp [kPa]	Relative deviation
Coarse	1.62	0.4	6×10^6	22.47	-
Medium	1.25	0.2	13×10^6	23.46	2.94%
Fine	0.99	0.1	26×10^6	23.13	1.43%

4.3.4. Global models

4.3.4.1. Heat transfer

The global thermal model for plate-fin heat exchangers is based on the ε – NTU method. The product between the equivalent heat transfer coefficient and the heat transfer surface is obtained by considering the series of thermal resistances related to the internal and external flows.

$$UA = \frac{1}{R_{\text{int}} + R_{\text{ext}}} = \frac{1}{\frac{1}{U_{\text{int}} A_{\text{int}}} + \frac{1}{U_{\text{ext}} A_{\text{ext}}}} \quad (4.8)$$

The internal global heat transfer coefficient U_{int} is determined by employing the Nusselt number correlation, for fully developed flow, of the specific fin considered. This correlation is a result stemming from the single fin model. It is presented in Equation (4.20), as part of Section 4.5.1. The heat transfer coefficient associated with the external channels is obtained from the respective value of Nusselt number, which is derived from the product of two contributions: (i) the fully developed Nu value, derived from Equation (4.20); (ii) the average Nusselt variation with respect its fully developed value in the entrance region:

$$\overline{\text{Nu}}_e = \frac{1}{\frac{s}{2l} - 1} \int_1^{\frac{s}{2l}} \Psi(\chi) d\chi, \quad (4.9)$$

where $\Psi(\chi)$ expresses the Nusselt number variation with respect to the position of each elementary module within the external channels, and the Reynolds number. This is a result of the entry region study, and is presented in Section 4.5.2. Internal and external heat transfer areas A_{int} and A_{ext} are obtained by multiplying heat transfer area of the single fin module by the number of modules. The effectiveness of the heat exchanger is computed considering both internal and external fluids unmixed in a cross flow configuration.

$$\varepsilon = 1 - \exp \left[\frac{C_{\text{max}}}{C_{\text{min}}} \text{NTU}^{0.22} \left\{ \exp \left[-\frac{C_{\text{min}}}{C_{\text{max}}} \text{NTU}^{0.78} \right] - 1 \right\} \right] \quad (4.10)$$

Finally, the heat transferred between the two fluids is given by:

$$\dot{Q} = \varepsilon C_{\text{min}} (T_{\text{int},i} - T_{\text{ext},i}). \quad (4.11)$$

The thermophysical properties of the internal and external fluids are computed as a function of the mean temperature between the respective inlet and outlet sections. The computation of $\dot{Q}$ is performed iteratively, by updating the values of fluid properties at each step of the procedure, until convergence is reached.

4.3.4.2. Internal head losses

Internal head losses are computed by summing the losses in the inlet and outlet manifolds and in the internal channels:

$$\Delta p_{\text{int}} = \frac{1}{2} \rho_i \beta_i v_i^2 + \Delta p_{\text{ch}} + \frac{1}{2} \rho_o \beta_o v_o^2, \quad (4.12)$$

where β_i and β_o are the concentrated loss coefficients of the inlet and outlet manifolds, respectively. These values are derived from the CFD model presented in Section 4.3.3. The values of channel losses Δp_{ch} , in oil coolers and water radiators are derived from the Darcy-Weisbach equation:

$$\Delta p_{\text{ch}} = \frac{1}{2} \rho f_{\text{fd}} \frac{L}{y} v_{\text{ch}}^2. \quad (4.13)$$

In charge air coolers large temperature differences are often observed between the inlet and outlet of the internal channels. This causes significant variations in the thermophysical properties of the air, and hence in the local Reynolds number. Therefore, the head losses in the internal channels of charge air coolers are computed using the solver for steady-state one-dimensional flow in networks of pipes developed in Chapter 2. The air mass flow rate is assumed to be uniformly distributed among internal channels, thus only one of them is included in the reduced order model. The domain is discretised by an open-loop network presenting 200 nodes and 199 branches. Each branch exchanges heat with the external environment according to:

$$\dot{Q}_j = U_{\text{ch}}(T_{\infty} - T_j), \quad (4.14)$$

where U_{ch} is the global heat transfer coefficient of a single channel, and is obtained from Equation (4.8).

Air density is determined by means of an incompressible ideal gas law, considering a molar mass of 28.97 g/mol, while the other thermophysical properties are derived from linear interpolation of tabled data. At the channel inlet the temperature is set to a uniform value, while at the outlet a zero gradient condition is enforced.

4.3.4.3. External head losses

External head losses are determined by means of the well-known Darcy-Weisbach formula.

$$\Delta p_{\text{ext}} = \frac{1}{2} \rho f_{\text{fd}} \bar{f}_e \frac{S}{y} v^2 \quad (4.15)$$

The equivalent friction factor of external channels is given by the product of two components: (i) f_{fd} , that is related to fully developed flow regime; (ii) $\bar{f}_e$ which expresses the mean change of the friction factor in the entry region relative to its respective value for fully developed flow. The first of the two aforementioned contributions is obtained from the single fin friction factor correlation (4.19), which is a result of the fully developed flows

analyses and is presented later in Section 4.5.1. $\bar{f}_e$ is obtained from the integral average of $\Phi(\chi)$, which expresses the variation of f , with respect to its fully developed value, as a function of the position of each single fin module within the external channels, and the Reynolds number. $\Phi(\chi)$ stems from the entry region study, and is presented later in Section 4.5.2.

$$\bar{f}_e = \frac{1}{\frac{s}{2l} - 1} \int_1^{\frac{s}{2l}} \Phi(\chi) d\chi \quad (4.16)$$

Air properties are evaluated at the mean temperature between inlet and outlet sections, furthermore the air outlet temperature is derived from the global heat transfer model, which is presented in Section 4.3.4.1.

4.4. Experimental apparatus and procedures

The experimental tests are conducted on the cooling component test bed of Virtual Vehicle Research GmbH, which is depicted in the schematic of Figure 4.8a. The test bed consists of a small open loop wind tunnel powered by a radial fan, that can provide a maximum air mass flow rate of 3 kg/s. The heat exchangers are mounted at the end of the test section, that has the shape of a truncated pyramid, as shown in Figure 4.8b. The air mass flow rate is measured with a V-Cone flow meter ($\pm 0.5\%$ accuracy), and then the air is heated up to the specified inlet temperature. Air temperature is measured by employing calibrated type K thermocouples, that have an estimated error of 0.2 K. The inlet air temperature and humidity are measured just before the contraction, while the outlet temperature is measured by twelve thermocouples arranged in three rows of four sensors each.

The internal flow rate of the heat exchanger is measured with an impeller device, that is equipped with a magneto-inductive sensor. The sensor employed for air and water flows features an accuracy of $\pm 1.0\%$, while the one used for oil flows is accurate up to $\pm 0.15\%$ of the flow rate.

The internal flow temperatures are measured near the inlet and outlet sections of the heat exchanger by using calibrated Pt100 sensors, which have an estimated error of 20 mK. The pressure drops related to the internal and external flows are measured by differential pressure transducers. Pressure taps used in the wind tunnel have an accuracy of $\pm 1.0\%$, while the ones employed for the internal flow are 0.25% accurate for all fluids.

For each heat exchanger, the internal and external head losses and heat exchanged are measured for four values of internal flow rate and for four to five values of external flow rate. Then, the values of internal and external head losses are averaged over the external and internal fluid flow rates respectively, to obtain an average head loss for each flow rate.

From this point on, the different flow rates associated with internal flow in the heat exchangers will be identified by their corresponding inlet Reynolds number Re_{D_p} , which is calculated on the basis of the diameter of the inlet and outlet pipes.

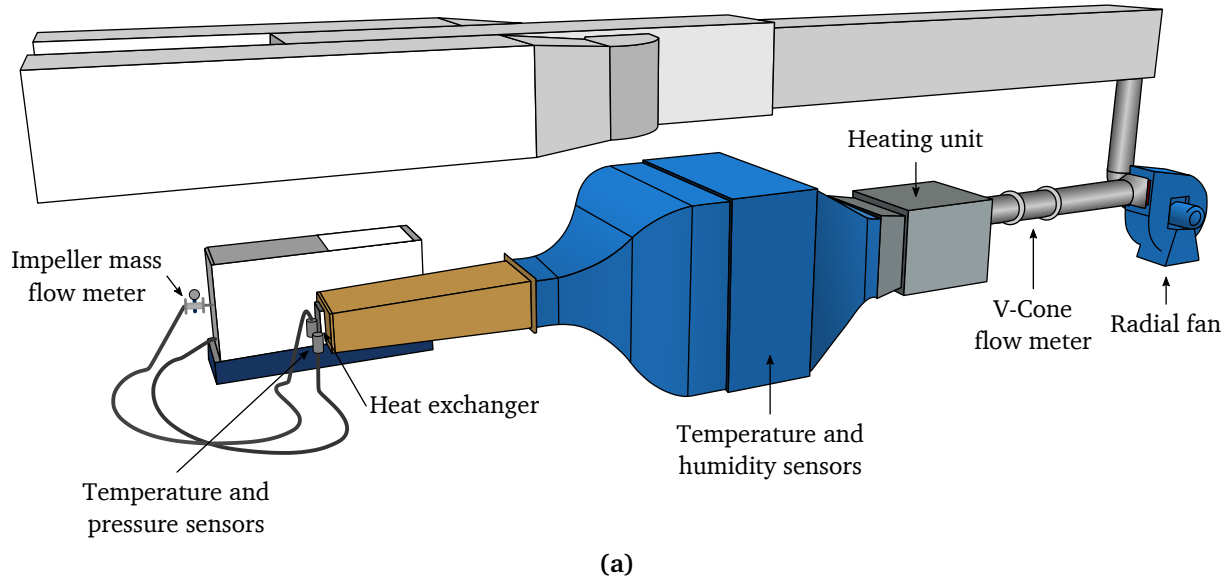


Figure 4.8. Drawing of the experimental apparatus, in which the main components are highlighted (a), and a picture showing one of the examined heat exchangers mounted to the end of the test section of the wind tunnel (b).

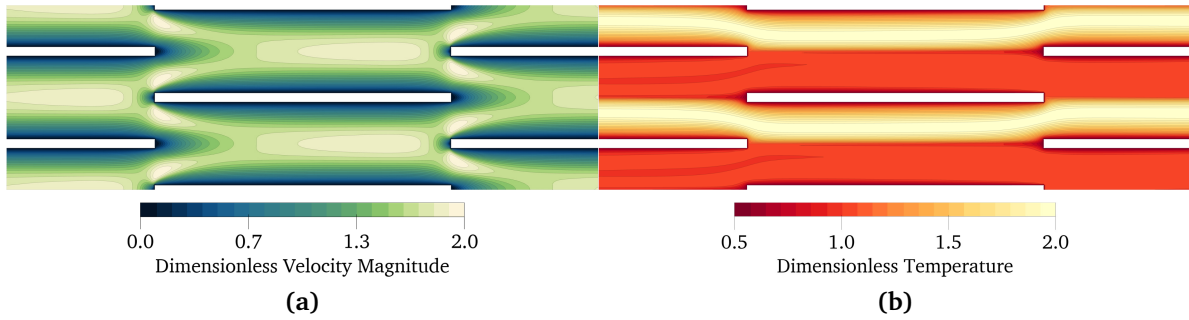


Figure 4.9. OSF1: colour maps of dimensionless velocity magnitude (a) and temperature (b) along a longitudinal cross-section, as obtained by CFD at $Re_y = 100$ and $Pr = 190$.

4.5. Results and discussion

4.5.1. Single fin model

In the following, the results of the CFD analyses are presented in terms of dimensionless velocity and temperature fields. The fluid temperature field is expressed in dimensionless form in terms of the temperature difference $\vartheta = (\bar{T}_w - T)/(\bar{T}_w - \bar{T}_b)$, where $\bar{T}_w$ and $\bar{T}_b$ are the average wall and bulk temperatures, respectively. The velocity and temperature fields are displayed by means of longitudinal sections taken cross-sections located at the half height of the fins. Figure 4.9 shows the velocity and temperature fields for the OSF1 fin, related to an oil flow at $Re_y = 100$. The laminar boundary layers growth is apparent, and behind each wall small wakes are formed due to flow separation. The dimensionless temperature field in Figure 4.9b shows a marked asymmetric distribution, despite the symmetric velocity field observed in Figure 4.9a. This behaviour is caused by the presence of fillet radii between the vertical and horizontal walls of the fin, which determine the formation of asymmetric half-channels at each fin interruption. In addition, this phenomenon seems to be emphasised by high values of the Prandtl number as confirmed by the numerical results which will be presented in Chapter 5.

Figure 4.10a shows the distribution of local friction coefficient on the walls of the fin: as expected, the maximum values are found near the leading edge of the walls and on the front faces. In this case the total friction resistance is worth about half of the total losses, meaning that the rest is given by pressure drag. The distribution of local heat transfer coefficient (Figure 4.10b) of the OSF1 fin features maximum values near the leading edges and on the front faces of the walls, while negative peaks are due to the hot current that heats the fin walls, as shown in Figure 4.9b.

Figure 4.11 shows dimensionless velocity and temperature fields of the OSF2 fin, corresponding to a flow of water at $Re_y = 800$. In this case the asymmetric offset determines a preferential path for the flow. Figures 4.12 and 4.13 show the dimensionless velocity and temperature fields of the WF1 and WF4 fins, related to an air flow at $Re_y = 10000$, respectively. The two fins show similar velocity distributions: flow separations occur in cor-

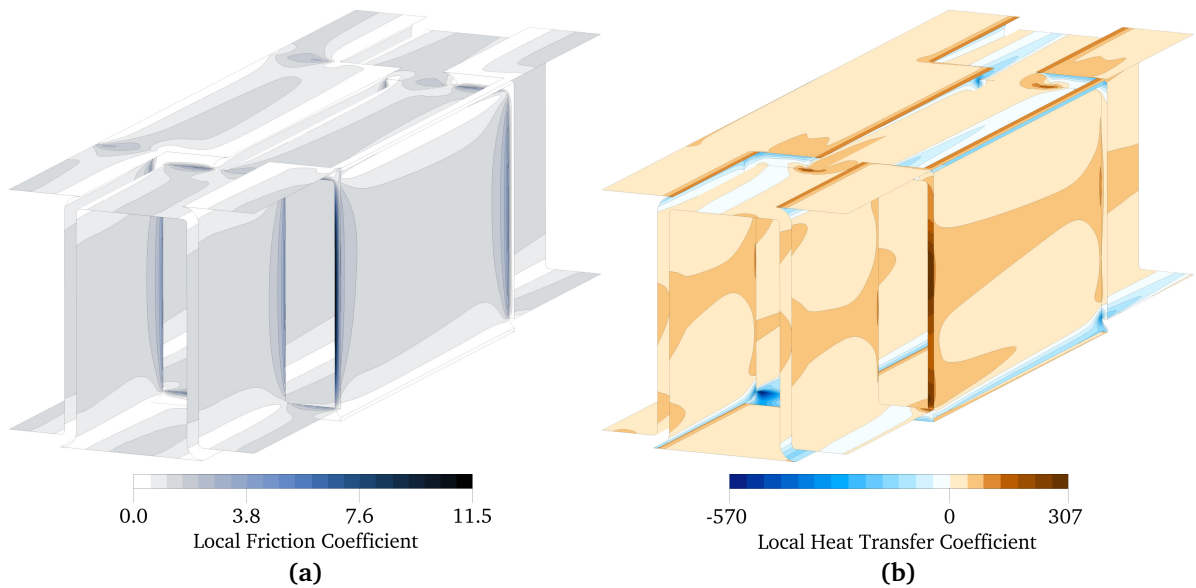


Figure 4.10. OSF1: distributions of local friction coefficient (a) and local heat transfer coefficient (b), as obtained by CFD at $Re_y = 100$ and $Pr = 190$.

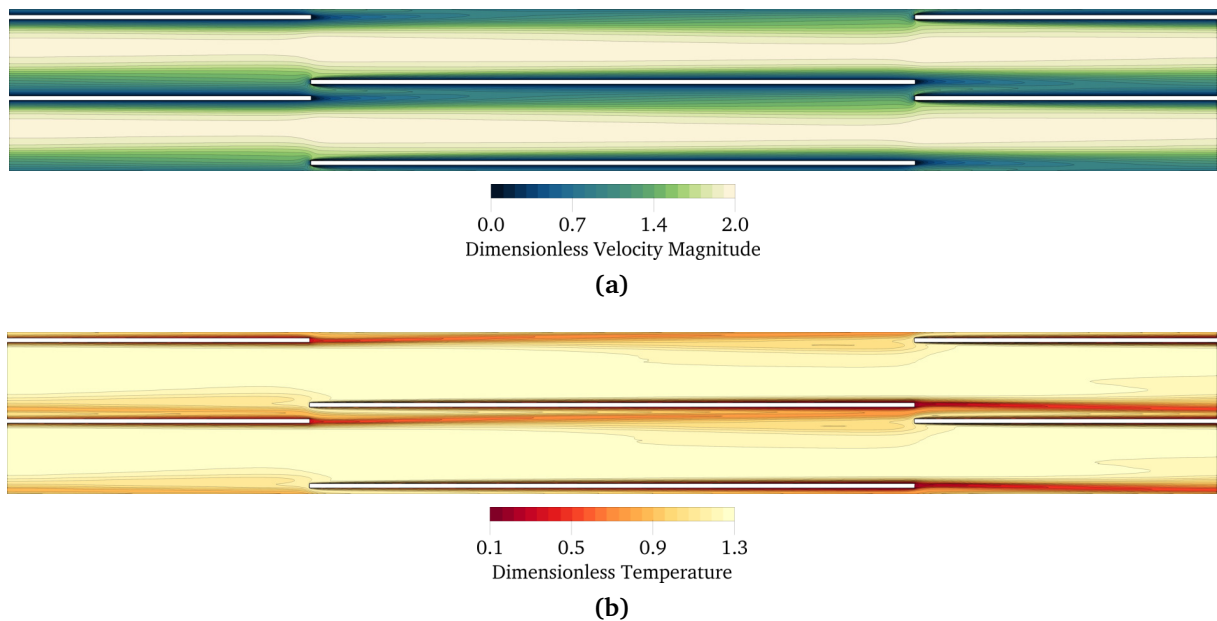


Figure 4.11. OSF2: colour maps of dimensionless velocity magnitude (a) and temperature (b) along a longitudinal cross-section, as obtained by CFD at $Re_y = 800$ and $Pr = 7$.

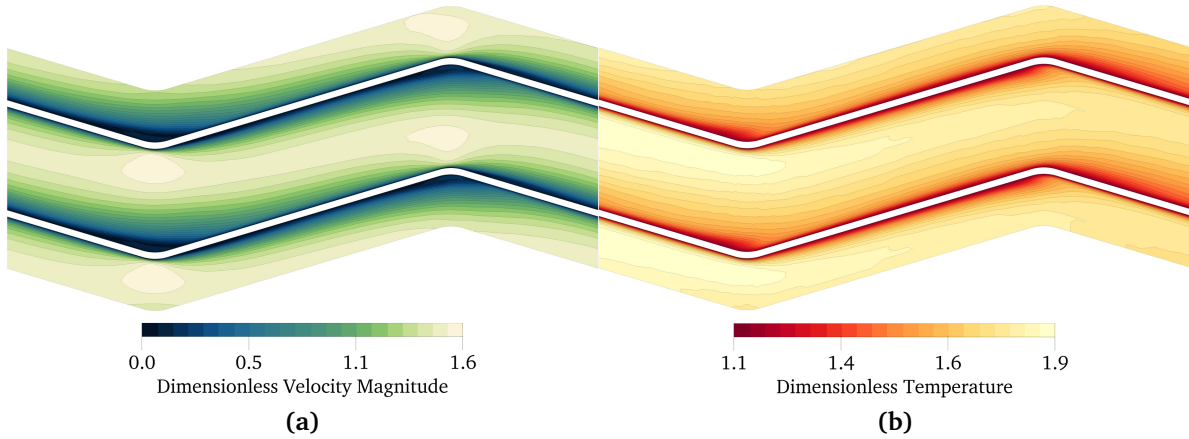


Figure 4.12. WF1: colour maps of dimensionless velocity magnitude (a) and temperature (b) along a longitudinal cross-section, as obtained by CFD at $Re_y = 10000$ and $Pr = 0.71$.

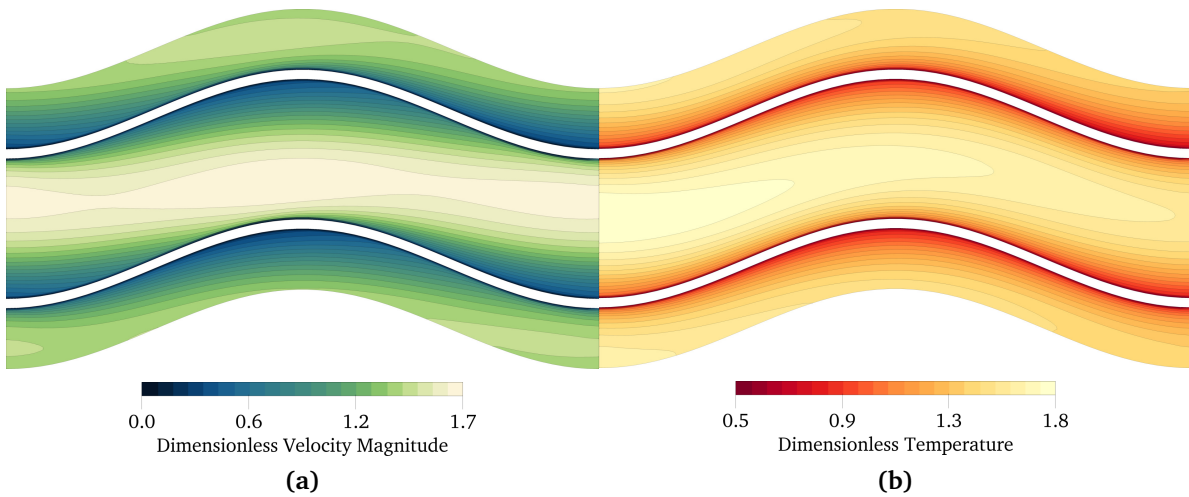


Figure 4.13. WF4: colour maps of dimensionless velocity magnitude (a) and temperature (b) along a longitudinal cross-section, as obtained by CFD at $Re_y = 10000$ and $Pr = 0.71$.

response of the troughs of the fins, while the maximum velocity values are found near the crests. As expected, temperature hot spots are present in correspondence of the flow recirculation regions. Similar results were obtained for WF2 and WF3. Figure 4.14a shows the local friction coefficient of the WF3 fin: skin friction drag is uniformly distributed on the walls with slight maximum and minimum peaks are positioned in close proximity to the fin crests and troughs, respectively. The distribution of local heat transfer coefficient shown in Figure 4.14b feature regions of great heat transfer effectiveness at the half height of the fins, while regions of negative heat transfer coefficient are observed where the vertical walls of the fin form an angle of less than 90 degrees with the horizontal ones.

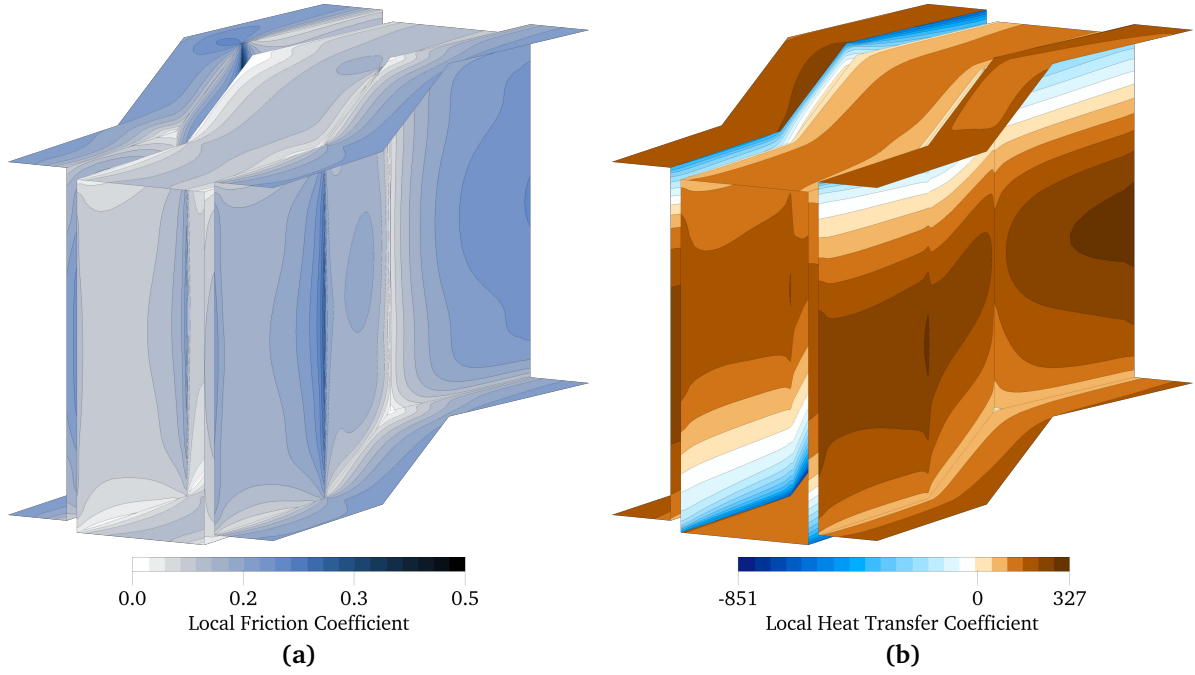


Figure 4.14. WF3: distributions of local friction coefficient (a) and local heat transfer coefficient (b), as obtained by CFD at $Re_y = 10000$ and $Pr = 0.71$.

Integral results on head losses and heat transfer are presented in terms of the equivalent Darcy friction factor and the mean Nusselt number. The equivalent friction factor associated with single fins is computed by means of the well known Darcy-Weisbach formula, by replacing the hydraulic diameter with the fin height:

$$f = \frac{y(\bar{p}_i - \bar{p}_o)}{l \rho \bar{v}_i^2}, \quad (4.17)$$

where overbars indicate integral mean field values over the inlet (i) and outlet (o) sections. The Nusselt number is evaluated from the average fluid bulk temperature between the inlet and outlet sections $\bar{T}_b = 0.5(T_{b,i} + T_{b,o})$:

$$Nu = \frac{\dot{Q} y}{\lambda A_{\text{ref}}(\bar{T}_w - \bar{T}_b)}, \quad (4.18)$$

where $\bar{T}_w$ represents the integral mean of the temperature field over the heat transfer area A_{ref} , which includes the fluid-solid interfaces and the top and bottom boundaries. The values of Nusselt number and the friction factor derived from the CFD models are used to obtain correlations, as functions of the Reynolds number alone.

$$\log f_{\text{fd}} = b_1 \log^{b_2} Re_y + b_3 \quad (4.19)$$

$$Nu_{\text{fd}} = c_1 Re_y^{c_2} + c_3 \quad (4.20)$$

Since each fin is associated to a specific fluid depending on the type of cooler, the effect of Pr is included in the c_i coefficients. The values of the coefficients b_i and c_i are found by fitting Equations (4.19) and (4.20) to numerical results by means of a variant of the Levenberg-Marquardt algorithm [235], implemented in the SciPy library [237]. The values of b_i and c_i are reported in Table 4.8 for each fin.

Numerical results of offset-strip fins are compared with those estimated by means of the correlations presented by Joshi and Webb [208]:

$$f_F = 8.12 \text{Re}_{D_h}^{-0.74} \left(\frac{l}{D_h} \right)^{-0.41} \alpha^{-0.02}, \quad (4.21)$$

$$j = 0.53 \text{Re}_{D_h}^{-0.5} \left(\frac{l}{D_h} \right)^{-0.15} \alpha^{-0.14}, \quad (4.22)$$

and Manglik and Bergles [206]:

$$f_F = 9.6243 \text{Re}_{D_h}^{-0.7422} \alpha^{-0.1856} \delta^{0.3053} \gamma^{-0.2659} \times \left[1 + 7.669 \cdot 10^{-8} \text{Re}_{D_h}^{4.429} \alpha^{0.920} \delta^{3.767} \gamma^{0.236} \right]^{0.1}, \quad (4.23)$$

$$j = 0.6522 \text{Re}_{D_h}^{-0.5403} \alpha^{-0.1541} \delta^{0.1499} \gamma^{-0.0678} \times \left[1 + 5.269 \cdot 10^{-5} \text{Re}_{D_h}^{1.340} \alpha^{0.504} \delta^{0.456} \gamma^{-1.055} \right]^{0.1}, \quad (4.24)$$

where α , δ , and γ are:

$$\alpha = \frac{x-t}{y-t}, \quad \delta = \frac{t}{l}, \quad \gamma = \frac{t}{x-t}. \quad (4.25)$$

Figure 4.15 shows the comparison with the correlations available in the literature for the OSF1 fin, considering oil at $Pr = 190$ as the working fluid. There is a good agreement between the CFD f -values and those estimated by means of Joshi and Webb's correlation

Table 4.8. Values of b_i and c_i coefficients in Equations (4.19) and (4.20).

ID	b_1	b_2	b_3	c_1	c_2	c_3	Re_y -range
OSF1	-1.704	0.646	3.043	0.020	1.392	10.22	$15 \leq Re_y \leq 70$
				41.74	0.140	-57.89	$70 < Re_y \leq 300$
OSF2	11.37	-0.247	-9.915	0.021	0.805	3.571	$100 \leq Re_y \leq 2000$
WF1	-1418	0.0008	1419	0.028	0.862	10.51	$1000 \leq Re_y \leq 30000$
WF2	-1033	0.0012	1034	0.205	0.636	0.728	$800 \leq Re_y \leq 10000$
WF3	-5.635	0.158	6.490	0.045	0.832	2.621	$800 \leq Re_y \leq 10000$
WF4	2085	-0.0006	-2083	0.570	0.580	-16.46	$1000 \leq Re_y \leq 30000$

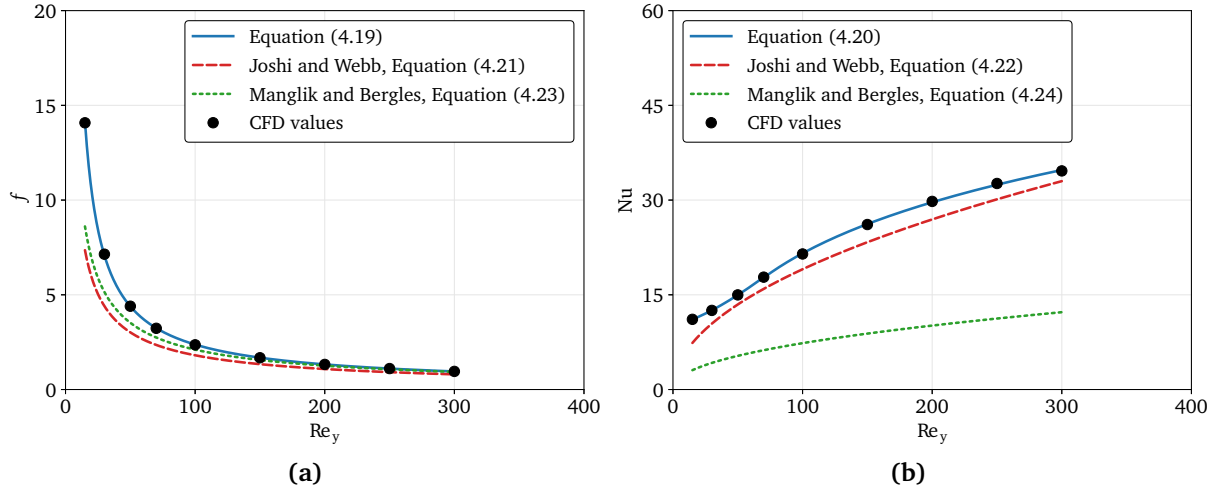


Figure 4.15. OSF1: friction factor (a) and Nusselt number (b) data: present study vs literature ($Pr = 190$).

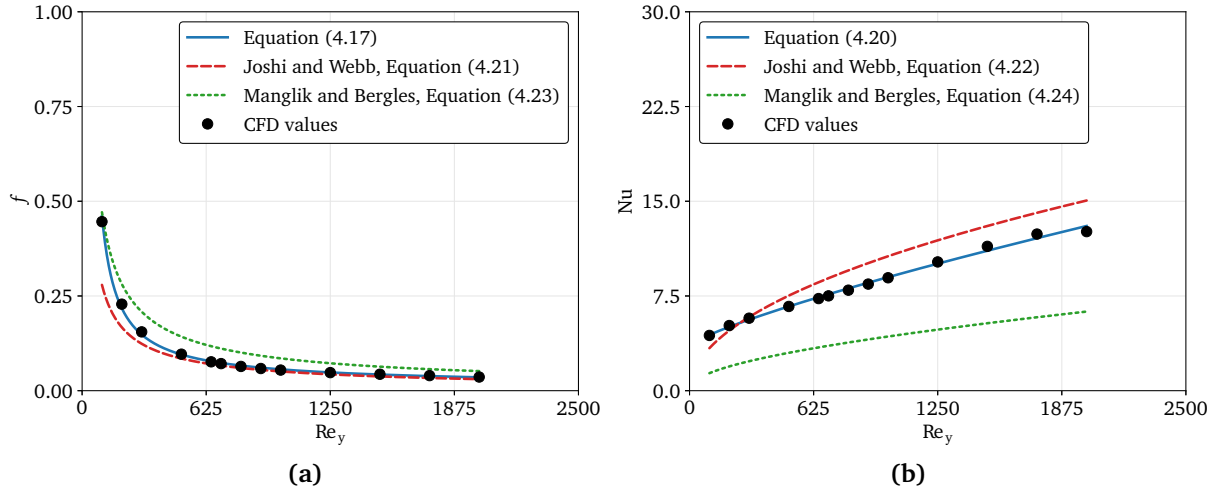


Figure 4.16. OSF2: friction factor (a) and Nusselt number (b) data: present study vs literature ($Pr = 7$).

towards the upper end of the Re -range considered. The deviation grows with decreasing Reynolds, and at $Re_y = 15$ the numerical f -value is almost twice the one obtained from Equation (4.21). The model developed by Manglik and Bergles gives a better agreement, as the CFD values are exactly matched for $Re_y \geq 150$, and at low Re -values the deviations between numerical and estimated f -values are lower than those resulting from Joshi and Webb's model. Reynolds number values between 15 and 150 are typical of internal channels of oil coolers. Therefore, an accurate prediction of the friction factor in this range is critical for a precise evaluation of the internal channels head losses. Both models developed by Joshi and Webb, and Manglik and Bergles are seen to underestimate present numerical Nu -values. Both of these models were developed on the basis of experimental data on air and water flows. Therefore, significant deviations are expected when extrapolating them to a

Pr-value of 190. In addition, the Chilton-Colburn analogy does not hold for laminar flows and value of Prandtl number greater than 160.

The comparison between the numerical f -values obtained for OSF2 and the considered models from the literature leads to similar results to those observed for OSF1, as shown in Figure 4.16a. However, in this case, the most accurate estimation of numerical results is provided by Joshi and Webb's correlation. In this instance, the numerical Nu-values are overestimated by Joshi and Webb's model, while they are underestimated by Manglik and Bergles's correlation, as illustrated in the plot of Figure 4.16b. It should be noted that the correlations by Joshi and Webb and Manglik and Bergles were developed for offset-strip fins with a centred or symmetric offset, whereas OSF2 has an off-centre offset; therefore, deviations in the estimations of both f and Nu are to be expected.

The values of f and Nu obtained from the CFD analyses for the wavy fins are compared with the models developed by Awad and Muzychka [232]:

$$\begin{aligned} f_F &= (f_{F,1}^2 + f_{F,2}^2)^{0.5}, \\ f_{F,1} &= 24 \text{Re}_{D_h}^{-1} \frac{L_e}{2l} (1 - 1.3553\alpha + 1.9467\alpha^2 - 1.7012\alpha^3 \\ &\quad + 0.9564\alpha^4 - 0.2537\alpha^5), \\ f_{F,2} &= 3.44 \text{Re}_{D_h}^{-0.5} \left(\frac{D_h}{0.5 L_e} \right)^{0.5}, \end{aligned} \quad (4.26)$$

$$\begin{aligned} j &= (j_1^5 + j_2^5)^{0.2}, \\ j_1 &= \frac{7.541}{\text{Re}_{D_h} \text{Pr}^{1/3}} (1 - 2.610\alpha + 4.970\alpha^2 - 5.119\alpha^3 \\ &\quad + 2.702\alpha^4 - 0.548\alpha^5), \\ j_2 &= 0.664 \text{Re}_{D_h}^{-0.5} \left(\frac{D_h}{0.5 L_e} \right)^{0.5}, \end{aligned} \quad (4.27)$$

and Dong *et al.* [217].

$$f_F = 1.16 \text{Re}_{D_h}^{-0.309} \left(\frac{x}{2y} \right)^{0.3703} \left(\frac{x}{4a} \right)^{-0.25} \left(\frac{L_d}{2l} \right)^{-0.1152} \quad (4.28)$$

$$j = 0.0836 \text{Re}_{D_h}^{-0.2309} \left(\frac{x}{2y} \right)^{0.1284} \left(\frac{x}{4a} \right)^{-0.153} \left(\frac{L_d}{2l} \right)^{-0.326} \quad (4.29)$$

Since the present work focuses on fully developed flow, the model presented by Dong *et al.* is slightly modified, by discarding the terms related to the total wavy fin length L_d . However, it should be noted that this modification leads to a loss of accuracy, since the model also loses its dependence on the wavelength. Figures 4.17 and 4.18 show values of friction factor and Nusselt obtained from CFD, the respective correlations derived from numerical results, and the comparison with data from the literature for WF1 and WF4. For

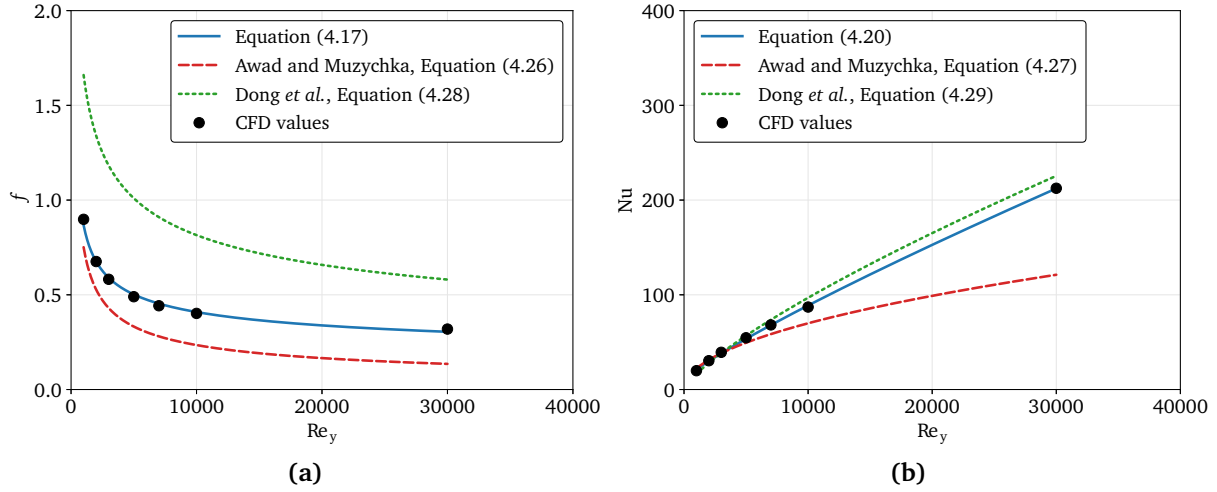


Figure 4.17. WF1: friction factor (a) and Nusselt number (b) data: present study vs literature.

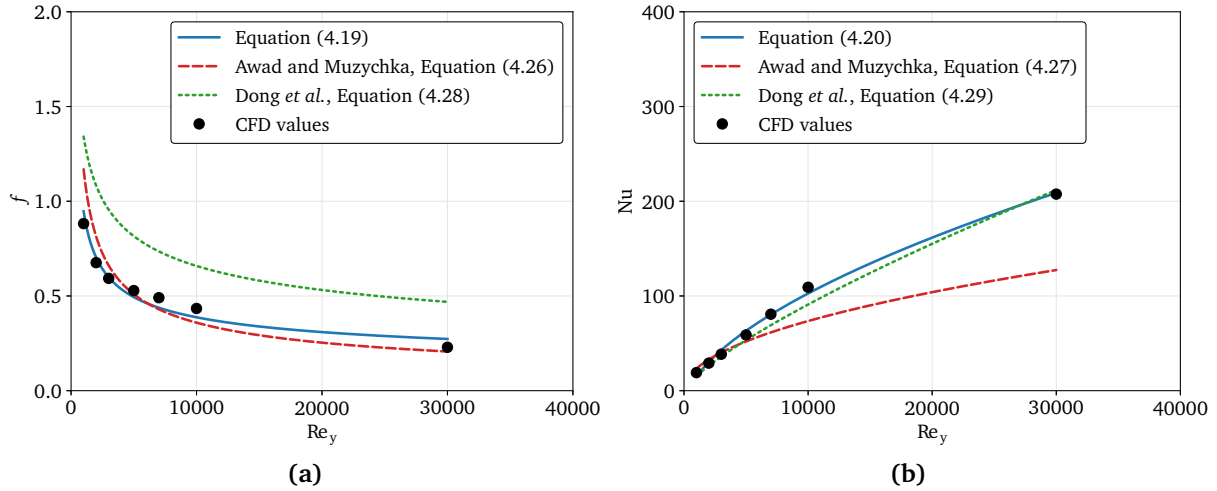


Figure 4.18. WF4: friction factor (a) and Nusselt number (b) data: present study vs literature.

both geometries, the modified correlation by Dong *et al.* overestimates the present numerical f -values, while the analytical model by Awad and Muzychka is in better agreement with the CFD-values, particularly for WF4. The numerical Nu -values are matched by Awad and Muzychka's model with small deviations at low Re values, while the deviations increase with the Re and are maximum for $Re_y = 30,000$. The values of Nusselt number estimated by means of the correlation by Dong *et al.* are in good agreement with CFD-values, as the observed deviations are of $\pm 20\%$ at most. Similar results were found for WF2 and WF3.

Overall, the correlations proposed in the literature are adequate for the calculation of the friction factor for the geometries and flow regimes considered. In most cases, noticeable deviations were found regarding the prediction of the Nusselt number. The uncertainties related to the estimations done by means of the correlations by Dong *et al.* for both f and

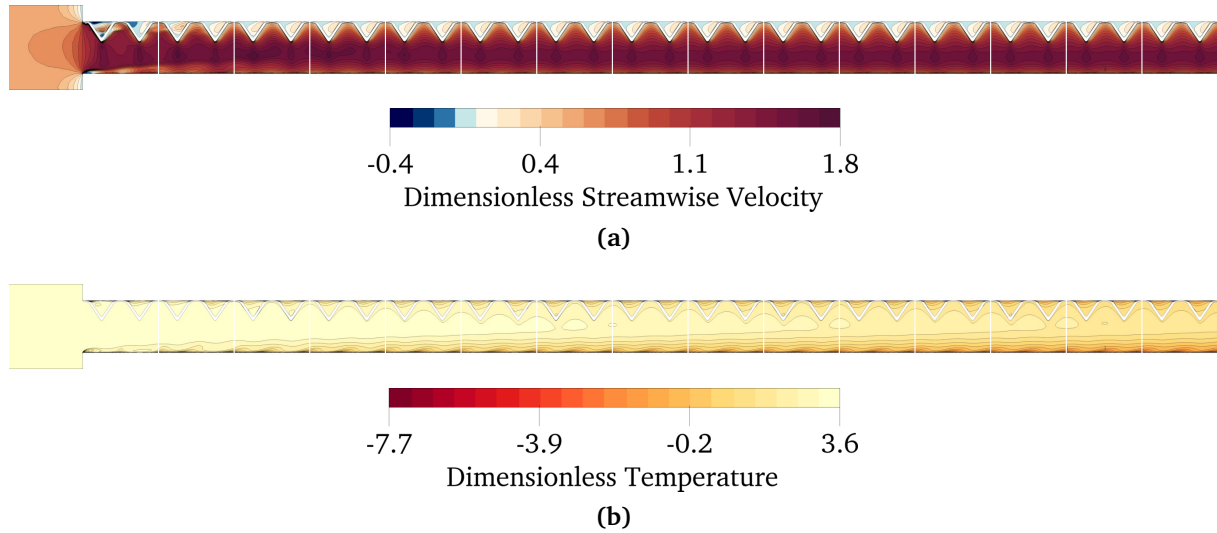


Figure 4.19. WF1 entrance region: colour maps of dimensionless streamwise velocity component (a) and dimensionless temperature (b) of the first fifteen elementary modules, obtained by CFD at $Re_y = 10000$ and $Pr = 0.71$.

Nu may be attributed to the modifications done to adapt the model for fully developed flow. The analytical model by Awad and Muzychka's performs much better for the estimation of f , while significant deviations are observed when estimating the values of Nusselt number. It must also be emphasised that the multitude of possible combinations of the dimensional parameters of the fins makes it very difficult, if not impossible at all, to define universal correlations. The CFD analyses presented here not only provide accurate and reliable results for the specific geometries analysed in this work, but also constitute an additional set of results that may be useful to broaden the range of validity of possible general correlations.

4.5.2. Entry region

The results of the CFD analyses on velocity and temperature fields are presented by means of contour plots of dimensionless streamwise velocity component and temperature $\vartheta = (\bar{T}_w - T)/(\bar{T}_w - \bar{T}_b)$, on spanwise-centred vertical cross-sections, which include the first fifteen single-fin modules. Figure 4.19 shows the dimensionless velocity and temperature fields obtained for WF1. Near the inlet of the fin stack flow separations are present along the lower and upper edges and along the leading edge of the walls of the fins. Figure 4.19b shows the dimensionless temperature field which reaches full development roughly near the fifteenth module. Figure 4.20 shows the dimensionless streamwise velocity and temperature fields of WF4. As for WF1, the flow separates near the sharp edges in correspondence of the inlet section. Both the velocity and temperature fields are fully developed by the fifteenth single fin module.

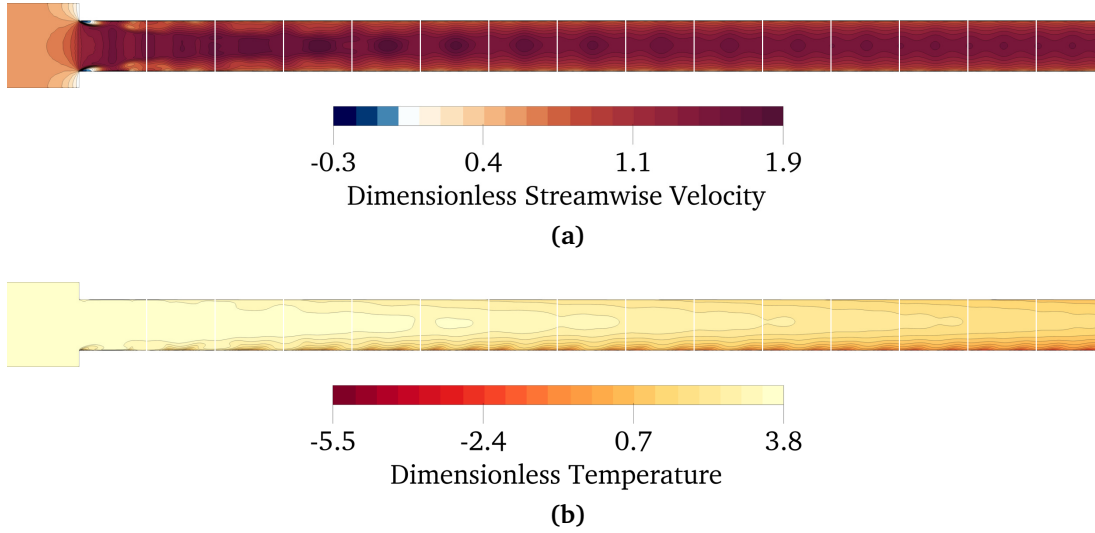


Figure 4.20. WF4 entrance region: colour maps of dimensionless streamwise velocity component (a) and dimensionless temperature (b) of the first fifteen elementary modules, obtained by CFD at $Re_y = 10000$ and $Pr = 0.71$.

Entrance effects on head losses and heat transfer effectiveness of WF1 and WF4 are evaluated by computing the deviation of the Darcy friction factor and Nusselt number with respect to the corresponding fully developed values, for each j -th single fin module:

$$\Phi_j = \log \left(\frac{1}{f_{fd}} \frac{y(\bar{p}_{i,j} - \bar{p}_{o,j})}{l \rho \bar{v}_{i,j}^2} \right), \quad (4.30)$$

$$\Psi_j = \frac{1}{Nu_{fd}} \frac{\dot{Q}_j y}{\lambda A_{ref} (\bar{T}_{w,j} - \bar{T}_{b,j})}. \quad (4.31)$$

Figures 4.21a and 4.22a show that, for both fin geometries, there is a strong increase of the friction factor for the first unit module, equal to three times the fully developed value for WF1 and four times for WF4. On the other hand, for the next four to six modules, f decreases with respect to f_{fd} , and the lowest peaks are found for the lowest among the Re -values considered. For both geometries, the friction factor becomes stable around the tenth single fin module. In both cases, in correspondence of the thirtieth module, f grows slightly, due to the sharp edge present at the outlet of the fin stack. The variation of the Reynolds number does not cause significant changes of entrance effects on the friction factor.

Figures 4.21b and 4.22b show the variation of Nusselt number in the entrance region for WF1 and WF4, respectively. In both cases, Nu increases with each fin module until it reaches its fully developed value near the tenth fin module. WF1 shows a greater sensitivity to the Reynolds number compared to WF4, and in both cases, as Re_y increases entrance effects become less relevant. The decreased heat transfer effectiveness observed in the entrance region, is a result of the misalignment between the flow and the fin walls at the inlet of the channel, which leads to the formation of large separation bubbles that reduce local

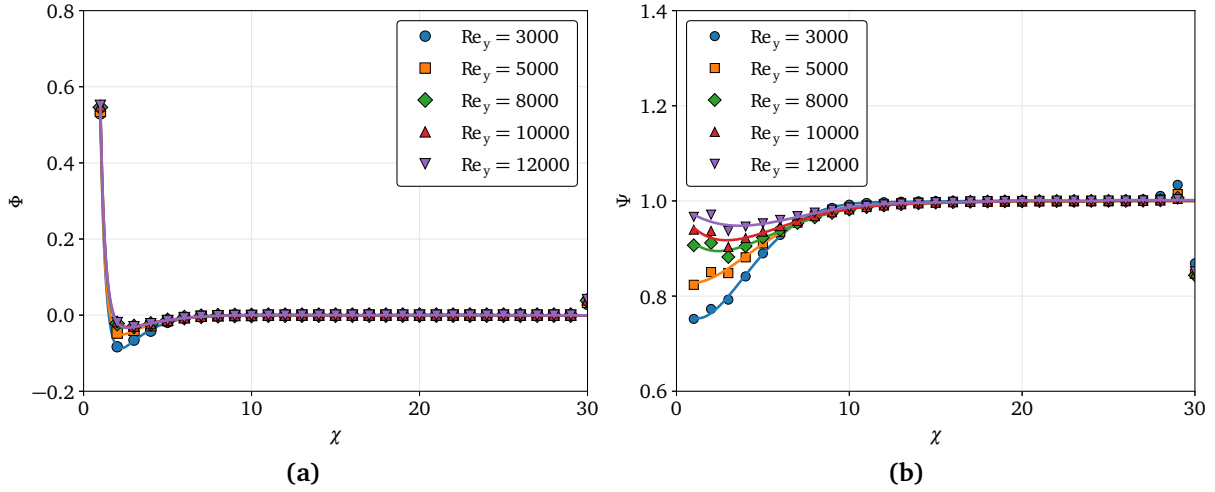


Figure 4.21. WF1 entrance region: distributions of relative friction factor (a) and Nusselt number (b).

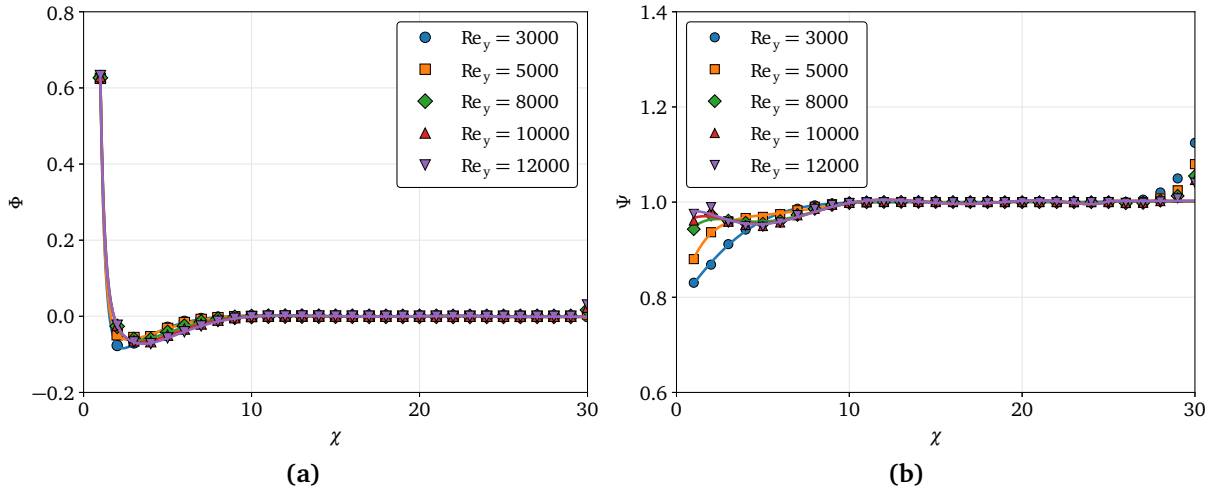


Figure 4.22. WF4 entrance region: distributions of relative friction factor (a) and Nusselt number (b).

heat transfer effectiveness in the entrance region. In general, these distributions of Nusselt number show opposite trends to those usually observed for straight-walled channels and fins, in which Nu is higher in the entrance region and decreases as the flow develops.

The results of the CFD analyses are employed to derive correlations that express the variations of f and Nu with respect to their fully developed values as a function of the module number χ and Reynolds number. The friction factor and Nusselt deviations due to outlet effects are not considered.

$$\Phi = \sum_{k=0}^7 \sum_{r=0}^4 d_{kr} Re_y^r \log^k \chi \quad (4.32)$$

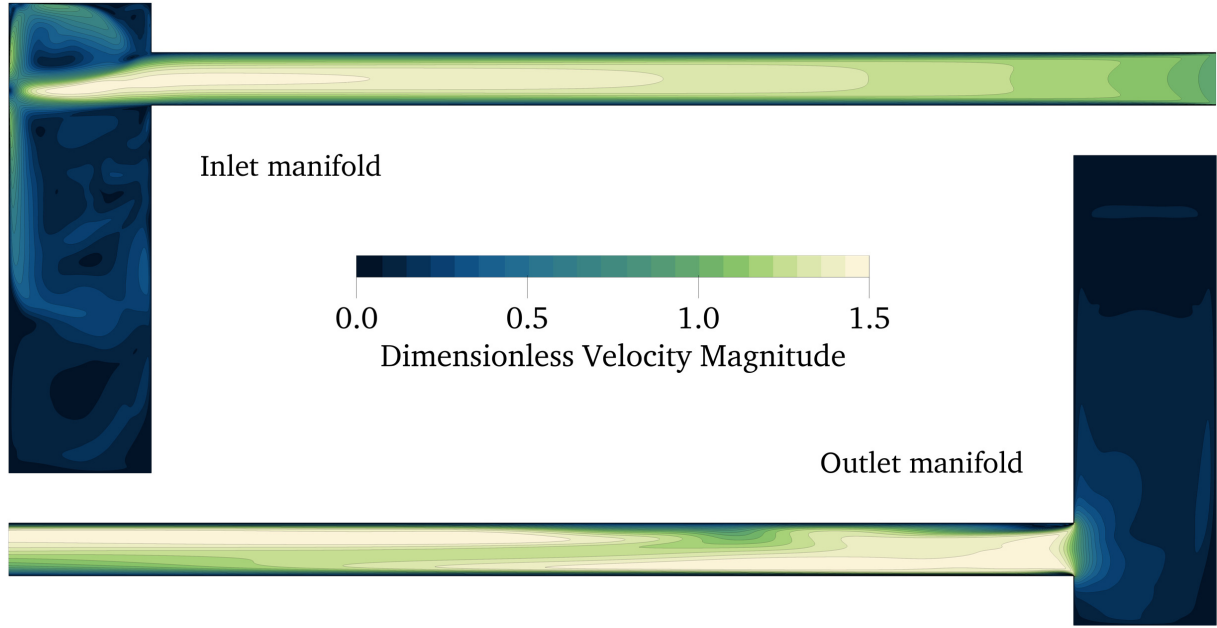


Figure 4.23. Global head losses model: colour maps of dimensionless velocity magnitude of inlet and outlet manifolds of O94 on vertical cross-sections.

$$\Psi = \sum_{k=0}^7 \sum_{r=0}^4 e_{kr} \text{Re}_y^r \chi^k \quad (4.33)$$

The values of the coefficients d_{kr} and e_{kr} have been omitted for the sake of brevity, while the correlations, along with the CFD results, are displayed in the plots of Figures 4.21 and 4.22.

4.5.3. Head losses in inlet and outlet manifolds

Figure 4.23 displays contour plots of dimensionless velocity at $\text{Re}_{D_p} = 2200$, related to the O94 heat exchanger. These colour maps are obtained on cross-sections located at the mid-planes of the inlet and outlet pipes, respectively. In the inlet tank, the fluid enters at high speed, impinging the wall opposite the entrance. The redistribution of the fluid across the wall generates two large-scale vortices. Fluid velocity is much more uniform in the outlet manifold, while near the outlet pipe it increases sharply, forming two recirculation regions, which cause major head losses, by restricting the effective outflow section.

Head losses associated with inlet and outlet manifolds are computed by means of the numerical models presented in Section 4.3.3, considering the integral mean values of the pressure field evaluated on cross-sections corresponding to the connection between the channels and the manifolds, and the locations of the pressure taps positioned in the inlet and outlet pipes of the heat exchangers during the experimental tests. The resulting values of Δp_i and Δp_o are employed to compute the concentrated loss coefficients of the inlet and outlet manifolds:

$$\beta_i = \frac{2 \Delta p_i}{\rho \bar{v}_i^2}, \quad \beta_o = \frac{2 \Delta p_o}{\rho \bar{v}_o^2}. \quad (4.34)$$

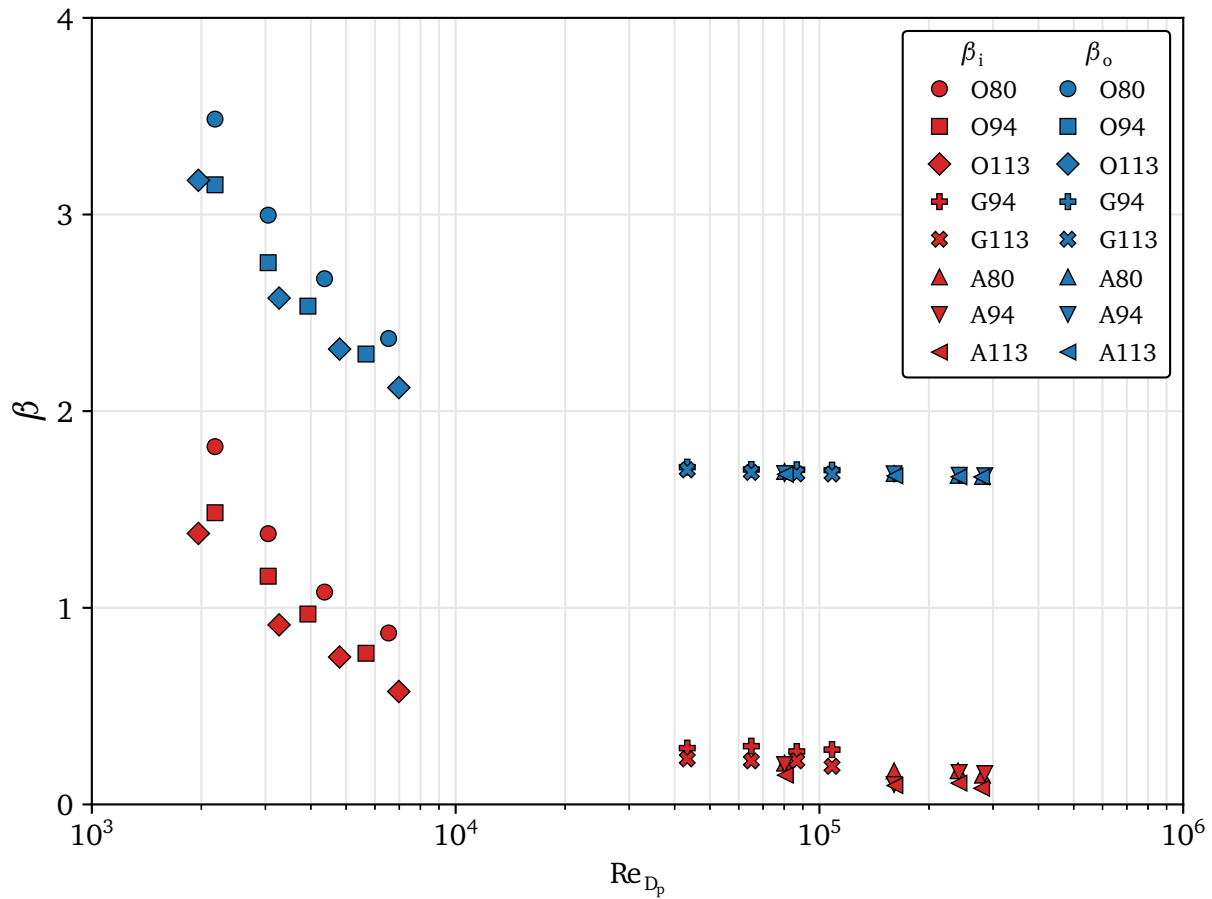


Figure 4.24. Global head losses model: concentrated loss coefficients of the inlet (red) and outlet (blue) manifolds.

Figure 4.24 reports the β_i and β_o values as a function of Re_{D_p} : at low Reynolds values, which correspond to the inner flows of oil coolers, the loss coefficients decrease with increasing Re_{D_p} and cooler thickness S . On the other hand, at high Re-values, which correspond to the flow rates of the water radiators and charge air coolers, both concentrated loss coefficients remain almost unchanged with varying Re_{D_p} and S . The values of the concentrated loss coefficients are used to derive suitable correlations, by following an analogous approach to the one employed in [238]:

$$\beta = k_1 Re^{k_2} + k_3, \quad (4.35)$$

where the coefficients k_i are computed by combining the ratio between the thickness of the

Table 4.9. Values of q_{ij} coefficients in Equation (4.36).

Coefficient	q_{11}	q_{12}	q_{21}	q_{22}	q_{31}	q_{32}
β_i	2385	-1.580	-0.826	-0.076	0.196	-0.509
β_o	272.2	1.791	-0.621	0.375	1.602	0.022

manifolds S and the diameter of their inlet and outlet pipes D_p with the q_{ij} values reported in Table 4.9.

$$k_i = q_{i1} \left(\frac{S}{D_p} \right)^{q_{i2}} \quad (4.36)$$

4.5.4. Global models

4.5.4.1. Heat transfer

Figure 4.25 shows the comparison between the experimentally obtained heat transfer rates and the numerical results from the global model for heat transfer, as described in Section 4.3.4.1. The remarkable agreement between the numerical and experimental data found for charge air coolers (Figures 4.25f, 4.25g, and 4.25h) demonstrates the validity of the selected modelling approach. Greater maximum deviations of $\pm 10\%$ and $\pm 20\%$ are found for water radiators and oil coolers, respectively. O94 features the highest deviations, reaching $\pm 38\%$. These deviations can be traced back to the fact that the models developed in this work are derived purely from nominal geometric data, physical principles, and established models. On the other hand, possible fouling or shape imperfections are not taken into account.

4.5.4.2. Internal head losses

The internal head losses, computed according to the approach presented in Section 4.3.4.2, are compared with the available experimental data in Figure 4.26. The results related to oil coolers and water radiators are remarkably accurate as, for the majority of the flow rates tested, the difference between the numerical and experimental results is less than $\pm 10\%$. On average, the results on charge air coolers are much less accurate, as in the majority of the tested cases the deviation between the numerical and experimental results is close to $+20\%$, moreover there are two outliers which show deviations of $+120\%$ and $+185\%$. The higher deviations found in the analyses of charge air coolers suggest that the proposed methodology may have some limitations, which could be addressed by performing CFD analyses of the complete heat exchangers, including the computation of the temperature field, along with an ideal gas model for the working fluid. This approach could lead to more accurate results, but at the cost of much greater computational effort and complexity.

Figure 4.27 shows the values of pressure drop associated with the inlet and outlet manifolds, and the internal channels expressed as a fraction of the total head losses. In the case of oil coolers ($2000 \leq Re_{D_p} \leq 7000$), the major contribution to the total losses is given by the internal channels, while both manifolds play a minor role. As the Reynolds number increases, the contribution of the outlet manifold increases dramatically until it becomes the main cause of head losses for water radiators and charge air coolers. Conversely, the contribution of internal channels decreases until it becomes of the same order of magnitude as the one of the inlet manifold, which slightly grows with increasing Re_{D_p} . As the thickness of the

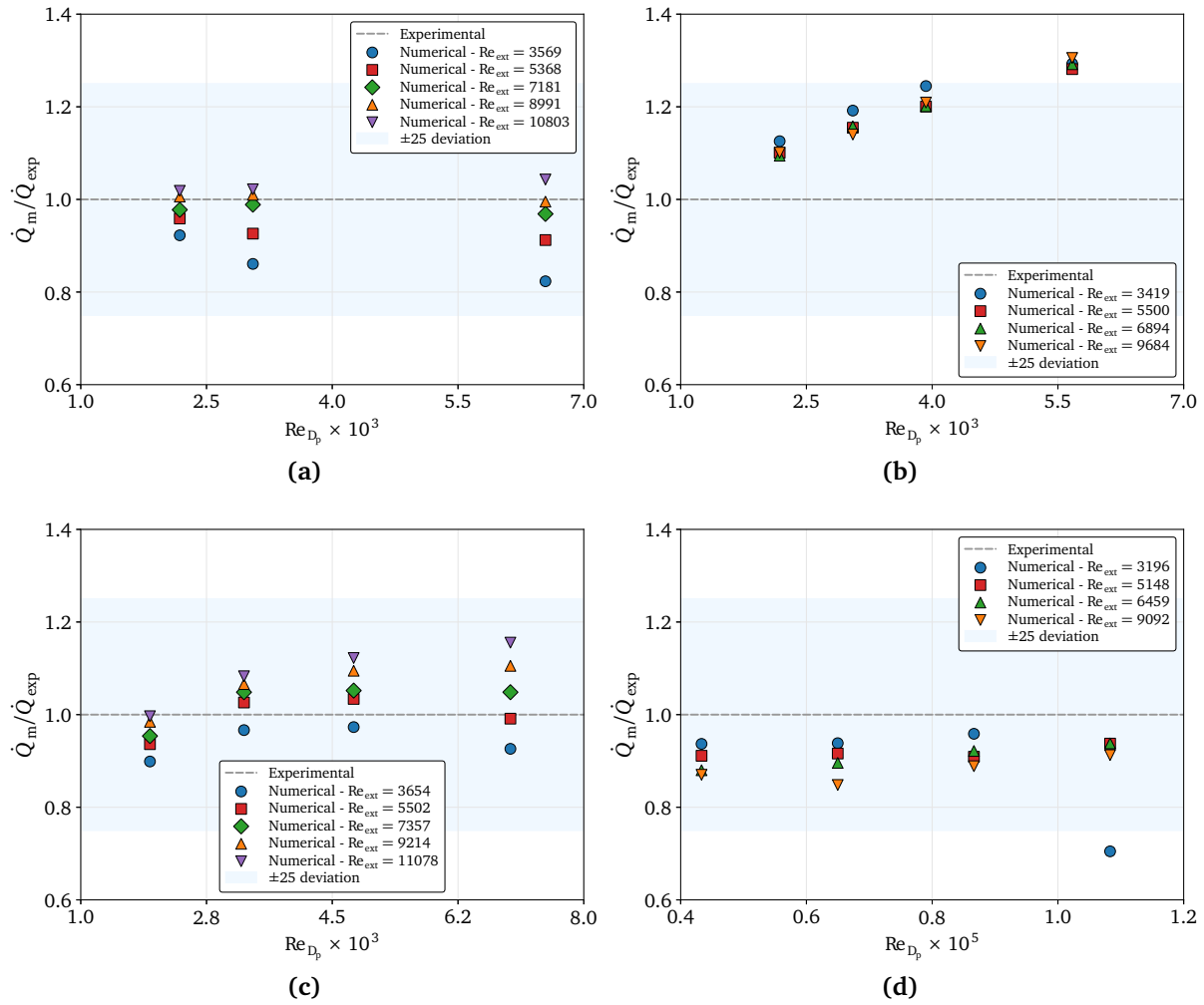


Figure 4.25. Global thermal model: comparison between the numerical results and available experimental data for O80 (a), 094 (b), 0113 (c), G94 (d) coolers.

heat exchanger, S , increases, the contribution of the outlet manifold grows, resulting in a corresponding decrease in the contribution of the internal channels. Meanwhile, the relative loss associated with the inlet manifold remains almost unchanged with varying S . Similarly, it can also be noted that the large fluid density variation in charge air coolers tends to increase the contribution of the outlet manifold at the expense of the internal channels one. The relative loss of the inlet manifold is less affected by the large temperature difference. These results underline the importance of considering the head losses of the manifolds when carrying out optimizations on complete heat exchangers, as these losses alter the relative weight between the heat transfer effectiveness and head losses in the internal channels. For instance, in charge air coolers, optimising the geometry of the internal fins to reduce head losses of the core at the expense of heat transfer effectiveness may not be advantageous, given that roughly 90% of the total head losses are given by the manifolds.

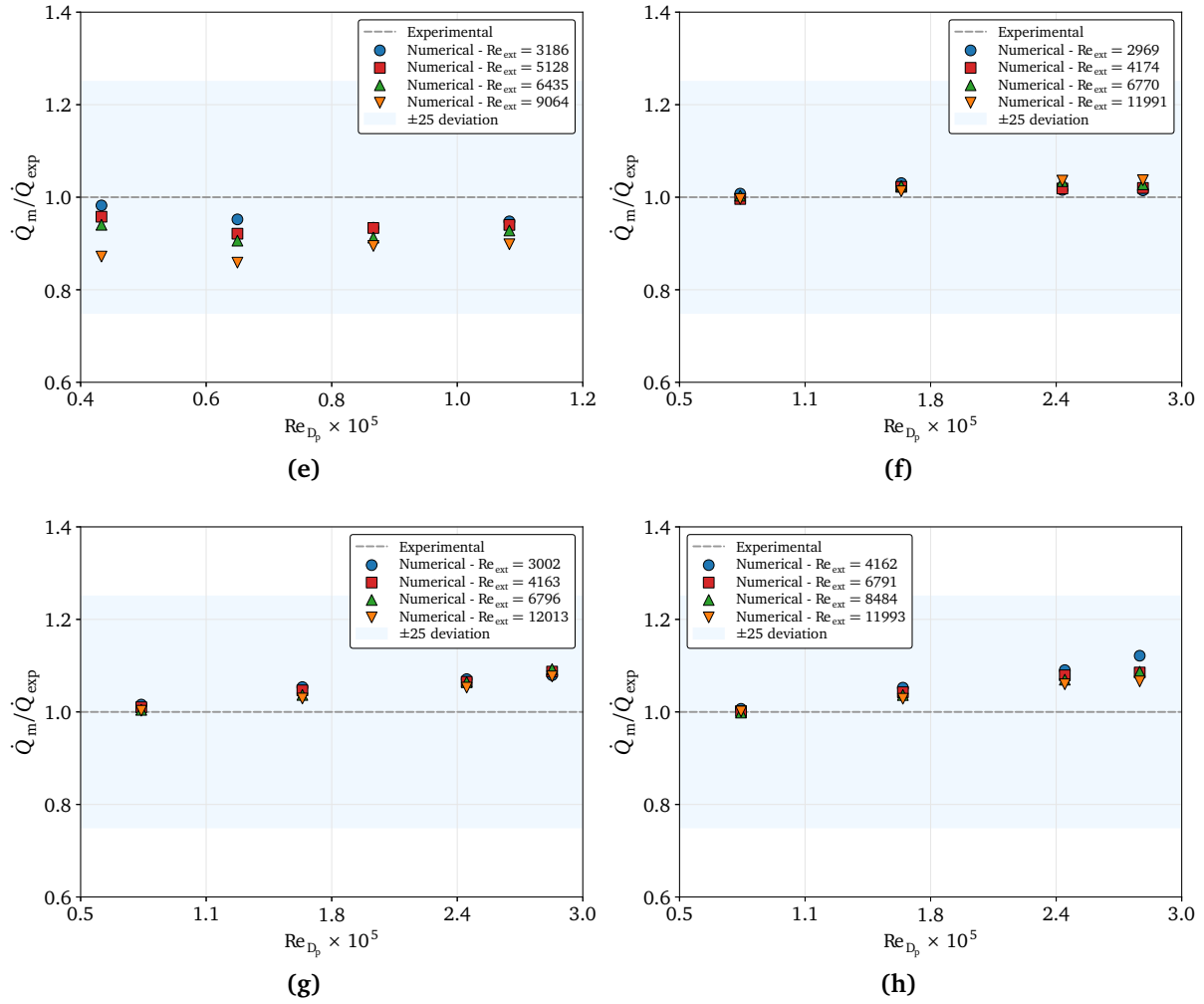


Figure 4.25. Global thermal model: comparison between the numerical results and available experimental data for G113 (e), A80 (f), A94 (g), and A113 (h) coolers.

4.5.4.3. External head losses

Figure 4.28 shows the comparison between the numerical values of external head losses and the experimental ones. For most of the tested cases the deviation between the numerical and experimental results is less than $\pm 25\%$. Furthermore, many of the heat exchangers considered feature an average deviation of $\pm 10\%$ across the tested flow rates. The greater uncertainties observed in some cases may be due to the fact that, in the experimental test bed, the outlet of the external side of the heat exchangers is in free air, as seen in Figure 4.8a. This can lead to uncertainties in the measurement of the outlet air temperature and pressure. Overall, these results confirm the validity of the proposed approach.

4.6. Concluding remarks

In this work an original multiscale modelling approach for plate-fin heat exchangers was developed purely on the basis of nominal geometric parameters, physical principles, and established models. The proposed methodology was presented and validated through the

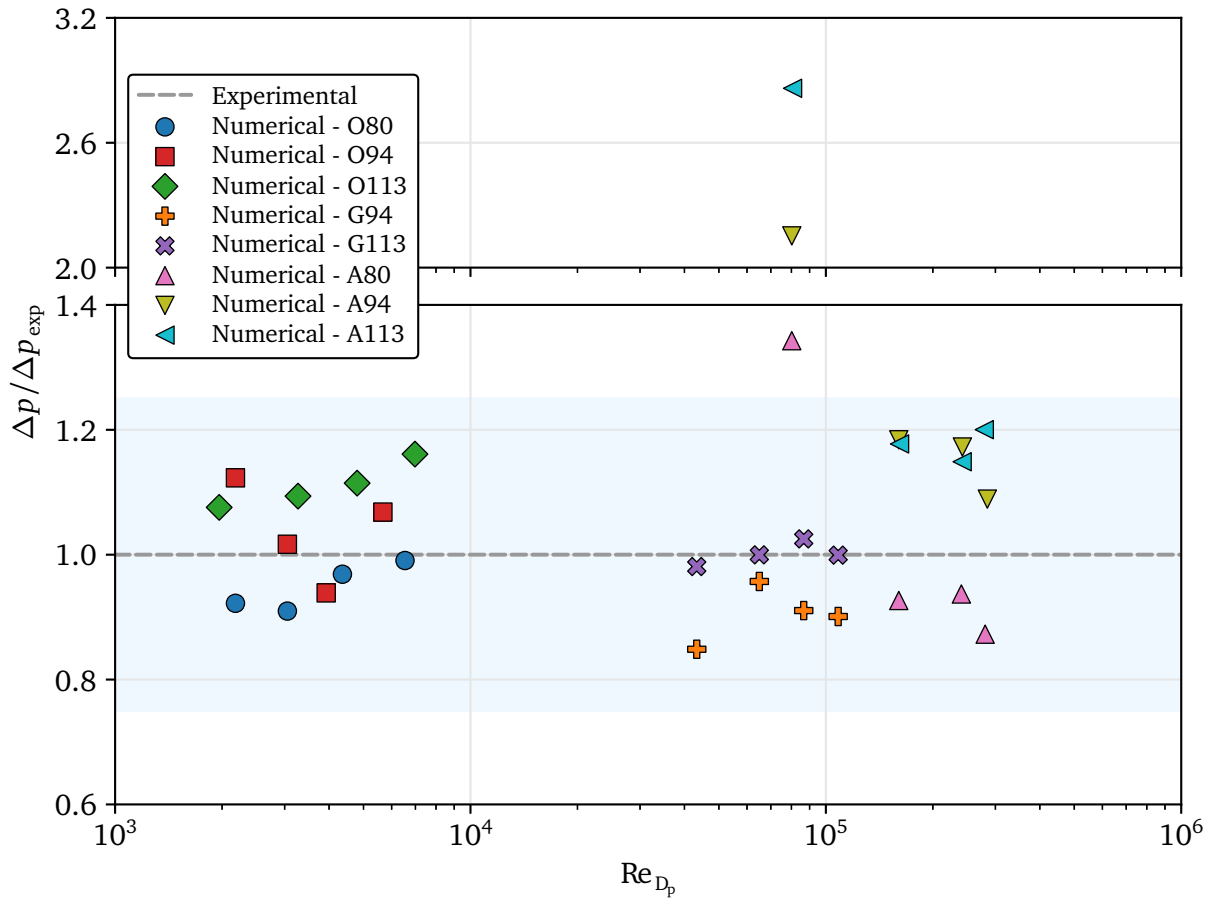


Figure 4.26. Global head losses model, internal flow: comparison between the numerical results and available experimental data for all the considered heat exchangers.

analysis of a case study comprising several heat exchangers encompassing different geometric configurations and working fluids. These cooler were tested at an external facility, to determine their values of heat transfer rates and head losses for several flow conditions.

The internal and external finned surfaces were studied by developing CFD models of single fin modules under the hypothesis of fully developed flow. In addition, entrance effects in external channels were analysed by means of dedicated CFD models. Overall models for the thermal-hydraulic performance of plate-fin heat exchangers were developed on the basis of the numerical results of the extended surfaces. The overall model for heat transfer was derived on the basis of $\varepsilon - NTU$ method using heat transfer coefficients from the numerical analyses. Internal and external head losses were computed by means of the Darcy-Weisbach formula using the numerically obtained values of the Darcy friction factor. In addition, numerical analyses were performed to determine head losses in the manifolds, an in-house numerical method was devised for the derivation of internal losses in charge air coolers.

Numerical results on extended surfaces were compared with selected correlations from the literature, showing that, although quite a large amount of data is available on these fin geometries, dedicated numerical analyses are still necessary to obtain accurate results for fin configurations used in actual industrial applications. The presented approach was

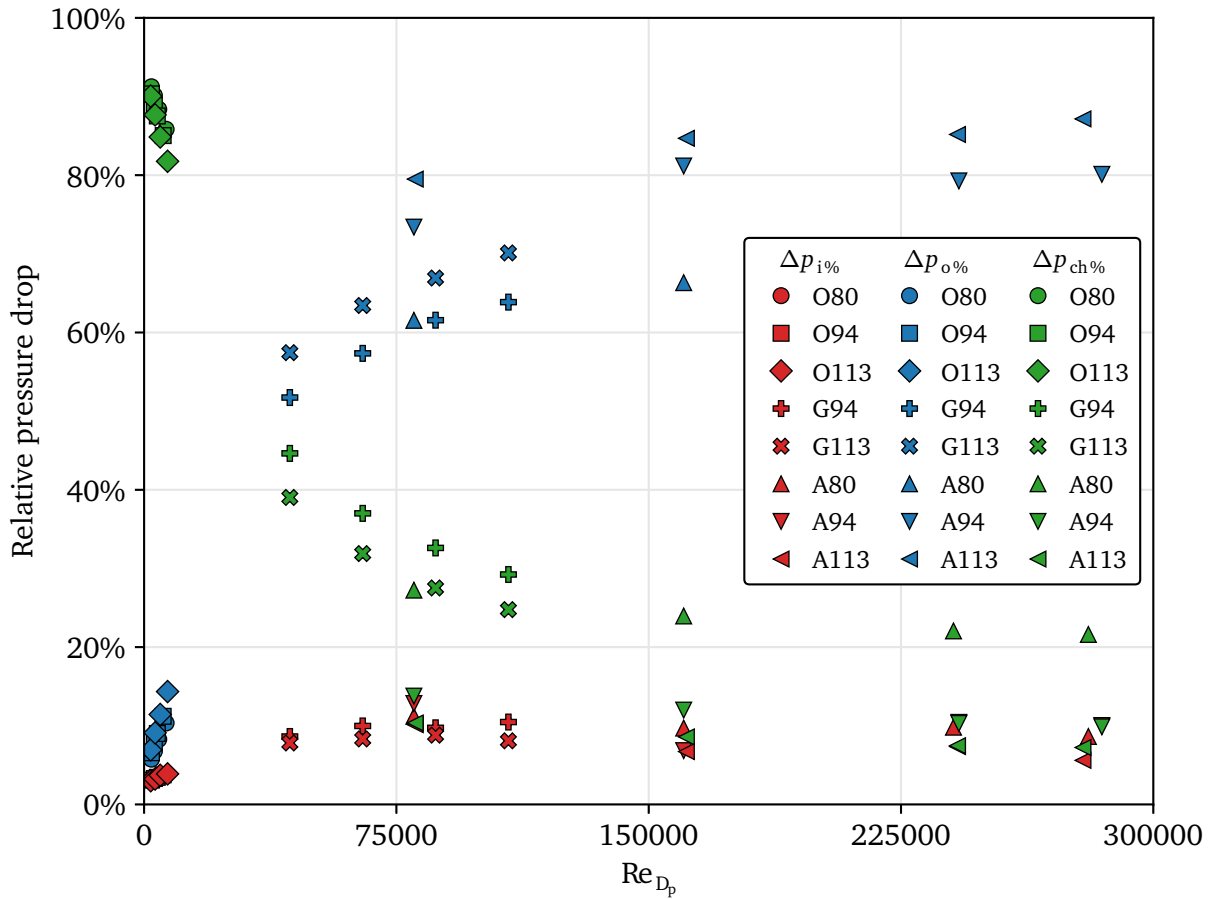


Figure 4.27. Global head losses model, internal flow: relative pressure drops of the inlet manifold (red), the outlet manifold (blue) and the internal channels (green).

validated by comparing numerical results from the overall model with the available experimental data, showing a remarkable agreement, thus confirming the validity of the proposed approach. In addition, the comparison between the sources of internal head losses showed that in oil coolers, operating at low values of Reynolds numbers, almost all the losses are caused by the internal channels. Conversely, in water radiators and charge air coolers, typically operating at high Reynolds number values, 90% of the total losses are given by the manifolds, underlining the importance of considering this contribution during verification and optimisation procedures of these types of coolers.

The approach presented in this work is an improvement over common modelling techniques, as it retains the simplicity of a global modelling procedure, while achieving levels of accuracy which are typical of higher-order methods. Furthermore, the adopted workflow can be practically applied to heat exchangers of any architecture.

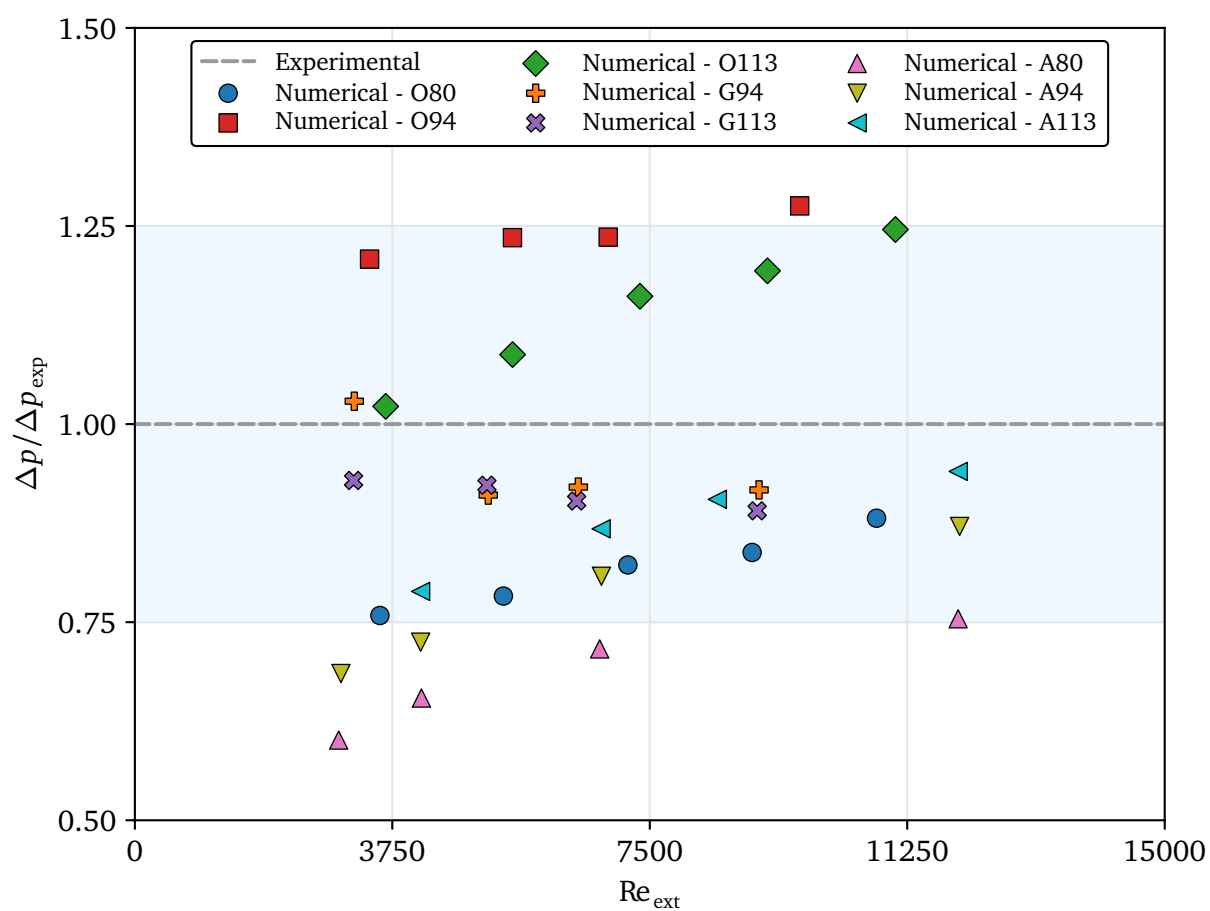


Figure 4.28. Global head losses model, external flow: comparison between the numerical results and available experimental data for all heat exchangers considered.

CHAPTER 5

Parametric analysis of offset-strip fins

5.1. Introduction

5.1.1. Engineering and technical background

In today's industrial landscape, improving the efficiency of thermal processes and cycles is of paramount importance, due to rising energy costs and environmental concerns. Heat exchangers play a critical role in most industrial thermal processes and improving their performance through advanced design techniques is of increasing technical interest [239]. In compact heat exchangers used for the thermal management of specific systems, such as components of electric power trains in automotive applications and power generation units, Offset-Strip Fins, commonly referred to by the acronym OSF, are among the most used extended surfaces. OSFs, compared to conventional straight channels, feature augmented heat transfer effectiveness, which is mainly achieved by the periodic disruption and regrowth of boundary layers along the fin walls. Secondary contributions to the heat transfer enhancement are given by a slightly increased surface area, and by the periodic vortex shedding at the trailing edges of the fins, which favours fluid mixing. Of course, the improved heat transfer performance is accompanied by an increase in the head losses, which can be significant in certain configurations. Therefore, it is essential to select the most appropriate geometric parameters in order to obtain acceptable compromises between heat transfer effectiveness and pressure losses [240].

This is an intricate task, as the geometry of offset-strip fins is defined by a wide number of parameters including, but not limited to, the fin pitch x , the fin height y , the offset length l , and the wall thickness t . Further complications arise from the large variety of fluids that can be used with these fins. These include air and other gases which are typical

NOMENCLATURE*Latin letters*

A	area [m^2]
b_m	coefficients in Equations (5.14) and (5.15)
c	vertical crush [m]
c_p	specific heat [$\text{J kg}^{-1} \text{K}^{-1}$]
C	heat capacity rate [W K^{-1}]
d_m	coefficients in Equations (5.37) and (5.38)
D_h	hydraulic diameter [m]
f	equivalent Darcy friction factor [—]
H	heat exchanger height [m]
j	Colburn factor [—]
k_n	coefficients in Equations (5.16), (5.39), and (5.40)
l	offset length [m]
L	heat exchanger core length [m]
n	Prandtl number scaling exponent [—]
N_{ch}	number of internal channels [—]
Nu	Nusselt number [—]
NTU	number of transfer units [—]
p	fluid pressure [Pa]
P	wetted perimeter [m]
Pr	Prandtl number [—]
q	heat flux [W m^{-2}]
$\dot{Q}$	heat [W]
r	external corner radius [m]
Re	Reynolds number $u y / \nu$ [—]
s	smoothing function
S	heat exchanger thickness [m]
St	Stanton number [—]
t	fin thickness [m]
T	temperature [K]
$\vec{u}$	fluid velocity [m s^{-1}]
U	overall heat transfer coefficient [$\text{W m}^{-2} \text{K}^{-1}$]

 W manifold width [m] x fin pitch [m] y fin height [m]*Greek letters*

β	concentrated loss coefficient [—]
β_r	blockage ratio [—]
ε	effectiveness [—]
λ	thermal conductivity [$\text{W m}^{-1} \text{K}^{-1}$]
ν	kinematic viscosity [$\text{m}^2 \text{s}^{-1}$]
ρ	density [kg m^{-3}]
ϑ	dimensionless temperature difference [—]
σ	smooth step sharpness
χ	generic independent variable

Subscripts

b	bulk
ch	channel
ext	external
f	fluid
F	Fanning
he	heat exchanger
i	inlet
int	internal
l	laminar
o	outlet
ov	overlap
rs	response surface
s	solid
t	turbulent
w	wall

Superscripts

*	dimensionless
---	---------------

of the aerospace and power generation industries, as well as water mixtures and hydraulic or lubricating oils, which are more common in heavy-duty vehicle applications, such as agricultural and earth-moving equipment.

Although over the years many studies have been carried out on staggered fin arrays, the resulting predictive models can be considered valid and reliable only within narrow ranges of dimensional and flow parameters. For this reason, the present analysis is aimed at the integration of the results obtained thus far through a Design Of Experiment (DOE) procedure, carried out by means of computational techniques. The main goal of this work consists in obtaining a set of continuous response surfaces for the equivalent Darcy friction factor f and the Colburn factor j , with suitable accuracy levels over a comprehensive part of the geometric and flow parameter space. These response surfaces, in the form of multivariate correlations, could also be employed for the geometric optimisation of heat exchangers for a wide range of industrial applications. Furthermore, this work provides for the first time a derivation of a Prandtl number scaling exponent, which is specific to offset-strip fins, and computed as a function of the geometrical parameters.

5.1.2. Literature survey

One of the most relevant texts regarding the design of heat exchangers is the one by Kays and London [202], in which the authors provided a large experimental data set for large variety of fin geometries, including offset-strip fins. Analogous experimental data about actual fin cores can be found in [203]. The first predictive models for the thermal-hydraulic performance of OSFs were introduced by Manson [241]. Other very well-established correlations are those proposed by Wieting [204], which consist of piecewise defined power laws obtained from experimental tests on 22 fin cores carried out in laminar and turbulent regimes. Wieting suggested extrapolating the laminar and turbulent curves to determine the values of f and j in the transition region. Joshi and Webb [208] developed an analytical model for the determination of heat transfer and friction performance of offset-strip fins in laminar flow, and a semi-empirical model for turbulent flow. Both models accounted for the thickness of fin walls, and for microscopic geometric features such as surface roughness and imperfections that arise from the manufacturing process. These models featured remarkable accuracy levels ($\pm 20\%$), however they also featured a complex formulation, so the authors introduced more convenient power laws, analogous to the ones proposed by Wieting [204], but with improved accuracy thanks to a larger database, and an empirical relation for the transition Reynolds number as a function of the geometrical parameters. Joshi and Webb conducted their experiments with an aqueous ethylene glycol solution as the working fluid, and considered additional data about air flows available in the literature, thus the derived correlations can be considered reasonably valid over the range $0.71 \leq \text{Pr} \leq 7$. Mochizuki *et al.* [205] tested several offset-strip and slotted fin cores, considering air as the working fluid, and developed correlations in the form proposed by Wieting [204]. A similar work is

the one by Dubrovsky and Vasiliev [207]. Manglik and Bergles [206] presented a revision of the data available in the literature by attempting to unify the definitions of hydraulic diameter and heat transfer area, as these are not given in some works or their formulation appears unclear. The authors focused on the transition region and concluded that, for their configurations, the equivalent friction factor and the Colburn factor vary continuously with Re and that there is no noticeable discontinuity in the transition region. Finally, the authors mentioned the problem of a high-frequency sound that can be emitted by some fin cores at certain flow regimes, but did not elaborate further. Similar high-intensity tonal noise emissions were observed by Roesler *et al.* [242]. Dong *et al.* [243] obtained accurate correlations ($\pm 10\%$ deviations) by introducing an additional parameter to account for entrance effects in the fin cores. Kim *et al.* [213] investigated offset-strip fin performance by means of computational methods. The authors tested several turbulence models based on the eddy viscosity approach, including the standard $k-\varepsilon$, realizable $k-\varepsilon$, and the $k-\omega$ SST. The latter was found to be the most accurate with respect to the experimental results presented by Manglik and Bergles [206]. In addition, the $k-\omega$ SST turbulence model showed promising accuracy levels when used to model oscillatory flows with steady state simulations [244]. Kim *et al.* [213] also introduced an additional geometric parameter called blockage ratio which indicates the portion of the fin frontal area occupied by the fin walls. The authors also investigated the effects of the Prandtl number on heat transfer performance, conducting CFD analyses with Pr -values ranging from 0.72 to 50. Significant variations of the Colburn factor were found, suggesting that using the traditional Pr exponent (1/3) for the computation of the Colburn factor is not an adequate approximation. Yang and Li [209] studied entire offset-strip channels by means of steady-state CFD analyses to account for entrance effects. The authors used the blockage ratio like Kim *et al.* [213], but with a slightly different definition. On the basis of these correlations, Yang *et al.* [245] proposed an original approach based on relative entropy generation to determine optimal OSF geometric and flow parameters. In an effort to obtain improved estimates of OSF performance, Yang *et al.* [246] introduced the actual fin efficiency parameter to overcome the major limitations and simplifying assumptions introduced by the classical fin efficiency definition [58]. Peng *et al.* [247] proposed an innovative offset-strip fin architecture obtained by slightly bending an array of plain rectangular channels. Michna *et al.* [248] investigated the thermal-hydraulic performance of OSFs at very high Reynolds number values $10000 \leq Re_{D_h} \leq 120000$, associated with automotive and aerospace applications where maximum compactness is sought at the expense of increased fan pumping power. Significant deviations were observed when these results were compared with those obtained by extrapolating the correlations of Manglik and Bergles [206] and Muzychka and Yovanovich [249].

Several authors investigated the effect of the Prandtl number of the working fluid on the heat transfer performance of offset-strip fins. Tinaut *et al.* [250] carried out experiments with lubricating oil, concluding that for such fluids the Prandtl number scaling exponent is

closer to 0.4, as supposed to the traditionally used value of $1/3$. Hu and Herold [251, 252] tested several working fluids with Prandtl number values ranging from 3 to 150, and showed that the use of correlations developed on the basis of gas flows can lead to significant overestimations of the Colburn factor, possibly causing significant design errors. Guo *et al.* [253] developed correlations for heat transfer and pressure losses for carbon steel OSFs operating with lubricating oil, concluding that the thermal-hydraulic performance is mainly influenced by the aspect ratio and offset length [254]. In a following contribution [255] the authors investigated the effects on heat transfer performance given by the flow angle, which provided maximum improvement at 45° . Xiang, Du, and Zhao [256] developed empirical correlations from numerical analyses performed with Pr-values ranging from 2 to 13, considering the effects of fin offset, unlike most authors. Du *et al.* [257] employed a Genetic Algorithm to optimise the performance of an oil-to-air heat exchanger equipped with trapezoidal OSFs. Furthermore, the authors compared the j and f values of the optimised solution with those from the models by Wieting [204] and Manglik and Bergles [206], noting significant deviations attributed to the adoption of a trapezoidal cross-section and the high Pr-value of the considered working fluid. Recently, Kedam *et al.* [258] proposed an empirical predictive model based on an Artificial Neural Network (ANN), that was trained on a set of heat transfer and pressure loss data in air flows. This modelling approach allowed for the prediction of j and f values with deviations of 1%-2% with respect to the data points. Similarly, Chandan *et al.* [259] employed a stacking ensemble modelling approach and machine learning algorithms to derive a predictive model for the friction factor of OSF-equipped micro channels, with a mean absolute error of 0.403%.

Offset-strip fins are also widely used in multiphase flows, such as refrigerant cycles in electric vehicles heat pumps [260], as they offer significant improvements in terms of heat transfer effectiveness compared to plain channels [261]. Several authors investigated evaporation and condensation phenomena in OSFs considering different refrigerants such as R133 [262], HFE-7100 [263], R134a [264], and others [265–268].

The careful analysis of the literature shows that in almost every work, different definitions are adopted for the hydraulic diameter, the heat transfer area and the other geometric parameters, and it is easy to fall into errors because the same symbols are used to describe different quantities. This makes the comparison between results difficult and sometimes impossible, as some authors do not specify how they computed the reference velocity, the hydraulic diameter and the heat transfer area. Therefore, it is necessary to be cautious and pay attention to what each research group identifies as reference quantities. Furthermore, it is clear that heat transfer performance is strongly influenced by the working fluid, i.e. by its Prandtl number [211], but up to now a single and robust approach to address this problem has not been proposed yet. Finally, it is interesting to note how the proposed correlations become increasingly more complex as new parameters are added through the years, suggesting that no definitive conclusions have yet been drawn on the heat transfer and friction

properties of offset-strip fins, and that there is still ample room for new contributions to the subject.

5.1.3. Modelling approach

The present research deals with the analysis of the single-phase thermal-hydraulic performance of offset-strip fins with straight vertical walls, centred or symmetric offset, and filleted corners. The basic geometric parameters such as fin pitch, fin height, and wall thickness are defined as in the work by Yang and Li [209], but in this case different symbols are used to avoid any ambiguity. In addition, analytical expressions for hydraulic diameter and heat transfer area are provided to allow for their exact computation.

A Design of Experiment analysis of heat transfer performance and pressure losses is carried out by means of computational techniques. The values of geometric and flow parameters are selected considering the most recurrent configurations in real industrial applications. A full factorial DOE is performed with respect to the dimensionless fin pitch, the offset length, and the flow regime, which is defined by pairs of Reynolds and Prandtl numbers. The resulting configurations are studied by means of CFD analyses of basic fin modules obtained by exploiting the periodicity of the geometry and of the flow along the streamwise and spanwise directions. The solution to the incompressible and periodic flow is attained by means of a validated conjugate heat transfer solver, based on a Finite Volume discretisation of the governing equations. Integral results on pressure losses and heat transfer effectiveness are presented by an equivalent Darcy friction factor and average Nusselt number, which are based on the fin height. These values are used to derive separate response surfaces for laminar and turbulent flow, which are then combined by means of a smoothing hyperbolic tangent function. The effect of the Prandtl number on heat transfer performance is managed by introducing the Colburn factor, and by computing the Pr scaling exponent as a function of the geometric parameters of the fins by performing dedicated CFD analyses. The numerical results and response surfaces are compared with some of the most relevant correlations available in the literature for a selected group of geometric configurations.

Finally, the response surfaces obtained in this work are validated by computing the internal head losses and overall heat transfer rates of three oil coolers, which feature offset-strip fins within the internal channels. These results are compared with the available experimental data and numerical results obtained in a previous work, by performing dedicated CFD analyses for the examined fin geometry.

5.2. Problem statement

This Chapter is focused on offset-strip fins with straight vertical walls and symmetric or centred offset. The front and top views of the examined offset-strip fin geometry are depicted in the drawings of Figure 5.1, along with the key dimensional parameters. The fin pitch, denoted by x , is the distance between two consecutive vertical walls, including the thick-

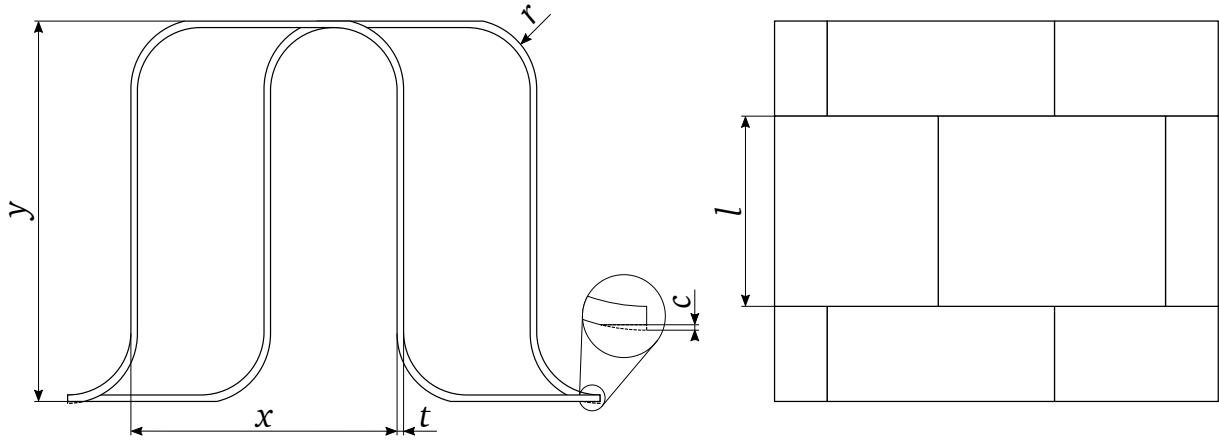


Figure 5.1. Front and top views of the examined offset-strip fin geometry, along with the most significant dimensions.

ness of one of the two walls t . The external corner radius between vertical and horizontal fin walls is denoted by r , while the fin height is identified by y . During the production process of compact heat exchangers the fins undergo mechanical deformations as a result of the stacking of finned plates and brazing sheets. The most marked effect consists of a vertical crushing, which is reproduced by cutting the outer fillet radii by an amount c , so that they are no longer tangent to the horizontal walls. c is set to 0.5% of the fin height to introduce a small deformation, that is not too excessive for the configurations with small pitch values. The centred offset constraint determines a relationship between fin pitch, the maximum value of the external fillet radius, and wall thickness:

$$x = 4r_{\max} - 2t. \quad (5.1)$$

The flow cross-sectional area of the considered OSF geometry equates to:

$$A_f = 2xy - A_w, \quad (5.2)$$

where A_w stands for the portion of frontal area occupied by the fin walls:

$$\begin{aligned} A_w = & 2r^2 \left[\arcsin\left(\frac{c}{r}\right) - \frac{\pi}{2} - \arcsin\left(\frac{c-r}{r}\right) \right] \\ & + 2c \left[\sqrt{r^2 - c^2} - (c-r) \sqrt{\frac{2r}{c} - 1} \right] \\ & + 2t \left(y - \frac{\pi+2}{2} t + \pi r \right). \end{aligned} \quad (5.3)$$

The wetted perimeter is given by:

$$P = 4 \left\{ x + y + r \left[\pi - 2 - \arcsin\left(\frac{\sqrt{2rc - c^2}}{r}\right) \right] \right\} - 2\pi t - 4\sqrt{2rc - c^2}. \quad (5.4)$$

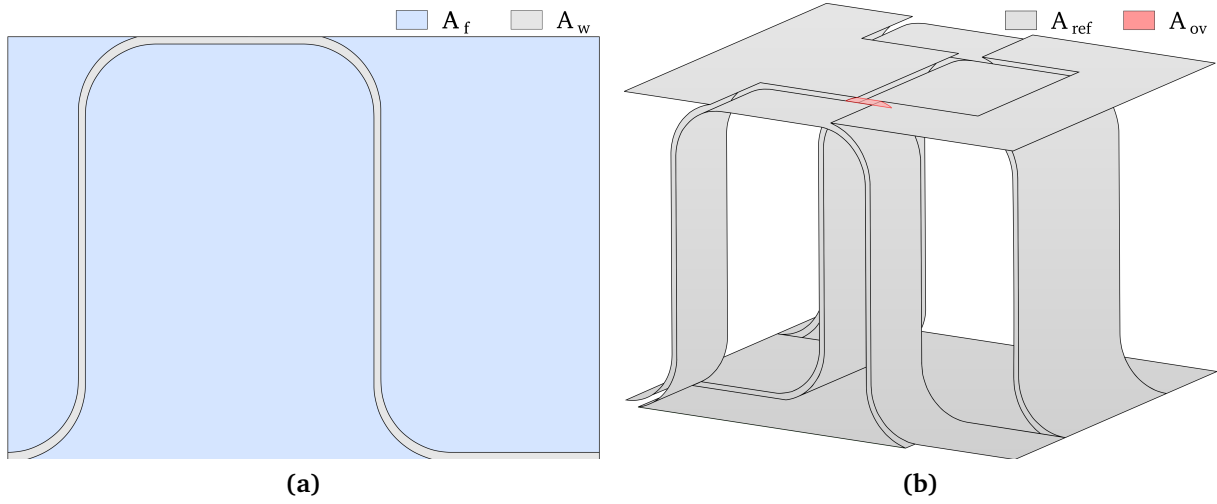


Figure 5.2. Flow cross-sectional area (a) and heat transfer area (b) of the examined offset-strip fins.

The heat transfer area A_{ref} consists of the whole fluid-solid interface of the fins, including leading and trailing edges, along with the area wetted by the fluid on the upper and lower horizontal surfaces:

$$A_{ref} = 2lP + 4(A_w - 2A_{ov}), \quad (5.5)$$

where A_{ov} is the portion of fin walls frontal area given by the overlap of two successive single fins.

$$\begin{aligned} A_{ov} = r^2 & \left[\arcsin\left(\frac{c+t-r}{r}\right) - \arcsin\left(\frac{c-r}{r}\right) \right] \\ & + (c+t-r) \sqrt{(t+c)(2r-c-t)} \\ & + (r-c) \sqrt{2rc-c^2} \end{aligned} \quad (5.6)$$

The reference flow cross-sectional area and heat transfer surface are depicted in the drawings of Figure 5.2.

5.3. Numerical methods

5.3.1. Governing equations

Offset-strip fins are modelled by means of basic unit geometries, which are obtained by exploiting the periodicity of the geometry and flow along the streamwise and spanwise directions. The flow is assumed incompressible, and periodic between the inlet and outlet sections of the examined periodic modules. The fluid velocity, pressure, and temperature fields are obtained from the numerical solution of the steady-state momentum, continuity,

and energy equations, which in dimensionless form read:

$$\begin{cases} u_j^* \frac{\partial u_i^*}{\partial x_j^*} = -\frac{\partial p^*}{\partial x_i^*} + \frac{\partial}{\partial x_j^*} \left[\left(\frac{1}{\text{Re}} + \nu_t^* \right) \frac{\partial u_i^*}{\partial x_j^*} \right] \\ \frac{\partial u_i^*}{\partial x_i^*} = 0 \\ u_j^* \frac{\partial T^*}{\partial x_i^*} = \frac{\partial}{\partial x_j^*} \left[\left(\frac{1}{\text{Re Pr}} + \frac{\nu_t^*}{\text{Pr}_t} \right) \frac{\partial T^*}{\partial x_j^*} \right] \end{cases} \quad (5.7)$$

The temperature field within the fin walls is derived from the numerical solution of the steady state heat equation.

$$\frac{\partial}{\partial x_j^*} \left(\lambda_s^* \frac{\partial T^*}{\partial x_j^*} \right) = 0 \quad (5.8)$$

The fin height y is assumed as the reference length of the problem, while the velocity scale, defined as the integral mean velocity over the inlet surface, is taken as unitary. The Reynolds number is defined as follows:

$$\text{Re} = \frac{\bar{u}_i y}{\nu}. \quad (5.9)$$

The dimensionless thermal conductivity of the fin walls is derived by scaling the thermal conductivity of aluminium by the one of the neighbouring fluid:

$$\lambda_s^* = \frac{\lambda_{\text{Al}}}{\lambda_f}, \quad (5.10)$$

the values of λ of aluminium and all the considered fluids are reported in Table 5.1.

At $\text{Re} \geq 1000$, the k - ω SST turbulence model [195], coupled with a low-Reynolds wall modelling approach, is employed to compute eddy viscosity ν_t^* in Equations (2), as it is the turbulence model that gives the best agreement between numerical and experimental results according to *et al.* [213]. On the other hand, for all flows at $\text{Re} \leq 200$ laminar analyses are carried out, therefore, in this case eddy viscosity is set to zero. At $\text{Re} = 500$ laminar simulations are performed as long as a satisfactory degree of numerical convergence is achieved, i.e. when all residuals are of the order of 10^{-6} . Otherwise, the turbulence model is employed. In the turbulent analyses, a unitary value for the turbulent Prandtl number Pr_t is considered, on the basis of the Reynolds analogy.

Table 5.1. Reference values of thermal conductivity of aluminium and oil, water, and air flows.

	Aluminium	Oil	Water	Air
$\lambda \text{ [W m}^{-1} \text{ K}^{-1}]$	237.0	0.129	0.609	0.024

5.3.2. Boundary conditions

Fully developed flow conditions are attained by applying suitable field mapping boundary conditions for the velocity and temperature fields, between the inlet and outlet sections. At each iteration of the solution algorithm, the values of dimensionless velocity and temperature at each cell centre on the outlet patch are applied to the respective cell centre on the inlet patch, after being scaled to maintain the desired integral mean values at the inlet: $\bar{u}_i^* = 1$ and $\bar{T}_i^* = 0$. At the outlet, the surface mean pressure is set to a zero reference value $\bar{p}_o^* = 0$, while at the inlet a zero gradient condition is applied to pressure. Periodic conditions are enforced along the spanwise direction. At the top and bottom boundaries a uniform entering heat flux is imposed: $q''^* = 1$. At the walls, no-slip conditions are applied to the velocity field and a zero pressure gradient is imposed. Finally, continuity of temperature and heat flux is enforced at all fluid-solid interfaces. Table 5.2 summarises the boundary conditions described herein.

5.3.3. Spatial discretisation

A Finite Volume conjugate heat transfer solver, previously validated in [269], and implemented in the open source software OpenFOAM 7 [196], is employed for the numerical analyses. The previously discussed dimensionless framework is implemented by setting up the thermophysicalProperties file as shown in Appendix B. Spatial discretisation of convective terms is addressed with second-order upwind schemes, while diffusive terms are handled with central differencing schemes with explicit non-orthogonality correction. The momentum and continuity equations are decoupled by means of the SIMPLE method [233]. The computational domains are generated using an automated mesh generation script implemented in Glyph, the Tcl-based scripting language for the commercial meshing tool Pointwise. Meshing strategies and parameters are chosen with aim of obtaining grids with satisfactory quality indicators, while maintaining a relatively simple implementation of the automated scripts and limiting the number of cells of each computational mesh. On average, the resulting computational grids feature mean values of equiangle skewness of 0.045, mean mesh non-orthogonality of 5.75 degrees, and a number of cells of roughly 2 millions. Figure 5.3 shows one of the generated computational grids, where the fluid and solid regions are clearly highlighted.

5.3.4. Mesh sensitivity analysis

The dependence of numerical results on the computational grid is assessed by comparing local velocity and temperature results, and integral results on head losses and heat transfer, which are represented by an equivalent Darcy friction factor and Nusselt number (for their definitions refer to Section 5.5), obtained from three different grids featuring the same topology. The analysis is conducted considering an oil flow at $Re = 50$ and an air flow at $Re = 12000$, the latter representing the worst case scenario in terms of the computation of

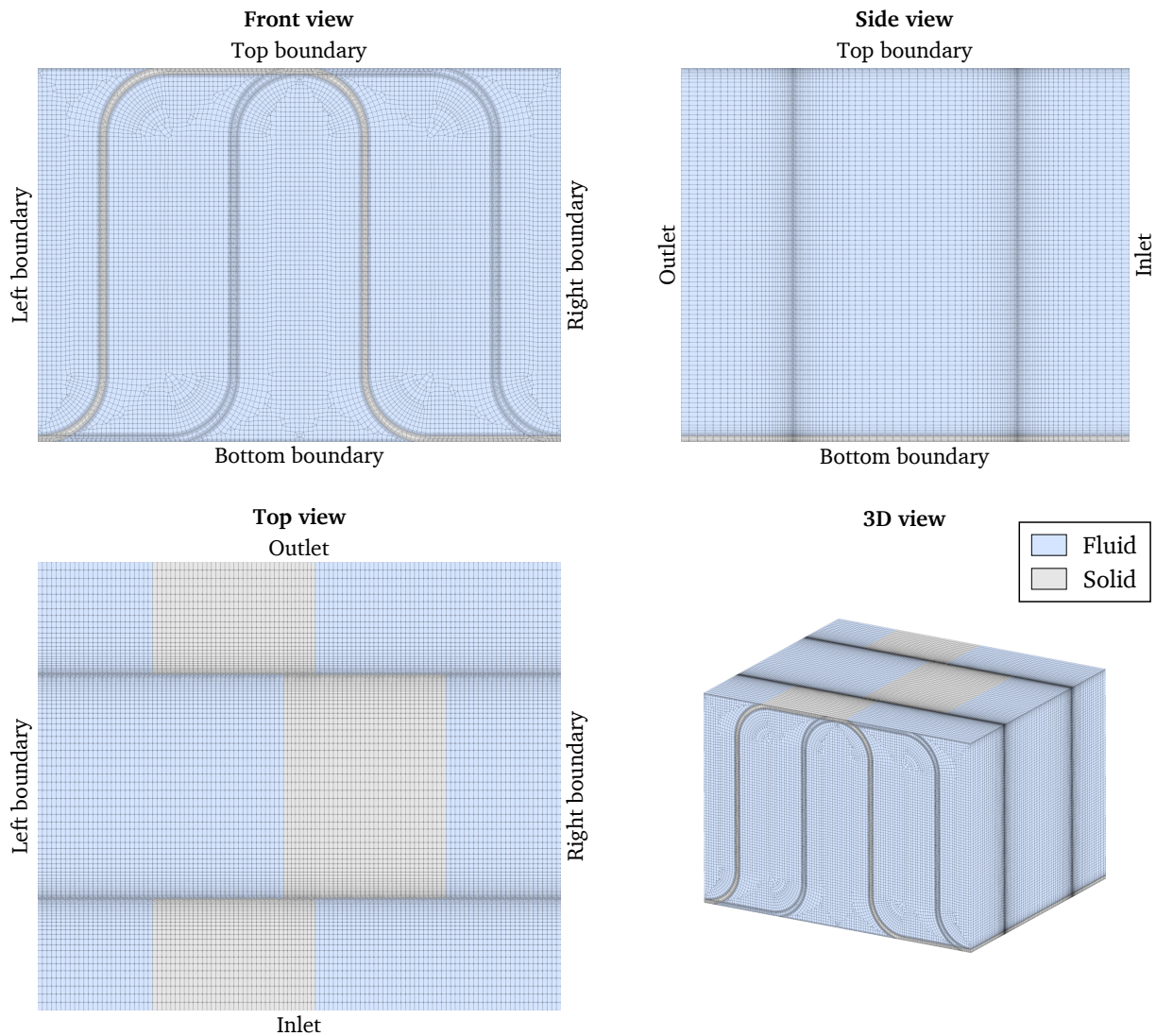


Figure 5.3. Selected views of one of the generated computational grids, related to the geometrical configuration defined by $x^* = 0.7$ and $l^* = 0.6$.

Table 5.2. Boundary conditions applied to the inlet, outlet, left and right boundaries, and top and bottom surfaces in fluid and solid regions.

Boundaries		inlet	outlet	left/right top/bottom		fluid-solid interface
fluid	$\vec{u}^*$	mapped	zero-gradient	periodic	no-slip	no-slip
	p^*	zero-gradient	fixed mean	periodic	zero-gradient	zero-gradient
	T^*	mapped	zero-gradient	periodic	fixed gradient	heat flux continuity
solid	T^*	zero-gradient	zero-gradient	periodic	fixed gradient	heat flux continuity

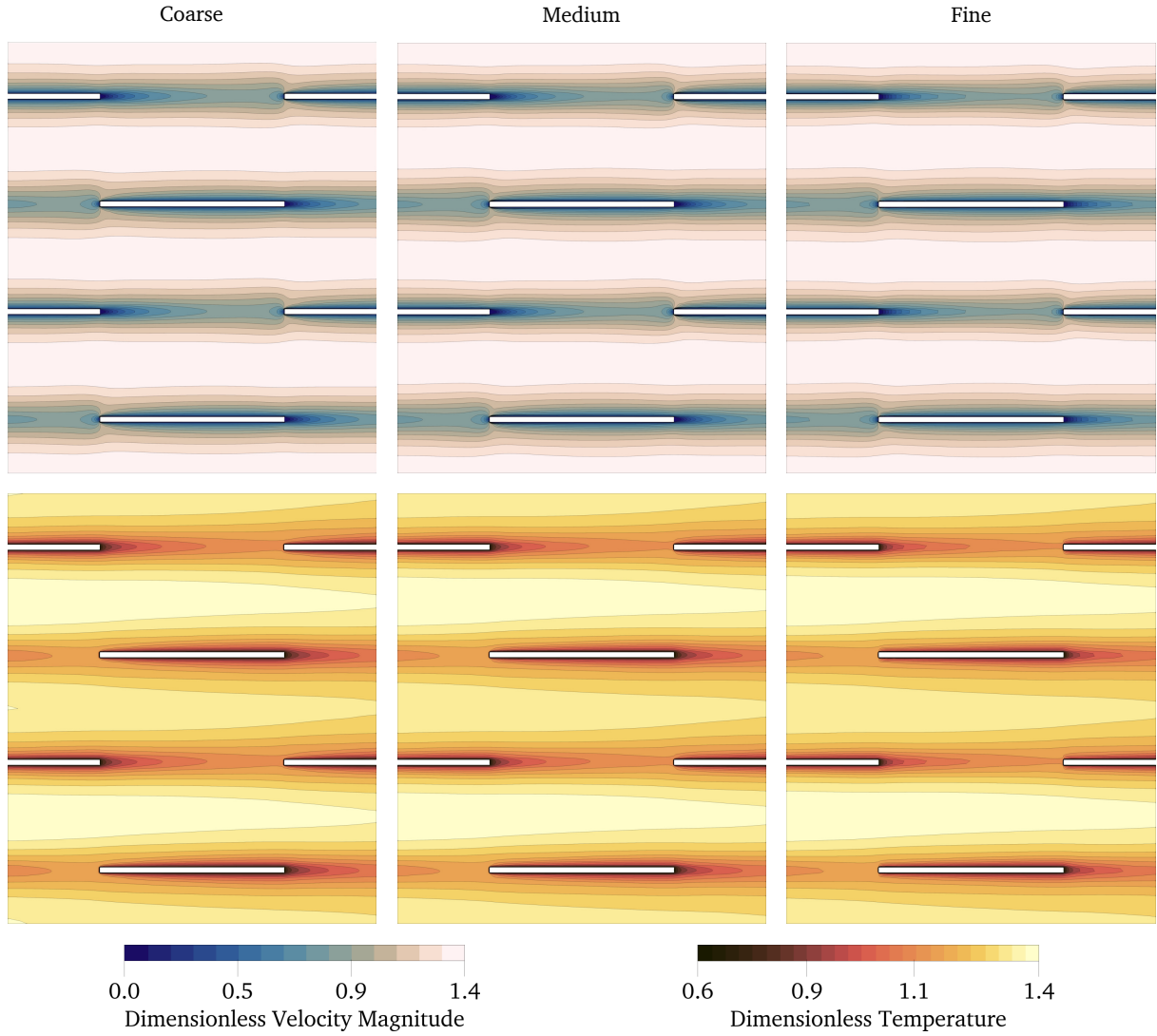


Figure 5.4. Contours of dimensionless velocity magnitude and temperature difference obtained from the coarse, medium, and fine meshes at $Re = 12000$ and $Pr = 0.71$. Colour schemes from [270].

wall normal gradients. Table 5.3 shows f and Nu values, along with the mean dimensionless wall distance, that are obtained from three computational meshes of an offset-strip fin with $x^* = 0.7$ and $l^* = 0.6$. The deviations in friction factor and Nusselt number related to each grid are computed by taking the value from the previous mesh as a reference. In laminar flow, at $Re = 50$, the three meshes produce very close results, showing deviations well below 0.2% when comparing integral results between the medium and fine grids. At $Re = 12000$ these deviations increase up to 7.2% for f and 3.3% for Nu . Figure 5.4 compares local results on dimensionless velocity magnitude and temperature difference (see Section 5.5) obtained from the three grids at $Re = 12000$, by means of contour plots on cross-sections positioned at half of the fin height. The three velocity contours feature similar distributions, with the most noticeable differences found in the wakes between two successive fin walls, thus affecting the prediction of pressure losses as seen in Table 5.3. On the other hand, in

the temperature contours the most noticeable differences are spotted in the sub-channels generated by the staggered fin profiles.

Although the deviations observed between results from the medium and fine grids are not insignificant, the meshing parameters associated with the medium grid are selected for the development of all computational grids for the following CFD analyses. This choice is aimed at achieving an acceptable trade-off between accuracy and the required computational effort, which is particularly relevant in this case since the total number of CFD analyses amounts to 704 (see Section 5.4). On average, the medium-sized grid requires roughly 27 core-hours for the solution of a single flow rate, while the fine grid requires a much higher 865 core-hours, representing an increase in computational time which is arguably not justified by the accuracy improvements shown in Table 5.3. Furthermore, it should be stressed that the deviations observed at $Re = 12000$ really represent the most penalising scenario, while at lower Reynolds values much lower differences between integral results are expected, as confirmed by the numerical analyses carried out at $Re = 50$.

5.4. Design of Experiment

The dependence of the thermal-hydraulic performance of OSFs on geometric and flow parameters is evaluated by means of a Design of Experiment procedure. The dimensionless fin pitch and offset length are assumed as the most influential geometric parameters [254], instead the fillet radius and wall thickness are considered of secondary importance. The levels of the dimensional parameters x^* , l^* and of the flow parameters, i.e. pairs of Re and Pr values, are chosen by considering a set of configurations which are representative of real industrial applications, including coolers for hydraulic and lubricating oils, water radiators, and air-to-air heat exchangers [238]. Thus, eight levels are selected for x^* and l^* , and eleven different flows are taken into account. The fillet radius is varied indirectly by computing it as a function of fin pitch and wall thickness, according to Equation (5.1). This

Table 5.3. Friction factor, Nusselt number, and mean dimensionless wall distance values obtained from the coarse, medium, and fine computational meshes of a fin with $x^* = 0.7$ and $l^* = 0.6$.

Re	Grid	Δ^*	$\bar{y}^+$	f	Nu	Δf	ΔNu
50	Coarse	0.014	-	3.43	14.9	-	-
	Base	0.007	-	3.44	15.0	0.4%	0.24%
	Fine	0.0035	-	3.44	15.0	-0.02%	0.16%
12000	Coarse	0.014	1.44	0.16	71.1	-	-
	Base	0.007	0.64	0.14	67.1	-9.2%	-5.6%
	Fine	0.0035	0.06	0.13	64.8	-7.2%	-3.3%

strategy allows for the removal of one geometric parameter from the DOE, which vastly reduces the number of numerical analyses to perform. On the other hand, the effects of pitch and radius variations on thermal-hydraulic performance may be confounded, as the fillet radius is aliased by Equation (5.1). In this case, this is deemed acceptable as the fillet radius is most likely to have a lower order influence on head losses and heat transfer with respect to the fin pitch [254]. The dimensionless wall thickness t^* is kept constant at a value of 0.017, again in an effort to reduce the number of simulations, and because in real applications the choice of the wall thickness values is limited, due to the commercially available aluminium sheets and the need to satisfy criteria of structural integrity and mechanical resistance of the heat exchangers. In an effort to obtain an accurate sampling of the parameter space a full factorial DOE with respect to the factors x^* and l^* and the (Re, Pr) couples is performed. This is equivalent to performing a fractional factorial DOE with respect to x^* , l^* , r^* , Re, and Pr. Table 5.4 reports the levels chosen for each geometric and flow factor, while the number of numerical analyses performed amounts to 704.

The configurations studied in the present work are compared to those analysed in some of the most relevant works available in the literature in the plots of Figure 5.5. The geometric configurations studied in the literature are spread very unevenly over the geometric parameter space, and some configurations are repeated among the different works. By examining the distribution of points in Figure 5.5a it is clear that the correlations proposed in the literature cannot be considered universal. Furthermore, given the irregularity of the distribution, it is also difficult to guess their actual field of applicability and to discern in which parts of the geometrical parameter space they represent an extrapolation. A clear advantage of the DOE approach presented in this work is the clear identification of the ranges of validity of the obtained response surfaces (Section 5.6), which are represented by the shaded areas of the plots of Figure 5.5. Nevertheless, such an advantage comes at the cost of a large computational effort. Moreover, an analysis of this scope would be practically unfeasible if conducted by means of experimental techniques, which were used the most in the papers available in the literature. The graph of Figure 5.5b clearly shows that most of the works available in the literature deal with air flows ($Pr = 0.71$) only, therefore the resulting correlations are to be considered reliable only for gas flows, as usually specified by the authors. The work presented by Kim *et al.* [213] is one of the few that considers higher Prandtl number fluids (up to 50), in fact the associated correlations are those with the widest field of applicability.

In closing, the present analysis should be seen as integrative with respect to what has been done so far by other researchers, as the main objective is to cover the space of dimensional and flow parameters in a more uniform and systematic way, in order to obtain more general and reliable response surfaces. Nevertheless, this work is by no means definitive, given the large number and variability of the dimensional and flow parameters associated with the use of offset-strip fins in compact heat exchangers.

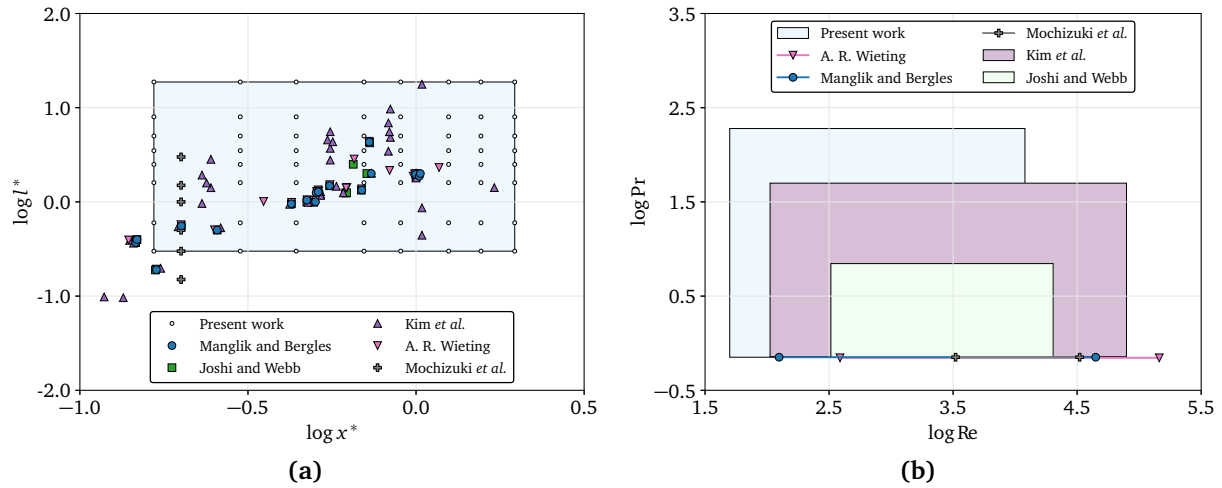


Figure 5.5. Comparison between the dimensional (a) and flow (b) parameters considered in this work with those analysed in previous works in the literature.

5.5. Numerical results

Numerical flow field results are hereby presented by means of dimensionless velocity and temperature contour plots, associated with cross-sections positioned at the fins half height. The dimensionless temperature difference is computed as in definition [58]:

$$\vartheta = \frac{\overline{T}_w - T}{\overline{T}_w - \overline{T}_b}, \quad (5.11)$$

where $\overline{T}_w$ is the integral mean temperature over the heat transfer area A_{ref} , while $\overline{T}_b$ is computed as the mean of the mass flow weighted averages of temperature on the inlet and outlet surfaces. Figures 5.6 and 5.7 depict an example of the aforementioned contour plots related to an offset-strip fin with $x^* = 0.7$ and $l^* = 0.6$. Furthermore, two reference flows are considered: a laminar oil flow ($\text{Re} = 50$, $\text{Pr} = 190$) and a turbulent air flow ($\text{Re} = 12000$, $\text{Pr} = 0.71$). As expected, in laminar regime the development of boundary layers along the fin walls is much more evident with respect to the case of turbulent flow, and coherently, greater velocity values are noted near the leading edge of the fins. The presence of the fillet radii determines an asymmetry between the two flow passages formed at each interruption of the profiles, resulting in slight asymmetries in the velocity and temperature fields. It is

Table 5.4. Selected levels for the geometric and flow factors.

x^*	0.166	0.3	0.44	0.7	0.9	1.25	1.56	1.966	
l^*	0.3	0.6	1.6	2.5	3.5	5.0	8.0	18.75	
Re	50	100	200	500	1000	2000	3000	5000	12000
Pr	190	190	190	190	7	7–0.71	7–0.71	0.71	0.71

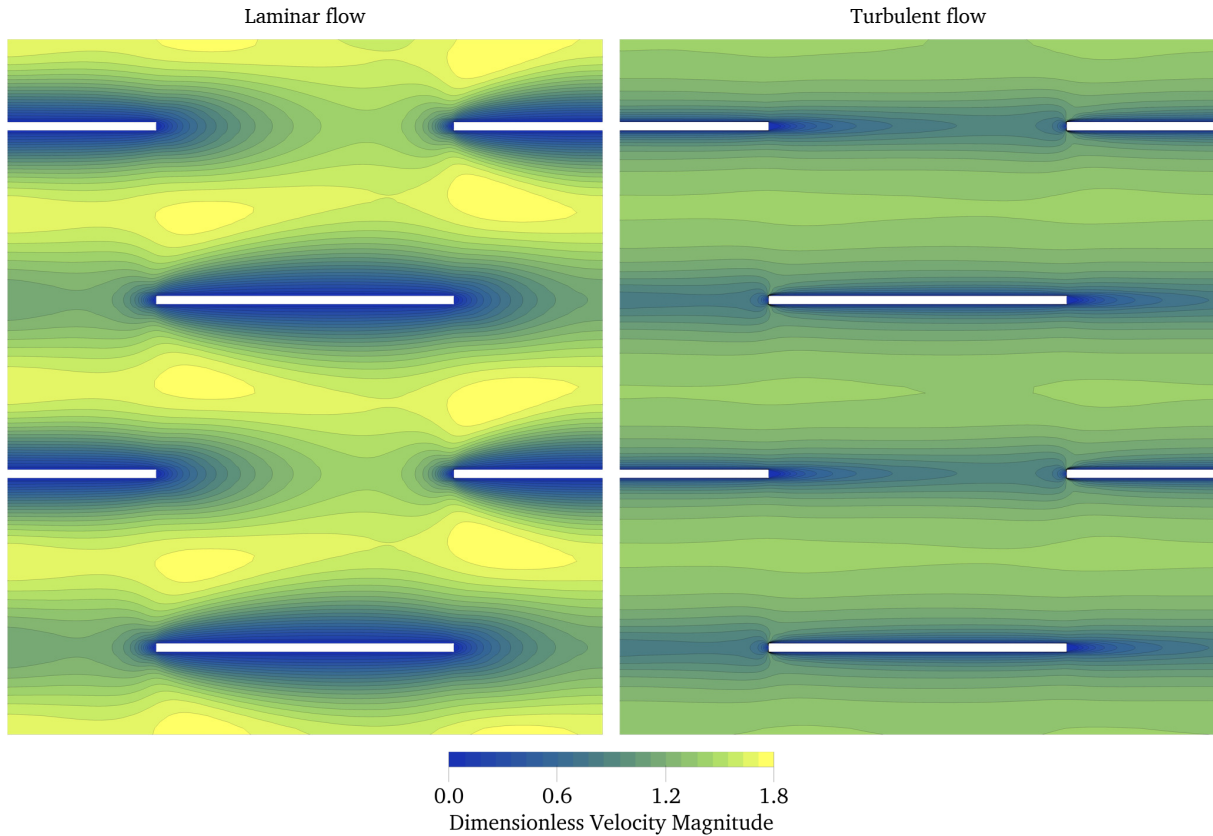


Figure 5.6. Contour plots of dimensionless velocity magnitude on height centred cross-sections, related to a laminar (left) and turbulent flow (right). Colour scheme from [270].

interesting to note that even though in laminar regime the velocity distribution features very mild asymmetries, the temperature field presents a really marked asymmetric distribution, suggesting that this behaviour is emphasised by large values of Prandtl number. In the turbulent regime, the wakes formed past each fin wall are much smaller along the spanwise direction, than those generated in laminar flow. However, these wakes are never completely dissipated due to the short offset length. In this case, the flow asymmetries are more evident in the velocity field than in the temperature one.

Figure 5.8 shows contour plots of Turbulent Kinetic Energy (TKE) on a longitudinal cross-section and in the fluid volume near the fin walls, along with the distributions of local Nusselt number and friction coefficient. The 3D iso-surfaces of TKE in Figure 5.8 are referred to TKE-values of 0.033, 0.035, and 0.046. The maximum values of TKE are noticed near the leading edge of the walls and in the wakes. Furthermore, it can be observed that regions exhibiting high TKE values, indicative of high turbulent dissipation, correspond to areas where higher values of the local friction coefficient can be observed on the fin walls. As expected, the largest values of local Nusselt number are observed near the leading edge of the walls, while far lower values are noted on the side walls. The regions of high local Nu are broadly consistent with zones of large turbulent kinetic energy.

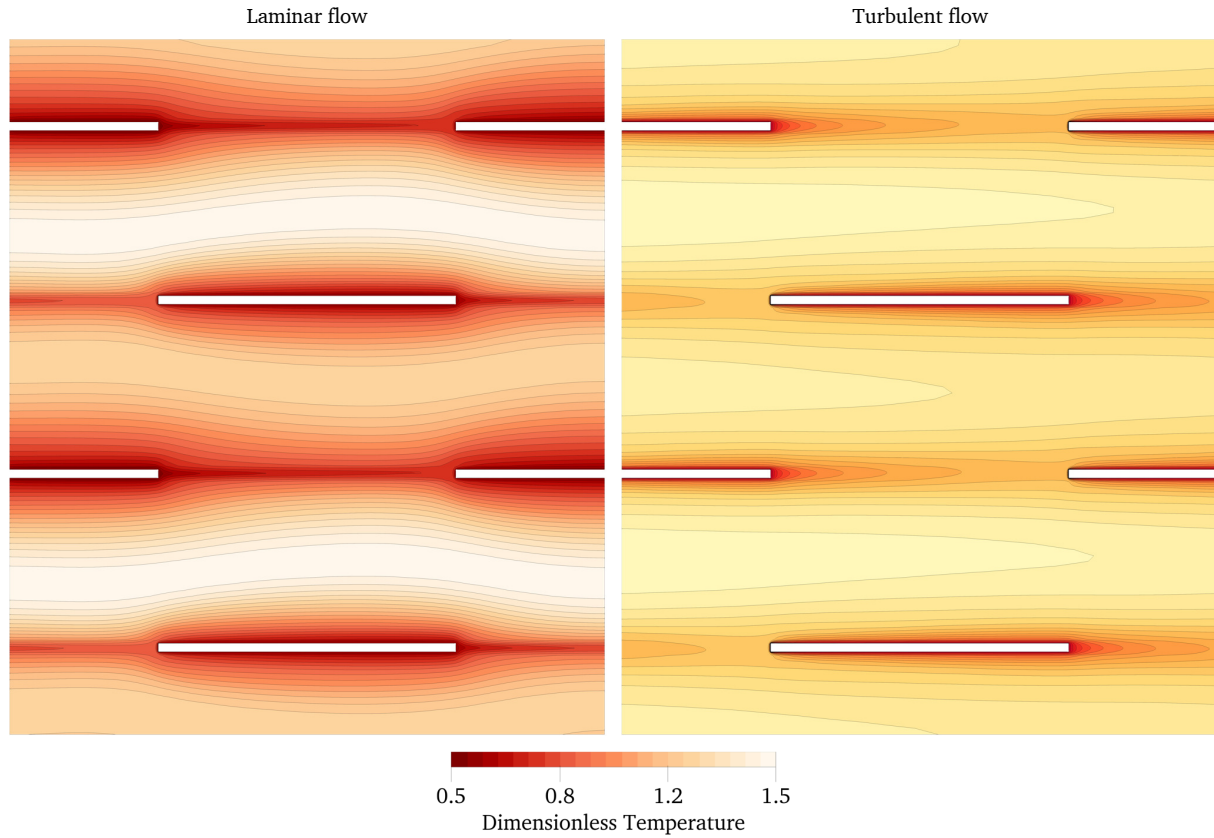


Figure 5.7. Contour plots of dimensionless temperature difference on height centred cross-sections, related to a laminar (left) and turbulent flow (right).

Integral head losses and heat transfer rates are evaluated by means of an equivalent Darcy friction factor and mean Nusselt number. The equivalent friction factor f is derived from the Darcy-Weisbach formula, where the reference length is the fin height:

$$f = \frac{y(\bar{p}_i - \bar{p}_o)}{l \rho \bar{u}_i^2}, \quad (5.12)$$

overbars denote integral mean values over inlet (i) and outlet (o) sections. The mean Nusselt number over the heat transfer area is:

$$\text{Nu} = \frac{\dot{Q} y}{\lambda A_{\text{ref}} (\bar{T}_w - \bar{T}_b)}, \quad (5.13)$$

where $\dot{Q}$ stands for the heat entering the system through the top and bottom surfaces. Figure 5.9 shows contour plots depicting the distributions of f and Nu over the examined configuration space, related to a laminar flow at $\text{Re} = 50$. As expected, the equivalent friction factor decreases with increasing fin pitch and offset length. Moreover, at small pitch values and for large l^* -values the contour lines become almost vertical, underlining the reduced dependence of f on the offset length in this region of the parameter space. Conversely, this effect is not as pronounced for large pitch values, since the friction factor

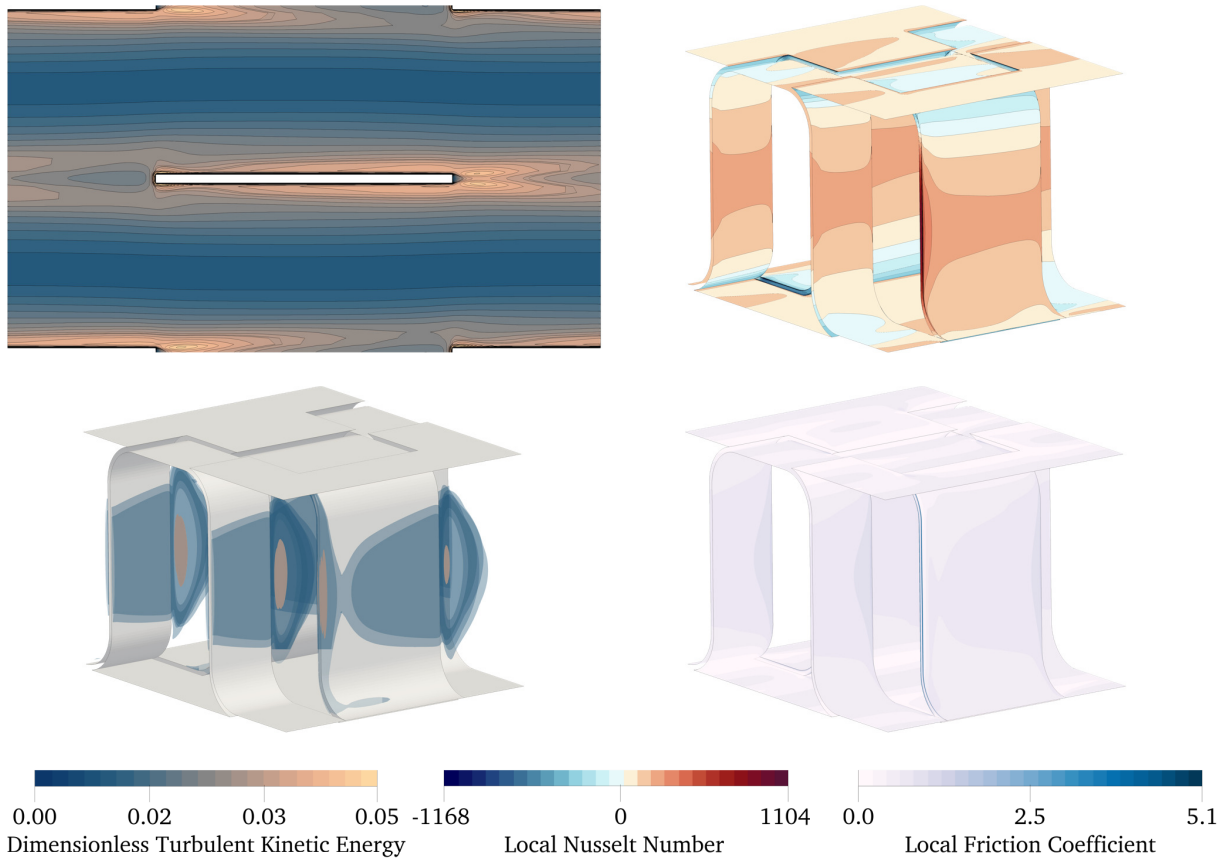


Figure 5.8. Distributions of turbulent kinetic energy, local Nusselt number, and local friction coefficient, obtained at $Re = 12000$ and $Pr = 0.71$.

maintains a more marked sensitivity with respect to the offset length. In laminar regime, the mean Nusselt number shows comparable trends to those of the friction factor, but in this case a much more pronounced dependence on the offset length is noted, especially towards the lower end of the considered fin pitch range. Figure 5.10 shows analogous contour plots, related to turbulent a flow at $Re = 12000$. The friction factor distribution closely resembles its laminar flow counterpart, while the mean Nusselt number features a local minimum for the values $x^* = 0.166$ and $l^* = 1.6$. This behaviour is investigated further by comparing the local Nusselt number distributions of three fins with a value of pitch of 0.166 and l^* -values of 0.6, 1.6, and 2.5. The black rectangles in Figure 5.11 highlight the subtle differences between the obtained Nu distributions in the central region of the upper surface of the fin profiles. The configuration with $l^* = 1.6$ exhibits a notable absence of the high Nu strip that is clearly spotted in the other two configurations. Moreover, as emphasised by the magenta rectangles, the fins with l^* -values of 1.6 and 2.5 feature regions of low Nusselt number in proximity to the fillet radius on the lower portion of the side walls. The fin with $l^* = 0.6$ features the largest of these regions and also lacks the region where large values of Nusselt number are spotted in the configuration with $l^* = 2.5$. The observed differences between the three Nu distributions are consistent with the Nusselt number minimum present in Figure 5.10b. Moreover, the latter appears to be related to the presence of the fillet radius

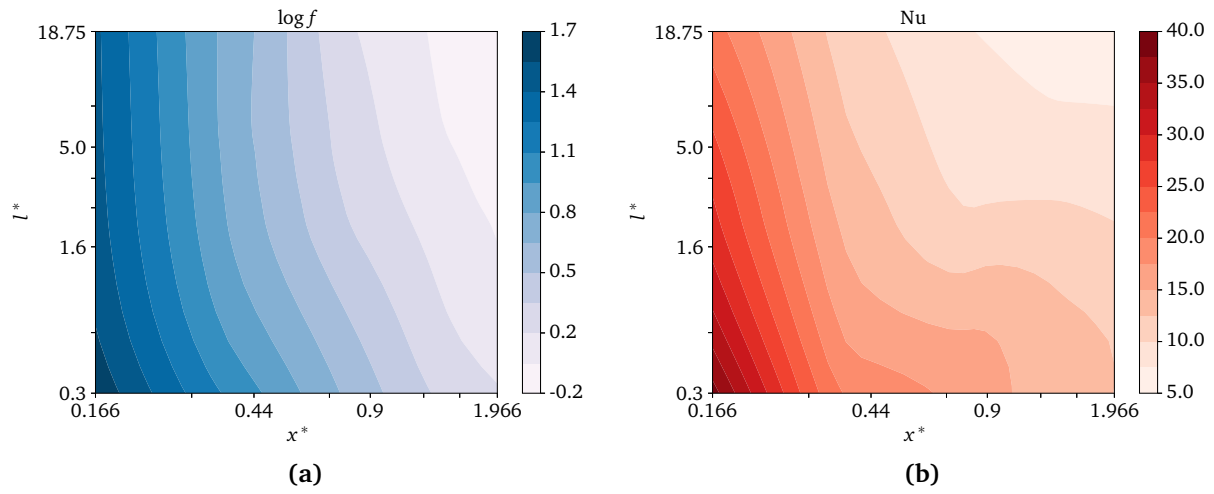


Figure 5.9. Contour plots of equivalent Darcy friction factor (a), and mean Nusselt number (b), obtained from the CFD analyses performed at $Re = 50$ and $Pr = 190$. In these plots the x^* and l^* axes are in logarithmic scale. However, the axis labels are in linear scale for better readability.

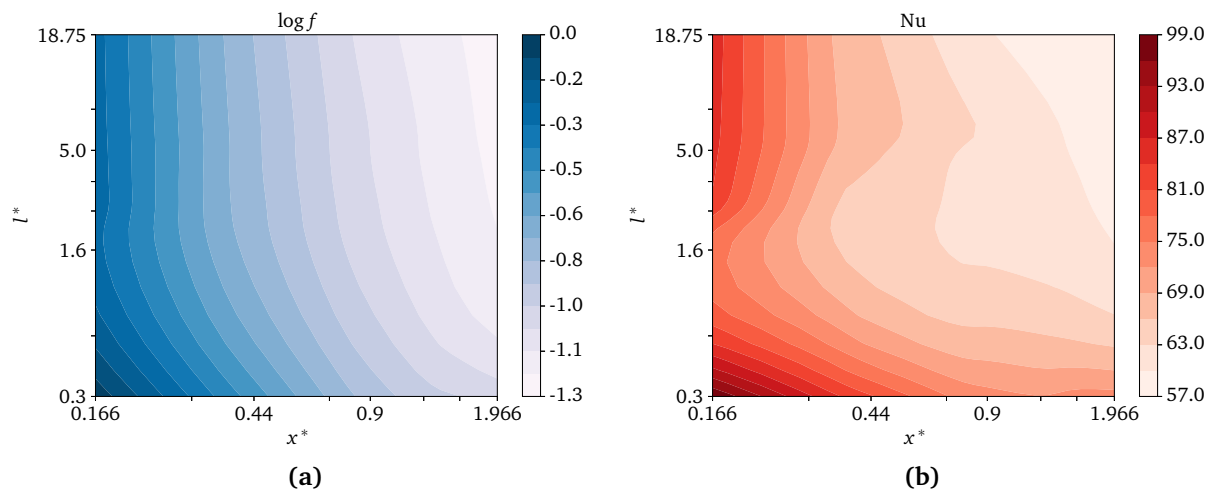


Figure 5.10. Contour plots of equivalent Darcy friction factor (a), and mean Nusselt number (b), obtained from the CFD analyses performed at $Re = 12000$ and $Pr = 0.71$. In these plots the x^* and l^* axes are in logarithmic scale. However, the axis labels are in linear scale for better readability.

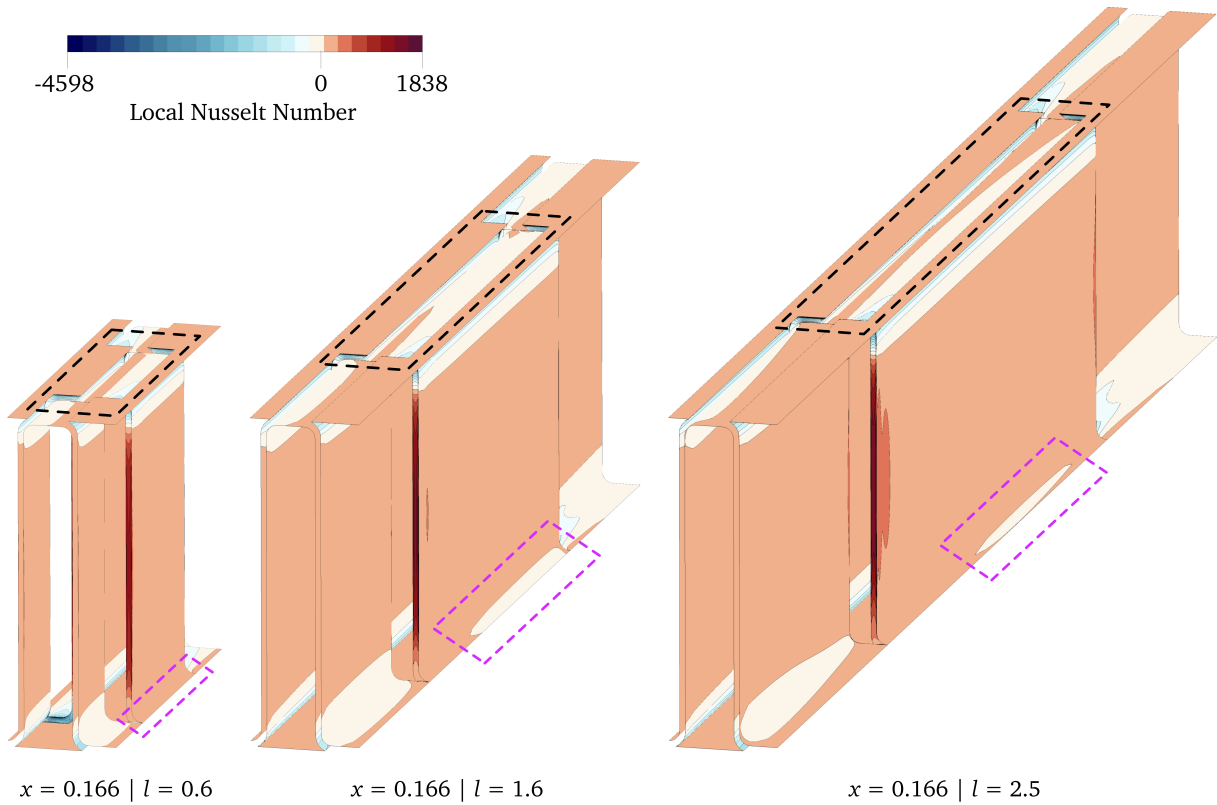


Figure 5.11. Distributions of local Nusselt number related to offset-strip fins with pitch $x^* = 0.166$ and values of offset length of 0.6 (left), 1.6 (middle), and 2.5 (right), obtained at $Re = 12000$ and $Pr = 0.71$.

between the vertical and horizontal walls of the fins, since most of the deviations between the distributions in Figure 5.11 are observed in correspondence of the radiused corners.

5.6. Derivation of response surfaces

The derivation of the response surfaces presents several challenges: first and foremost, the number of data points is quite large, and accurately representing them without the use of overly complex models is a challenging task. Furthermore, the selected levels for the geometric and flow factors are unevenly distributed, and they cover extremely wide ranges spanning several orders of magnitude, especially in the case of the Reynolds and Prandtl numbers.

An additional challenge arises from the laminar-turbulent transition, that occurs at different Reynolds numbers depending on the geometric parameters: in fins with the minimum pitch value ($x^* = 0.166$) the transition Reynolds number is between 500 and 1000, while for all the other configurations it is located roughly between 250 and 500. The offset length has a comparatively minor influence on the transition point with respect to the pitch, as it affects the transition of only a few fin configurations that feature a pitch of $x^* = 0.44$. In the frame of the derivation of response surfaces and correlations, the laminar-turbulent transition is usually addressed by deriving response surfaces for each flow regime and prescribing the

appropriate ranges of validity. Although this is a very straightforward and robust approach, it leads to piecewise-defined response surfaces, which may not be suitable for optimisation procedures as they are not continuous functions.

The response surfaces presented in the following are obtained by defining objective functions and fitting them to data points by means of a variant of the Levenberg-Marquardt algorithm [235], that is implemented in the SciPy library [237]. Although response surfaces obtained by least-squares regression may not be as accurate in approximating the base data as those obtained by ANN-based approaches, they are used in this analysis because of their advantages in terms of ease of implementation and application. All data points in the range $50 \leq \text{Re} \leq 200$ are employed for the fit of the laminar functions while the ones in the range $1000 \leq \text{Re} \leq 12000$ are used for the turbulent fit. The data points obtained at $\text{Re} = 500$ are used to fit the laminar or turbulent response surfaces depending on the specific geometric configuration. The transition is managed by combining the response surfaces obtained for laminar and turbulent regimes by means of a smoothing function, based on the hyperbolic tangent, that is used to connect the two surfaces around the suited transition point.

5.6.1. Equivalent Darcy friction factor

The dependence of the equivalent Darcy friction factor on the Reynolds number is modelled by polynomial functions in log-log scale:

$$\log f_l = b_1 \log \text{Re} + b_2, \quad (5.14)$$

$$\log f_t = b_3 \log^{-0.48} \text{Re} + b_4. \quad (5.15)$$

These formulations are used to correlate results on both pressure losses and heat transfer (see Section 5.6.2) as they provide very good accuracy levels when used for extended surfaces [238]. The values of the b_m coefficients found in Equations (5.14) and (5.15) are obtained from the following relation, as a function of the geometrical parameters:

$$b_m = k_1(x^*)^{k_2} + k_3 x^* + k_4 \log l^* + k_5 l^* + k_6. \quad (5.16)$$

Equation (5.16) is formulated by progressively adding polynomial and logarithmic based terms as long as these provide significant improvements to the accuracy of the response surface. The values of the k_n coefficients are determined through the aforementioned least-squares-based curve fitting procedure, and their values are listed in Table 5.5. In order to suitably combine Equations (5.14) and (5.15) a smoothing function based on the hyperbolic tangent is formulated:

$$s = \frac{1}{2} \left[1 + \tanh \left(\frac{\chi - \chi_0}{\sigma} \right) \right]. \quad (5.17)$$

In Equation (5.17) χ stands for the generic independent variable, χ_0 locates the smooth step on the independent variable axis, while σ determines the sharpness of the transition.

The laminar and turbulent response surfaces (Equations (5.14) and (5.15)) are combined into a single equation by using a variant of Equation (5.17) centred at the transition Reynolds number Re_{trans} :

$$\log f = s_{\text{Re}} \log f_t + (1 - s_{\text{Re}}) \log f_l, \quad (5.18)$$

$$s_{\text{Re}} = \frac{1}{2} \left[1 + \tanh \left(\frac{\log Re - \log Re_{\text{trans}}}{0.08} \right) \right]. \quad (5.19)$$

The transition Reynolds number is in turn determined by applying the previously defined smoothing function (Equation (5.17)) between the Re -values of 750 and 350 which have been designated as transition values, on the basis of the fin dimensionless pitch alone.

$$Re_{\text{trans}} = s_{x^*} 350 + (1 - s_{x^*}) 750, \quad (5.20)$$

$$s_{x^*} = \frac{1}{2} \left[1 + \tanh \left(\frac{x^* - 0.166}{1.1} \right) \right]. \quad (5.21)$$

The values of transition Reynolds number in Equation (5.20), along with the values of σ in Equations (5.19) and (5.21) were selected with the aim of maximising the accuracy of the overall response surface with respect to the data points. The plots of Figure 5.12 display CFD results for the equivalent Darcy friction factor, the obtained response surface expressed by Equation (5.18), and a comparison with some of the most relevant correlations available in the literature. The displayed configurations cover the limits of the geometric parameter ranges and their median values. First of all, near the transition, a notable discontinuity between the laminar and the turbulent friction factor distributions is noted, as suggested by many authors [204, 205, 208], but in contrast with the results proposed by Manglik and Bergles [206], who argue that a smooth transition from laminar to turbulent of the friction factor should be observed. This is what prompted the use of the previously discussed smoothing technique, that appears adequate to correctly represent the transition for the examined configurations, as the mean deviation between the response surface and the data points is of 8.45%, as seen in Figure 5.15a.

Table 5.5. Values of k_n coefficients in Equation (5.16).

	k_1	k_2	k_3	k_4	k_5	k_6
b_1	-345.50	1.00033743	345.5969	0.0651851	-0.00681313	-0.9987
b_2	4421.93	1.00032627	-4423.93	-0.367102	0.020911319	3.76899
b_3	-4091.5	0.00026439	0.820411	-0.282540	0.017530123	4095.36
b_4	1356.77	1.00001750	-1356.95	-0.049099	-0.00012869	-3.1953

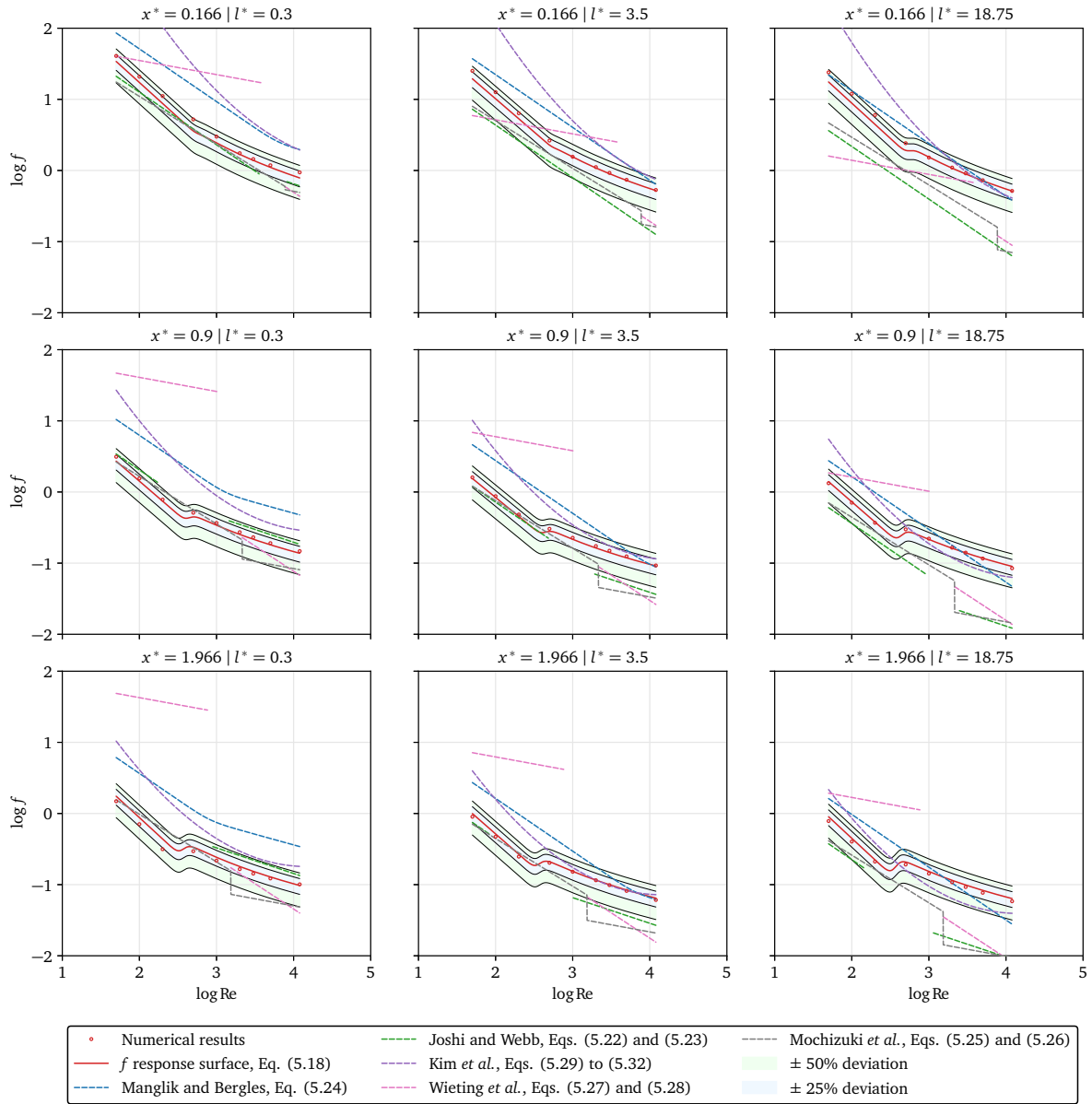


Figure 5.12. Equivalent Darcy friction factor: numerical results, developed response surface, and a comparison with the most relevant correlations from the literature for nine selected geometric configurations.

The numerical results and response surface obtained in this work are compared with some of the most relevant correlations available in the literature including the ones from:

- Joshi and Webb [208]

laminar flow: $\text{Re}_{D_h} \leq \text{Re}_{D_h}^*$

$$f_F = 8.12 \text{Re}_{D_h}^{-0.74} \left(\frac{l}{D_h} \right)^{-0.41} \alpha^{-0.02} \quad (5.22)$$

turbulent flow: $\text{Re}_{D_h} \geq \text{Re}_{D_h}^* + 1000$

$$f_F = 1.12 \text{Re}_{D_h}^{-0.36} \left(\frac{l}{D_h} \right)^{-0.65} \left(\frac{t}{D_h} \right)^{0.17} \quad (5.23)$$

- Manglik and Bergles [206]

$$f_F = 9.6243 \text{Re}_{D_h}^{-0.7422} \alpha^{-0.1856} \delta^{0.3053} \gamma^{-0.2659} \times \left[1 + 7.669 \cdot 10^{-8} \text{Re}_{D_h}^{4.429} \alpha^{0.920} \delta^{3.767} \gamma^{0.236} \right]^{0.1}, \quad (5.24)$$

- Mochizuki *et al.* [205]

laminar flow: $\text{Re}_{D_h} < 2000$

$$f_F = 5.55 \left(\frac{l}{D_h} \right)^{-0.32} \alpha^{-0.092} \text{Re}_{D_h}^{-0.67} \quad (5.25)$$

turbulent flow: $\text{Re}_{D_h} \geq 2000$

$$f_F = 0.83 \left(\frac{l}{D_h} + 0.33 \right)^{-0.5} \left(\frac{t}{D_h} \right)^{0.534} \text{Re}_{D_h}^{-0.20} \quad (5.26)$$

- Wieting [204]

laminar flow: $\text{Re}_{D_h} \leq 1000$

$$f_F = 7.661 \left(\frac{l}{D_h} \right)^{-0.384} \alpha^{-0.092} \text{Re}_{D_h}^{-0.712} \quad (5.27)$$

turbulent flow: $\text{Re}_{D_h} \geq 2000$

$$f_F = 1.136 \left(\frac{l}{D_h} \right)^{-0.781} \left(\frac{t}{D_h} \right)^{-0.534} \text{Re}_{D_h}^{-0.198} \quad (5.28)$$

- Kim *et al.* [213]

for $\beta_r < 20\%$

$$f_F = e^{7.91} \alpha^{-0.159} \delta^{0.358} \gamma^{-0.033} \text{Re}_{D_h}^{0.126 \ln \text{Re}_{D_h} - 2.3} \quad (5.29)$$

for $20\% \leq \beta_r < 25$

$$f_F = e^{9.36} \alpha^{-0.0025} \delta^{-0.0373} \gamma^{1.85} \text{Re}_{D_h}^{0.142 \ln \text{Re}_{D_h} - 0.623} \quad (5.30)$$

for $25\% \leq \beta_r < 30\%$

$$f_F = e^{5.58} \alpha^{-0.36} \delta^{0.552} \gamma^{-0.521} \text{Re}_{D_h}^{0.111 \ln \text{Re}_{D_h} - 1.87} \quad (5.31)$$

for $30\% \leq \beta_r \leq 35\%$

$$f_F = e^{4.84} \alpha^{-0.48} \delta^{0.347} \gamma^{0.511} \text{Re}_{D_h}^{0.0891 \ln \text{Re}_{D_h} - 1.49} \quad (5.32)$$

Equations from (5.22) to (5.32) are defined in terms of the dimensionless parameters α , β_r , δ , and γ which are:

$$\begin{aligned} \alpha &= \frac{x-t}{y-t}, \\ \beta_r &= \left(1 - \frac{1}{1 + \alpha\gamma + \gamma + \alpha\gamma^2} \right), \\ \delta &= \frac{t}{l}, \\ \gamma &= \frac{t}{x-t}. \end{aligned} \quad (5.33)$$

The comparison with the correlations from the literature is carried out by appropriately scaling the resulting values, to represent them in the frame of reference selected in this work. In particular, the Reynolds number is multiplied by the ratio between the hydraulic diameter and fin height y , and by the ratio between the flow cross-sections to correct for the reference length and velocity. The resulting f values are then scaled by the y/D_h ratio to match the reference length selected in this work. From the results, it is clear that, although various correlations have been developed over the years, none of them can be considered as universal, as the observed deviations change substantially depending on the geometric configuration. Although the correlations by Joshi and Webb [208] have a rather simple formulation, they show decent accuracy levels with respect to the numerical results, especially for small values of the fin pitch and offset length. On the other hand, other correlations featuring similar formulations such as those of Wieting [204] and Mochizuki *et al.* [205] show far lower accuracy levels. The relations by Manglik and Bergles [206] and Kim *et al.* [213] have much more complex formulations, but this does not always translate into improved

accuracy levels, as these two correlations produce similar estimations that tend to be better aligned with CFD results at large x^* and l^* values. As mentioned previously in Section 5.4, it is difficult to understand the cases in which the correlations from the literature represent extrapolations with respect to their actual validity ranges, and this could explain the very uneven accuracy levels observed in this comparison.

5.6.2. Colburn factor

Compared to friction and pressure losses, heat transfer effectiveness depends on one additional flow parameter, namely the Prandtl number. Traditionally, the dependence of the Nusselt number with respect to the Prandtl number is managed by introducing the Colburn factor:

$$j = \frac{\text{Nu}}{\text{Re} \text{Pr}^n} = \text{St} \text{Pr}^{n+1}, \quad (5.34)$$

where n is the scaling exponent for the Prandtl number, which is usually set to $1/3$ in most applications. Kays and London [202] discussed the validity of this assumption by comparing results from analytical laminar boundary layer solutions and turbulent flows in round tubes. The authors determined that the $1/3$ power law is a reasonable approximation to scale heat transfer results related to gas flows with $0.5 \leq \text{Pr} \leq 1.0$. However, they also questioned the use of this approach outside of gas flows and further suggested that the correct scaling exponent may depend on the particular geometry of the problem.

Following the suggestions from Kays and London, in this work the Prandtl scaling exponent is determined by performing four sets of CFD analyses at the Re-values of 2000 and 3000 and Pr-values of 0.71 and 7.0. The Prandtl exponent related to each geometric configuration is determined by the following equation:

$$n = \frac{\log \text{Nu}|_{\text{Pr}=7} - \log \text{Nu}|_{\text{Pr}=0.71}}{\log 7 - \log 0.71}. \quad (5.35)$$

The results are then averaged over the two considered Re-values to derive the final n distribution. Figure 5.13a shows the contour plot of the Prandtl number exponent obtained from CFD results scaled by its canonical value $1/3$. Most of the geometric configurations feature n -values very different from $1/3$: for fins with small pitch and offset length the scaling exponent is about half of its canonical value, while it is twice that for configurations with large x^* -values. The classical value of $1/3$ appears as a good approximation only for a limited number of configurations, which are identified by the white region in the plot of Figure 5.13a. Lastly, it is interesting to note that the sensitivity of the scaling exponent with respect to the offset length is pronounced only for small pitch values, while it almost disappears as x^* approaches its maximum value. The CFD derived n -values are also employed to

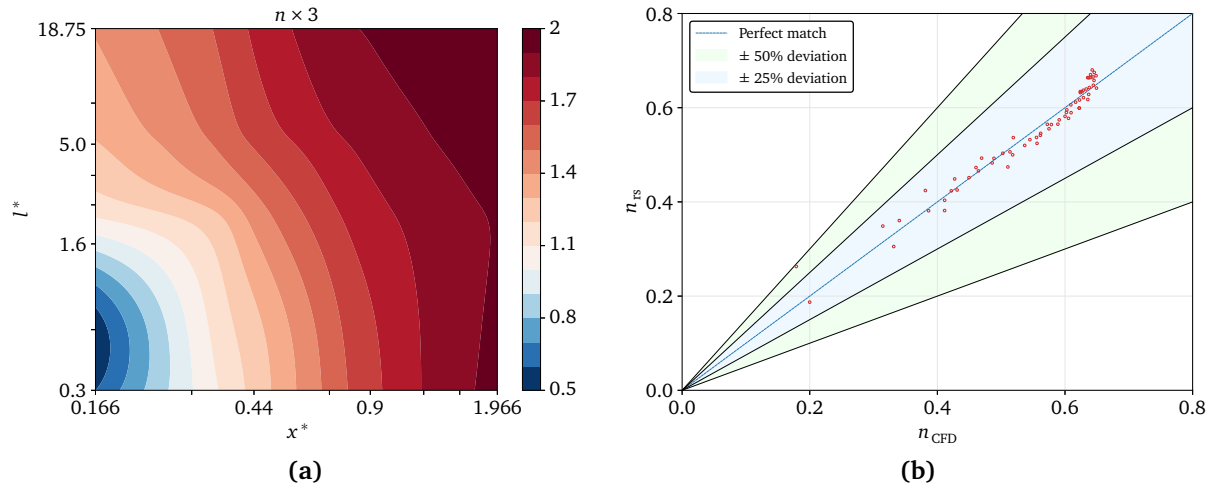


Figure 5.13. Contour plot of the Prandtl scaling exponent obtained by CFD normalised by its canonical value of $1/3$ (a) and deviations associated with the developed correlation (b). The x^* and l^* in (a) are scaled logarithmically, but the labels report linearly scaled values for improved readability.

derive a correlation as a function of the dimensional parameters:

$$n = (0.259 (l^*)^{-1/3} + 0.072) \log x^* + 0.044 \log l^* + 0.569. \quad (5.36)$$

Equation (5.36) allows to obtain the value of the Prandtl exponent with a very good approximation, as the average deviation with respect to the CFD results is less than 4% as shown in Figure 5.13b. Equation (5.36) is a good approximation for $0.71 \leq \text{Pr} \leq 7$ and $2000 \leq \text{Re} \leq 3000$, while outside these ranges this relation should be used with care if employed to scale heat transfer results unrelated to this work, as lower accuracy levels could be observed. In the present work these uncertainties are avoided by consistently using these n -values to compute the values of Colburn factor for all the considered flows, and purposely deriving correlations for the resulting j -values. This approach allows for the perfect alignment of heat transfer results in turbulent regime, i.e. for water and air flows, while noticeable discontinuities are expected between laminar (hydraulic oil) and turbulent results. Nevertheless, an accurate response surface for heat transfer performance can still be obtained by managing any jump or discontinuity by means of the smoothing function that was employed for the equivalent friction factor in Section 5.6.1. Of course, having an accurate description about the distribution of the Pr scaling exponent over the whole Reynolds and Prandtl numbers ranges would be ideal, but this would require a much higher computational effort.

The laminar and turbulent response surfaces for the Colburn factor are derived by means of the same procedure employed for the friction factor in Section 5.6.1:

$$\log j_1 = d_1 \log^{0.2} \text{Re} + d_2, \quad (5.37)$$

$$\log j_t = d_3 \log^2 \text{Re} + d_4. \quad (5.38)$$

The values of the d_m coefficients found in Equations (5.37) and (5.38) are expressed as a function of the geometric parameters by the two following relations:

$$d_1, d_2 = k_1 x^* + k_2 (x^*)^{k_3} + k_4 l^* + k_5 \log l^* + k_6 \log x^* \log l^* + k_7, \quad (5.39)$$

$$d_3, d_4 = k_1 (x^*)^{k_2} + k_3 (l^*)^{k_4} + k_5 \log l^* + k_6, \quad (5.40)$$

where the values of k_n coefficients are again determined via the curve fitting procedure described in the introductory part of Section 5.6. The resulting k_n -values are reported in Table 5.6.

The laminar and turbulent correlations are combined by means of the same smoothing function used for the equivalent Darcy friction factor, which is expressed by Equation (5.19). The transition Reynolds number is again derived by means of Equations (5.20) and (5.21), and the final response surface assumes the familiar form:

$$\log j = s_{\text{Re}} \log j_t + (1 - s_{\text{Re}}) \log j_l. \quad (5.41)$$

The charts of Figure 5.14 show numerical j factor results and a comparison with some of the most relevant correlations developed through the years:

- Joshi and Webb [208]

laminar flow: $\text{Re}_{D_h} \leq \text{Re}_{D_h}^*$

$$j = 0.53 \text{Re}_{D_h}^{-0.5} \left(\frac{l}{D_h} \right)^{-0.15} \alpha^{-0.14}, \quad (5.42)$$

turbulent flow: $\text{Re}_{D_h} \geq \text{Re}_{D_h}^* + 1000$

$$j = 0.21 \text{Re}_{D_h}^{-0.40} \left(\frac{l}{D_h} \right)^{-0.24} \left(\frac{t}{D_h} \right)^{0.02} \quad (5.43)$$

Table 5.6. Values of k_n coefficients in Equations (5.39) and (5.40).

	k_1	k_2	k_3	k_4	k_5	k_6	k_7
d_1	50.1014	-46.8119	1.06548	-0.0633	0.07918	1.06718	-8.5546
d_2	18.2244	-24.6789	0.76276	0.07360	-0.3988	-0.8820	10.3469
d_3	-111.656	0.000139	-6.7022	0.01874	0.29932	18.3093	-
d_4	1.4508×10^{-18}	-21.3283	45.7714	0.03263	-3.6445	-47.141	-

- Manglik and Bergles [206]

$$j = 0.6522 \text{Re}_{D_h}^{-0.5403} \alpha^{-0.1541} \delta^{0.1499} \gamma^{-0.0678} \times \left[1 + 5.269 \cdot 10^{-5} \text{Re}_{D_h}^{1.340} \alpha^{0.504} \delta^{0.456} \gamma^{-1.055} \right]^{0.1}, \quad (5.44)$$

- Mochizuki *et al.* [205]

laminar flow: $\text{Re}_{D_h} < 2000$

$$j = 1.37 \left(\frac{l}{D_h} \right)^{-0.25} \alpha^{-0.184} \text{Re}_{D_h}^{-0.67} \quad (5.45)$$

turbulent flow: $\text{Re}_{D_h} \geq 2000$

$$j = 1.17 \left(\frac{l}{D_h} + 3.75 \right)^{-1} \left(\frac{t}{D_h} \right)^{0.089} \text{Re}_{D_h}^{-0.36} \quad (5.46)$$

- Wieting [204]

laminar flow: $\text{Re}_{D_h} \leq 1000$

$$j = 0.483 \left(\frac{l}{D_h} \right)^{-0.162} \alpha^{-0.184} \text{Re}_{D_h}^{-0.536} \quad (5.47)$$

turbulent flow: $\text{Re}_{D_h} \geq 2000$

$$j = 0.242 \left(\frac{l}{D_h} \right)^{-0.322} \left(\frac{t}{D_h} \right)^{0.089} \text{Re}_{D_h}^{-0.368} \quad (5.48)$$

- Kim *et al.* [213]

for $\beta_r < 20\%$

$$j = e^{1.96} \alpha^{-0.098} \delta^{0.235} \gamma^{-0.154} \text{Re}_{D_h}^{0.0634 \ln \text{Re}_{D_h} - 0.623} \text{Pr}^{0.0348} \quad (5.49)$$

for $20\% \leq \beta_r < 25$

$$j = 1.06 \alpha^{-0.1} \delta^{0.131} \gamma^{-0.08} \text{Re}_{D_h}^{0.0323 \ln \text{Re}_{D_h} - 0.856} \text{Pr}^{0.0532} \quad (5.50)$$

for $25\% \leq \beta_r < 30\%$

$$j = e^{1.3} \alpha^{0.004} \delta^{0.251} \gamma^{-0.031} \text{Re}_{D_h}^{0.0507 \ln \text{Re}_{D_h} - 1.07} \text{Pr}^{0.051} \quad (5.51)$$

for $30\% \leq \beta_r \leq 35\%$

$$j = 0.2 \alpha^{-0.125} \delta^{0.21} \gamma^{-0.069} \text{Re}_{D_h}^{0.0005 \ln \text{Re}_{D_h} - 0.338} \text{Pr}^{0.0549} \quad (5.52)$$

Also in this case, when possible, the results coming from the correlations have been suitably scaled to adapt them to the reference quantities selected in this work. The explicit calculation of the Prandtl number exponent allows for a perfect alignment of the Colburn factors in turbulent regime, while considerable jumps occur near the laminar-turbulent transition. It is very interesting to note that with decreasing offset length and pitch, the discontinuity between laminar and turbulent results tends to be less pronounced until it almost disappears for the configuration with $x^* = 0.166$ and $l^* = 0.3$. Although the Prandtl number exponent was computed from results associated with Pr-values of 0.71 and 7, it represents an excellent approximation for scaling results referred to the much higher value of Prandtl number of 190, provided that they refer to a turbulent flow. This can be noted by examining the Colburn factor values obtained at $\text{Re} = 500$ and $\text{Pr} = 190$, which are perfectly aligned with water and air results. These results suggest that, in offset-strip fins, the Prandtl number exponent is most influenced by the flow regime and that this behaviour is very sensitive to the geometric configuration. On the other hand, if n is computed from reliable results, it can be considered valid over a large Prandtl number range, that effectively covers all the fluids of interest when dealing with compact heat exchangers.

In general, the correlations from the literature represent an acceptable approximation for the numerical results obtained in turbulent flow conditions and for small pitch and offset length values, while the deviations tend to increase with growing x^* and l^* . In laminar regime most of the correlations from the literature tend to greatly overestimate the Colburn factor for most of the geometric configurations. Nevertheless, it should be emphasised that most of these correlations were derived from experimental tests and CFD simulations related to air flows, and they were obtained by using the canonical value of $1/3$ for the Prandtl scaling exponent, therefore large deviations should be expected [251, 252].

The Colburn factor response surface developed in this work is in good agreement with the numerical results, as shown by the plot of Figure 5.15b. The mean deviation between the numerical results and the response surface is 8.3%.

5.6.3. Validation

The obtained response surfaces are validated by computing the internal head losses and overall heat transfer rates of three heat exchangers equipped with offset-strip fins that are part of the case study of Chapter 4, and comparing the results with the available experimental data. The details of the PFHE configurations considered are conveniently reported in the following.

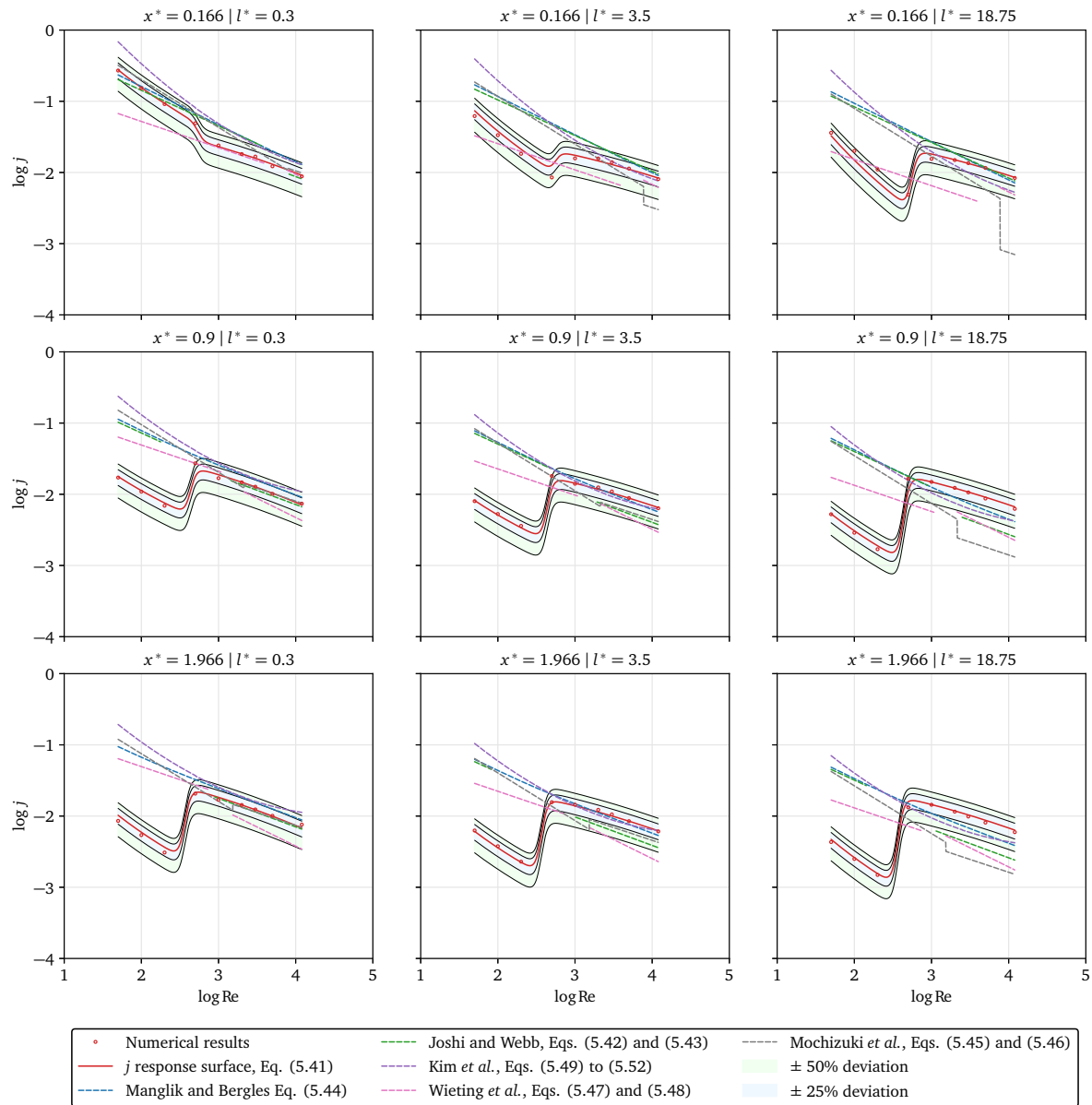


Figure 5.14. Colburn factor: numerical results, developed response surface, and a comparison with the most relevant correlations from the literature for nine selected geometric configurations.

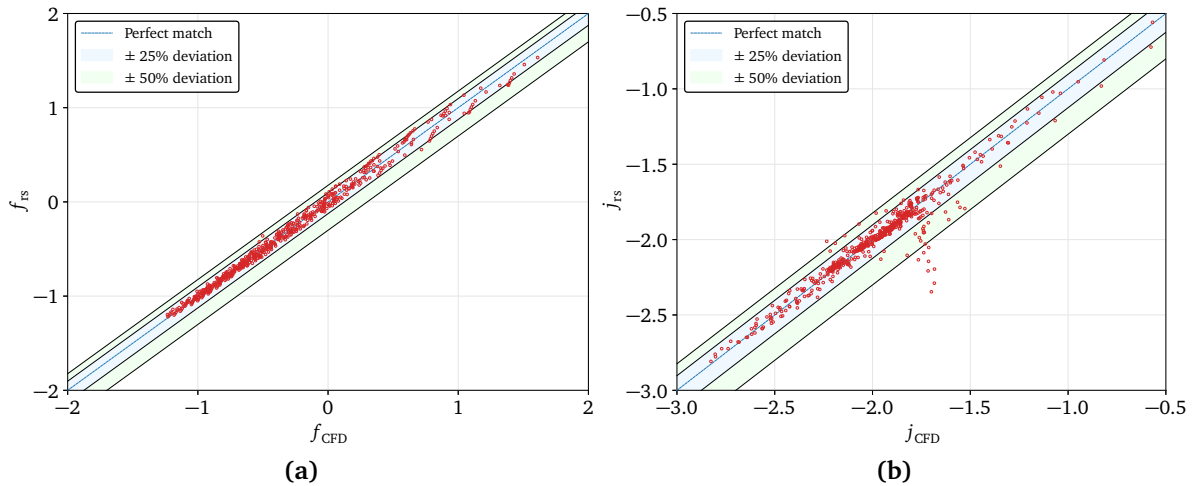


Figure 5.15. Deviations between numerical results and estimations from the derived response surfaces for the equivalent Darcy friction factor (a) and Colburn factor (b).

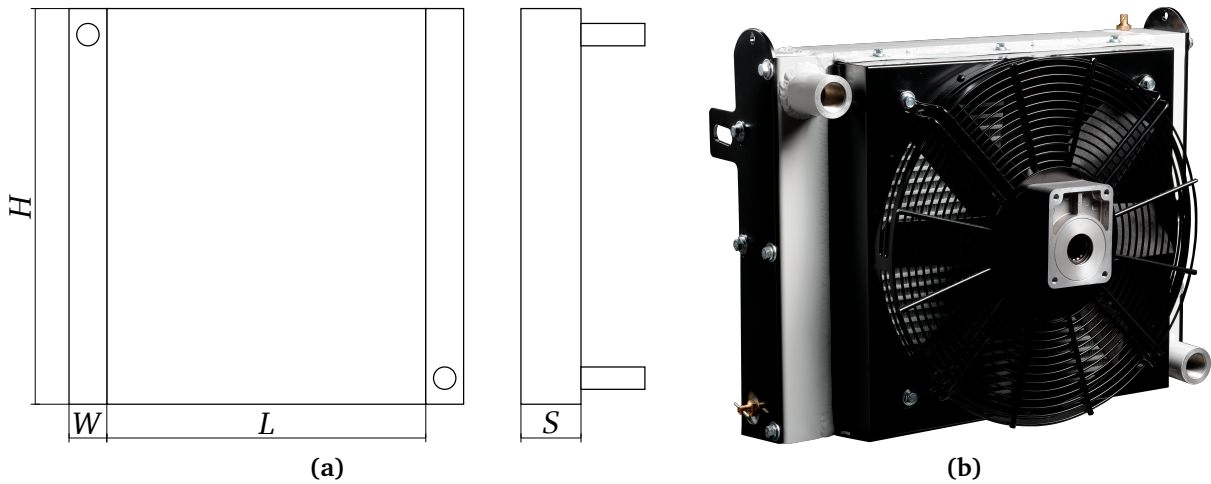


Figure 5.16. Schematic of the external geometry and relevant dimensions of the three examined oil coolers (a) and an actual picture of one of the tested coolers (b).

The examined units are oil coolers intended for lubricating or hydraulic oils, their external geometry is depicted in the drawing of Figure 5.16a, along with their most relevant dimensions: the height H , the main channel length L , the width of the two manifolds W , and the thickness S . The dimensional values of the coolers are reported in Table 5.7, along with an identification acronym for future reference. A picture depicting one of the actual heat exchangers is shown in Figure 5.16. The internal channels of the three oil coolers feature the same exact offset-strip fin geometry, with a height of $y = 3$ mm, a pitch of $x = 1.5$ mm, an offset length of $l = 5$ mm, and a wall thickness of $t = 0.2$ mm. The front and top views of the internal fin are depicted in the drawing of Figure 5.17a, together with the previously listed dimensions. In the external channels of each cooler, wavy fins are employed to enhance heat transfer. They feature a triangular cross-section and their geometry is shown in the drawing

of Figure 5.17b, along with the main dimensions. The internal losses and overall heat transfer rates associated to the three oil coolers are evaluated by using the global models for head losses and heat transfer of PFHEs developed in Chapter 4, and substituting the correlations for the thermal-hydraulic performance of the offset-strip fin in Figure 5.17a, that were derived in Section 4.5.1, with the response surfaces developed in this study. The details of the modelling methodologies used to derive the global heat-transfer performance of the PFHEs configurations considered are discussed in detail in Sections 4.3.4.1 and 4.3.4.2.

Figure 5.18 shows the comparison between the numerical results obtained in the present work, the corresponding results from Chapter 4 and the experimental data, regarding internal head losses and heat transfer rates. In these plots the values of pressure drop and heat transfer rate are scaled by their respective maximum experimental value. Numerical and experimental results are in very good agreement, as in both plots almost all the tested points feature deviations lower than 20%. On average, the deviation between the numerical and experimental values of the heat transfer rate is 8.8%, which is a slight improvement over the numerical results from Chapter 4, that feature a mean deviation of 9.9% with respect to the experimental data. The mean deviation between the numerical and experimental results on internal head losses is of 6.8%. These results are particularly noteworthy, since the response surfaces developed in this work were obtained considering only nominal geometric data and close to ideal fin geometries. On the other hand, finned surfaces in actual heat exchangers may feature severe shape deformations and residual metal scraps resulting from the manufacturing process, which, aside from experimental uncertainties, may be the major cause of deviations between numerical and experimental results. The response surfaces developed in this work allow for obtaining numerical results that, on average, are as accurate as the ones obtained from dedicated CFD analyses, thus corroborating the validity of the present analysis.

Table 5.7. External dimensions of the Geometrical parameters of the three examined oil coolers.

ID	S [mm]	H [mm]	W [mm]	L [mm]	N_{ch}
O80	80	304	60	500	20
O94	94	309	60	500	21
O113	113	305	60	500	20

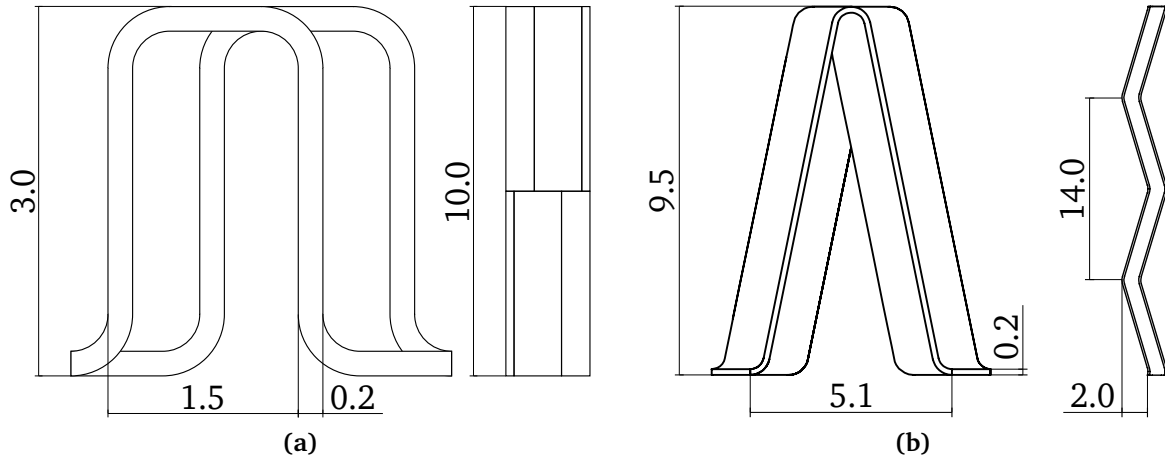


Figure 5.17. Drawings depicting the front and top views of the offset-strip (a) and wavy (b) extended surfaces employed in the internal and external channels, respectively.

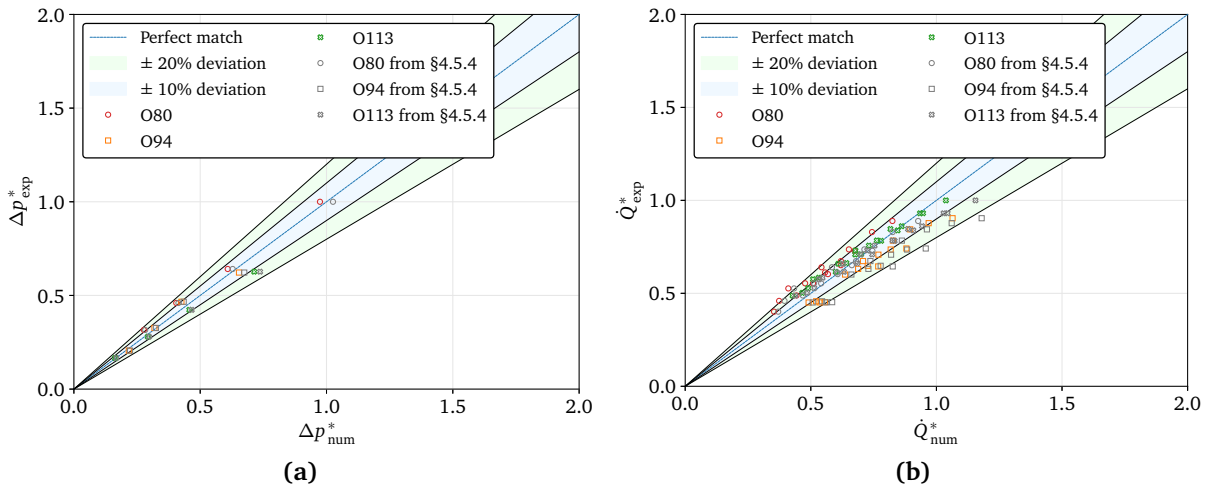


Figure 5.18. Comparison between the numerically obtained values of internal head losses (a) and heat transfer rates (b) with the numerical results from Chapter 4 and the available experimental data.

5.7. Concluding remarks

In this work, a comprehensive study on the heat transfer and friction properties of offset-strip fins with vertical walls, radiused corners, and centred offset was presented. The dependence of the equivalent Darcy friction factor f and Colburn factor j on geometric and flow parameters was evaluated through a DOE procedure, where each configuration was studied by CFD analysis of a periodic fin module. Numerical results were used to derive response surfaces for f and j with well-defined applicability ranges, improving on the correlations available in the literature, which are developed on configuration sets that are highly unevenly distributed over the design space. The derived response surfaces showed mean accuracy values of $\pm 8.4\%$ with respect to the CFD results, and mean and maximum deviations of $\pm 7.8\%$ and $\pm 20\%$, when validated against experimental data on complete heat exchangers. Novel

results on the effect of the Prandtl number on heat transfer performance were also provided through the direct calculation of the Pr scaling exponent, n , from numerical results. These results showed that, at least in the case of offset-strip fins, the common assumption of $n = 1/3$, typical of the Chilton-Colburn analogy and used in the vast majority of studies, can be regarded as valid and accurate only for a small subset of geometric configurations, and that n varies significantly as a function of the geometric parameters. Hence, for this specific case, an original correlation for the Prandtl scaling exponent was proposed here. In conclusion, the novel data and correlations proposed in the present work are expected to have a positive impact on the design and performance assessment of heat exchangers with offset-strip fins. Furthermore, the predictive model proposed for the Prandtl scaling exponent represents a promising approach to accurately and systematically model Prandtl number effects on heat transfer effectiveness.

CHAPTER 6

Parametric analysis of wavy fins

6.1. Introduction

6.1.1. Engineering and technical background

Compact heat exchangers working in harsh environments are often equipped with wavy or corrugated extended surfaces, as these present a far higher resistance to clogging and fouling compared to other commonly used fin configurations [121]. Specifically, Wavy Fins (WFs) see regular use in the external channels of heat exchangers mounted on agricultural and earth-moving machines, where the presence of dirt particles and pieces of straw severely limits the use of common extended surface arrangements as offset-strip fins, as these fragments can easily get stuck in between the tight rows of staggered ducts, adversely affecting the performance of the coolers and considerably complicating maintenance and cleaning operations of electric and hybrid agricultural machines [271].

Wavy fins arrays are constituted by a series of ducts with uniform cross-section, usually of rectangular or triangular shape, which is developed along a sinusoidal path defined by amplitude and wavelength. Fin configurations with a triangular or herringbone development such as those presented in Chapter 4 are also available [272], although sinusoidal ducts usually allow for a better heat transfer effectiveness to head losses ratio [273]. The corrugated side walls induce the periodic formation and disruption of counter-rotating vortices in the troughs formed by the walls [274]. These vortices cause the local interruption of the boundary layer, favouring heat transfer in correspondence of the recirculation regions [214]. The ratio between heat transfer enhancement and increase of the pressure losses is heavily affected by the fin pitch [217], the amplitude and wavelength of the corrugated channels, and the flow rate [121]. At the design stage, the influence of geometric and

NOMENCLATURE*Latin letters*

$2a$	peak-to-peak amplitude [m]
A	area [m ²]
b_m	coefficients in Equation (6.9)
d_m	coefficients in Equation (6.10)
D_h	hydraulic diameter [m]
$E(\cdot)$	complete elliptic integral of the second kind
f	equivalent Darcy friction factor [–]
j	Colburn factor [–]
l_a	arc length of a sine wave [m]
Nu	Nusselt number [–]
P	wetted perimeter [m]
Pr	Prandtl number [–]
Re	Reynolds number $u y / \nu$ [–]
t	fin thickness [m]
T	temperature [K]
$\vec{u}$	fluid velocity [m s ^{–1}]
w	fin wavelength [m]
x	fin pitch [m]
y	fin height [m]

Greek letters

γ	corrugation ratio $2a/\gamma$ [–]
ϑ	dimensionless temperature difference [–]

Subscripts

b	bulk
F	Fanning
i	inlet
o	outlet
rs	response surface
s	solid
t	turbulent
w	wall

Superscripts

*	dimensionless
---	---------------

flow parameters on the thermal-hydraulic performance of wavy fins is often evaluated by means of empirical models based on numerical and experimental data available in the literature. However, these models often cover the design space in a very uneven fashion with narrow applicability ranges [216], severely limiting the number of configurations that can be studied and introducing considerable uncertainties when they are employed in extrapolation. In an industrial context, the lack of confidence introduced by these shortcomings often forces the designers to evaluate individual configuration through experimental tests of prototypes, which considerably drive up the design costs and lead times.

This Chapter presents the first steps and preliminary results of a parametric analysis of the thermal-hydraulic performance of wavy fins aimed at the derivation of continuous multivariate response surfaces for the equivalent Darcy friction factor f and Colburn factor j analogous to those developed in Chapter 5 for offset-strip fins. This study is executed by following a Design Of Experiment procedure, where each configuration is analysed by developing a dedicated CFD model including a basic wavy fin geometry under periodic flow conditions.

6.1.2. Literature survey

One of the first studies on the performance of wavy fins was conducted by Kays and London [202], who presented heat transfer and friction data for three fin cores with equal wave length, similar amplitude, and different hydraulic diameters. The experiments were performed by considering several air flows, ranging from laminar flows at $Re_{D_h} = 300$ to turbulent ones at $Re_{D_h} = 10000$. In [215] the naphthalene sublimation technique was used to measure local and average heat transfer properties of flows in corrugated channels. The authors concluded that in laminar flows, which are stable up to $Re_{D_h} = 1000 - 1200$, the mean heat transfer coefficient only slightly exceeded the one of an equivalent straight channel most likely due to the increased surface area [220], while in turbulent flows at $Re_{D_h} = 6000 - 8000$ the heat transfer enhancement over plain fins was almost threefold. The configurations studied by Wang, Fu, and Chang [275] showed less pronounced improvements of 55% to 70% over straight channels, however, these advantages came at the cost of 66% to 140% increase of friction losses. Ali and Ramadhyani [276] observed a much lower limit for the stability of laminar flows at roughly $Re_{D_h} = 500$, due to the dependence of the transition Reynolds number on the geometric configuration [277]. Through the comparison of several corrugated channels against plain ducts, the authors determined that wavy fins provide optimal heat transfer enhancement in the range $1750 \leq Re_{D_h} \leq 2000$, while at higher Re_{D_h} -values the heat transfer performance over pressure losses benefits tend to diminish due to increased turbulence. Wang *et al.* [278] linked the degree of heat transfer enhancement provided by wavy fins to variations of their corrugation angle and wavelength to height ratio, while the fin pitch had a relevant effect only at low corrugation angles. Limited effects of fin pitch variations on heat transfer performance were also noted by Chen and Ren [279], particularly in low Reynolds flows. Instead, Zhang *et al.* [219] and Manglik *et al.* [214] demonstrated that excessively small fin aspect ratios can negatively affect the effectiveness of wavy fins as they magnify viscous effects which prevent the formation of vortices in the valleys formed by the corrugated walls. On the other hand, the strength of these vortices increases with growing corrugation ratios and Reynolds number, which induces Nusselt number improvements that roughly follow a 2/3 power law, which is typical of turbulent flows with local separations and reattachments [280].

The experimental data presented in [280] was used as a reference by Ciofalo *et al.* [281] for the validation of several numerical methods for modelling fluid flow and heat transfer in corrugated channels. The authors tested several turbulence modelling approaches including the standard $k-\varepsilon$, the Lam and Bremhorst low-Re $k-\varepsilon$ implementation [282], a Large Eddy Simulation (LES) method based on the Smagorinsky-Lilly model [283,284], and Direct Numerical Simulation (DNS). Among the tested models, the low-Reynolds $k-\varepsilon$ and LES approaches provided the overall best agreement between numerical and experimental data across the tested geometric configurations and Re-values. On the other hand, the $k-\varepsilon$ approach using wall functions provided acceptable results only at very high Re-values,

while the deviations between numerical and experimental data increased with decreasing Reynolds number values. LES approaches were used in [223] to gain deeper knowledge on the flow features responsible for the heat transfer enhancement in wavy fins. Ismail and Velraj [285] employed the k - ε model for the analysis of several wavy fin configurations and used numerical results to derive correlations for the equivalent Fanning friction factor f_F and Colburn factor j . Numerical results on heat transfer were in good agreement with experimental data, while substantial deviations were noted in case of head losses, as the numerical values of f_F underestimated their experimental counterparts by roughly 1.6 times.

Asako and Faghri [218] developed and validated a Finite Volume computational approach based on a transformation of coordinates which maps the physical domain of a wavy fin with sinusoidal longitudinal development to a simple rectangular geometry. Manglik *et al.* [214] performed laminar numerical analyses focusing on air flows in the range $10 \leq Re_{D_h} \leq 1000$. Similarly, Metwally and Manglik [220] developed two-dimensional solutions for laminar flows in the same Re_{D_h} -range. The study by Aliabadi *et al.* [221] is one of the few considering working fluids other than air, as the authors developed correlations for the Colburn factor specific to each fluid considered, which were air, water, and ethylene glycol.

Through the years several authors proposed empirical models for the prediction of the Colburn and friction factor in wavy fins. These are usually provided in the form of polynomials that are functions of the Reynolds number and selected ratios between geometric parameters. In these equations the coefficients are determined through curve fitting procedure based on experimental data as in [217, 286], and numerical results [221, 287]. Other authors as Awad and Muzychka [232] developed predictive models on the basis of analytical and asymptotic considerations for the laminar flow regime.

All these predictive models are very convenient tools to derive preliminary estimates on the thermal-hydraulic performance of wavy fins. However, as it will be shown later, many of these correlations are developed on the basis of sets of configurations which are very unevenly distributed across the design space. This translates into applicability windows that are often narrow and difficult to define, thus limiting the correct use of the empirical models, which are often employed in extrapolation with undefined accuracy levels.

6.1.3. Modelling approach

This study is focused on wavy or corrugated fins with a rectangular cross-section and a sinusoidal longitudinal development. The side walls have a uniform thickness, they are in-phase with each other and are slightly inclined with respect to vertical axis. The geometric parameters of the fins are defined as in most of the papers on corrugated channels present in the literature. In addition, accurate analytical formulas are provided for the computation of key geometric features as the hydraulic diameter, wetted perimeter, flow cross-sectional

area, and heat transfer surface.

The relationship between the geometric and flow parameters of wavy fins and their thermal-hydraulic performance is studied by means of a Design of Experiment procedure, which is performed using computational methods. The levels of the geometric parameters are selected with the aim of covering the most relevant applications for vehicular applications, and a full factorial DOE is performed with respect to the dimensionless pitch, wavelength and amplitude. To this end, a CFD model comprising a unitary fin geometric under periodic flow conditions is developed for each geometric configuration. The numerical solution is attained by means of a validated conjugate heat transfer solver based on the Finite Volume method.

Preliminary numerical results on heat transfer and friction losses are used to derive response surfaces for the equivalent Darcy friction factor and Colburn factor as a function of the numerical parameters. The dependency on the Reynolds number is introduced a posteriori by using scaling laws available in the literature, due to the limited amount of numerical results available. The validity of this approach is assessed by comparing the outcomes of the obtained response surfaces with some numerical and experimental results on single fins and complete heat exchangers that were presented in Chapter 4. The initial results are promising as the generalised response surfaces can produce numerical results as accurate as those retrieved from dedicated CFD models of the geometric configurations considered.

6.2. Problem statement

The present parametric analysis is performed on wavy fins presenting a rectangular cross-section with slightly sloped side walls, and sinusoidal longitudinal development. Figure 6.1 shows schematics of the front and top views of the wavy fin geometry considered, along with the most relevant geometric parameters. The pitch and height of the fin are denoted by x and y , respectively. The thickness of the walls is assumed uniform and it is represented by t . On the top view, w stands for the wavelength, while $2a$ refers to the peak-to-peak amplitude of the sine wave. The side walls are slightly inclined forming an angle $\alpha/2$ with the vertical axis.

The flow cross-sectional area of the examined wavy fin geometry is given by:

$$A_f = x y - A_w, \quad (6.1)$$

where A_w is the section of the overall frontal area occupied by the walls of the fin:

$$A_w = 4 t \left(\frac{x}{4} - t \right) + \frac{2 t y}{\cos \alpha/2}. \quad (6.2)$$

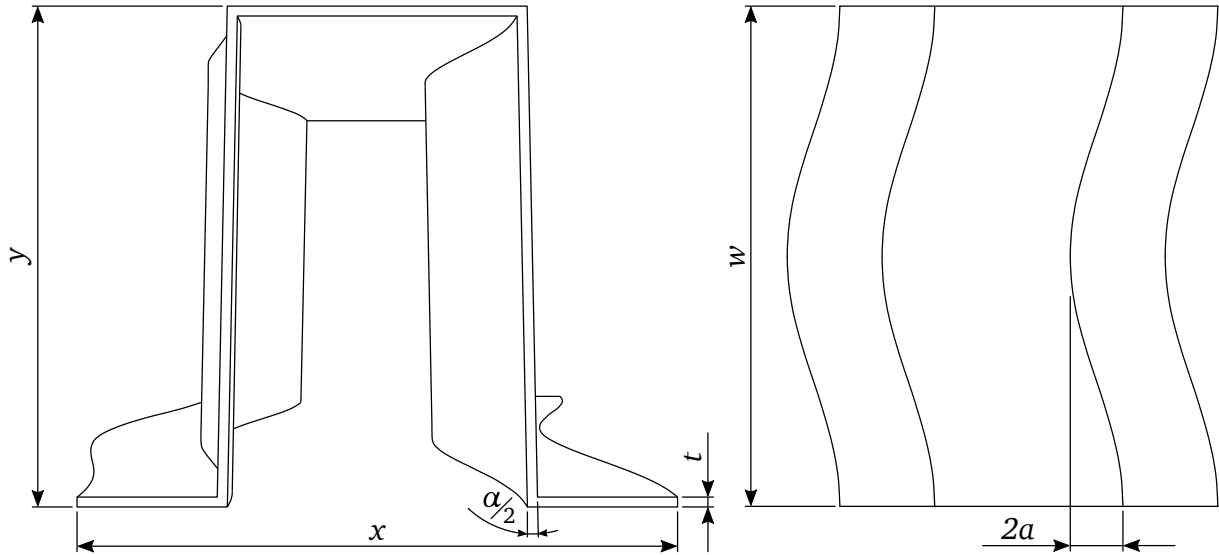


Figure 6.1. Front (left) and top (right) views of the examined wavy fin geometry, along with the most significant dimensions.

The wetted perimeter is:

$$P = 4 \left[2 \left(\frac{x}{4} - t \right) + (y - t) \left(\tan \alpha/2 + \frac{1}{\cos \alpha/2} \right) \right]. \quad (6.3)$$

The heat transfer area includes the fluid-solid interface along with the sections of the top and bottom surfaces which are in contact with the fluid:

$$A_{\text{ref}} = P l_a, \quad (6.4)$$

where l_a is the arc-length of the sinusoidal wave [232]:

$$l_a = 2w \frac{\sqrt{1 + \gamma^2 \pi^2}}{\pi} E \left(\frac{\gamma \pi}{\sqrt{1 + \gamma^2 \pi^2}} \right). \quad (6.5)$$

In Equation 6.5 $\gamma = 2a/w$ stands for the corrugation ratio of the wavy channel, while $E(\cdot)$ is the complete elliptic integral of the second kind. Examples of the flow cross-sectional area, wetter perimeter, and heat transfer area of the examined wavy fin geometries are depicted in the schematic of Figure 6.2.

6.3. Numerical methods

Wavy fins are analysed by developing Computational Fluid Dynamics models of basic unit geometries which are formed by considering the periodicity of the geometry and flow along the streamwise and spanwise directions. These numerical models are devised by considering the governing equations, boundary conditions, spatial discretisation schemes, and meshing parameters that were discussed in Chapter 5 as part of the parametric analysis of offset-strip

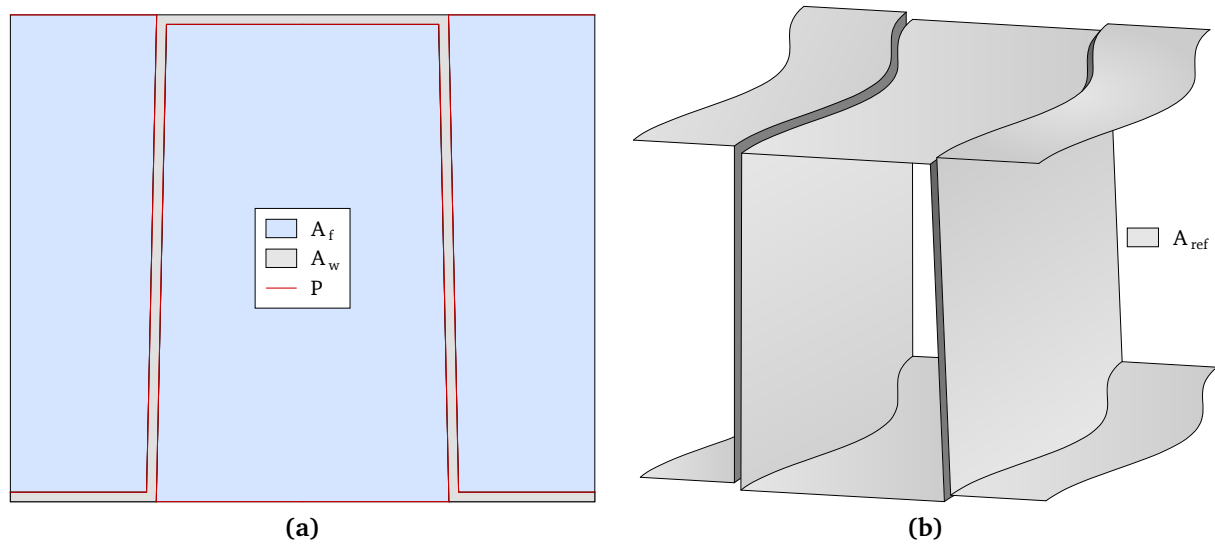


Figure 6.2. Flow cross-sectional area, wetted perimeter (a), and heat transfer area (b) of the examined wavy fins.

fins. In addition, a dimensionless framework analogous to those considered for the extended surfaces analysed Chapters 4 and 5 is adopted, where the height of the fins y is taken as the reference length and the remaining geometric parameters are scaled accordingly:

$$y^* = \frac{y}{y} = 1, \quad x^* = \frac{x}{y}, \quad w^* = \frac{w}{y}, \quad 2a^* = \frac{2a}{y}, \quad t^* = \frac{t}{y}. \quad (6.6)$$

The main differences between the models developed for wavy and offset-strip fins consist of the generation of the computational domains. In this case, the simpler geometry of wavy fins compared to OSFs allows for the generation of block structured computational grids. The mesh generation algorithm is implemented in a simple Glyph script for the commercial grid generation tool Pointwise. Figure 6.3 shows selected views of the computational mesh generated for the configuration defined by $x^* = 1.2$, $w^* = 1.6$, and $2a^* = 0.167$, where the fluid and solid regions are clearly highlighted.

6.4. Design of Experiment

The effects of the geometric and flow parameters of wavy fins on their thermal-hydraulic characteristics are assessed by means of a Design Of Experiment procedure. The non-dimensional pitch, wavelength, and amplitude are identified as the most relevant geometric parameters, while the inclination angle and thickness of the walls are considered of minor importance and are therefore assumed constant and excluded from the DOE. This assumption is also supported by common design practices associated with compact heat exchangers, where the selection of the thickness of the fins is often limited by the commercially available raw aluminium sheets and the need to guarantee the mechanical strength and stability of the heat exchangers. In addition, the inclination of the walls might be altered to facilitate

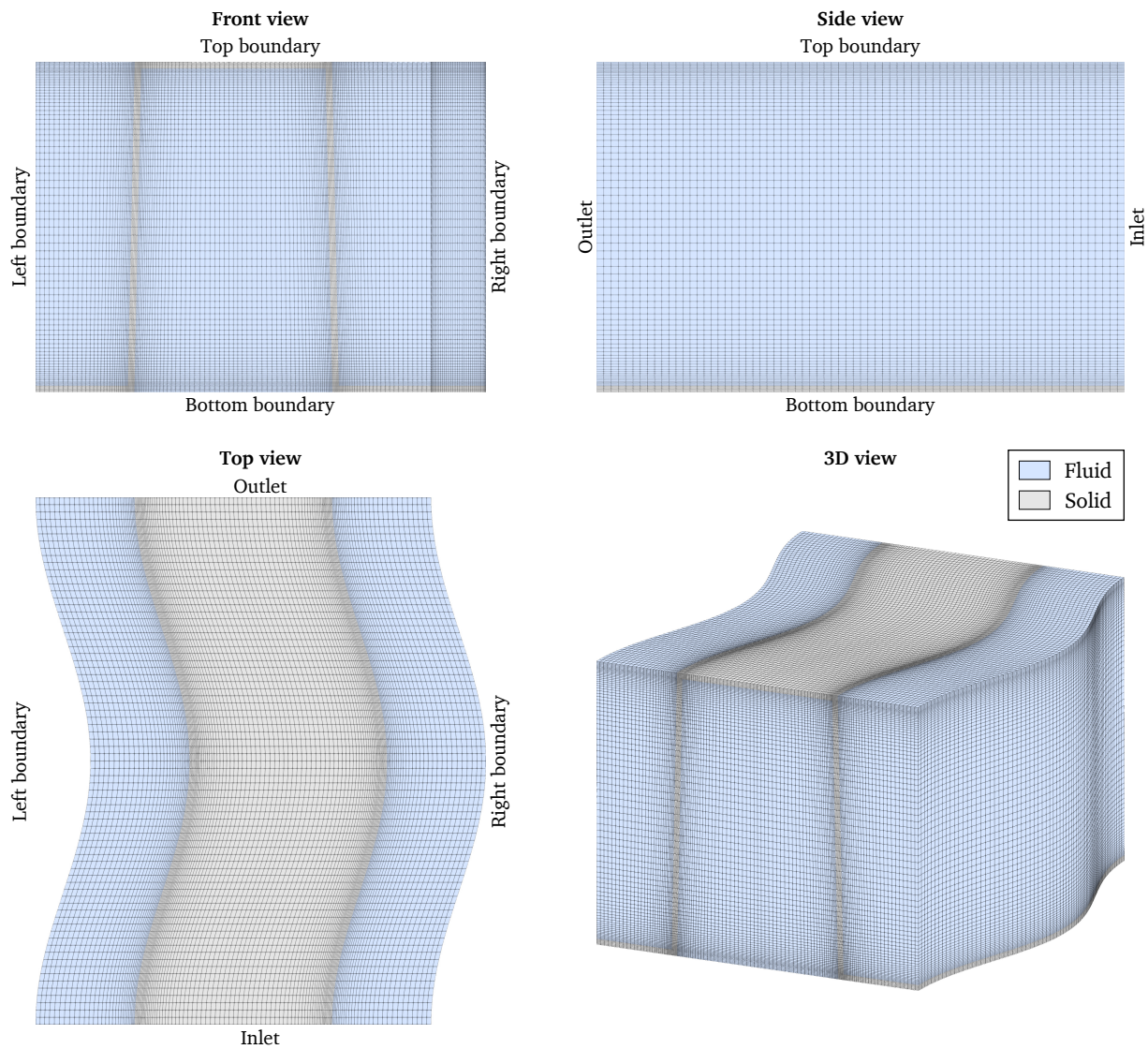


Figure 6.3. Selected views of one of the generated computational grids, related to the geometrical configuration defined by $x^* = 1.2$, $w^* = 1.6$, and $2a^* = 0.167$.

the manufacturing process of the fins and possibly reduced wear on the dies.

A full factorial DOE is performed with respect to the dimensionless fin pitch, wavelength, and amplitude and the flow regime, which is defined by couples of Reynolds and Prandtl numbers. Table 6.1 reports the levels of the geometric parameters x^* , w^* , and $2a^*$, which are selected to cover a range of configurations relevant for the design of compact heat exchangers used in vehicular applications, as those presented in the case study of Chapter 4. As previously anticipated, the thickness and inclination angle of the walls are assumed constant and set to $t^* = 0.02$ and $\alpha = 2.4^\circ$, respectively.

The geometric configurations considered in this work are shown in the three-dimensional plot of Figure 6.4a, where they are compared with those analysed in some of the most relevant studies on wavy fins available in the literature. The latter cover the design space in a very uneven and sparse manner, as most of the data points related to each publication are

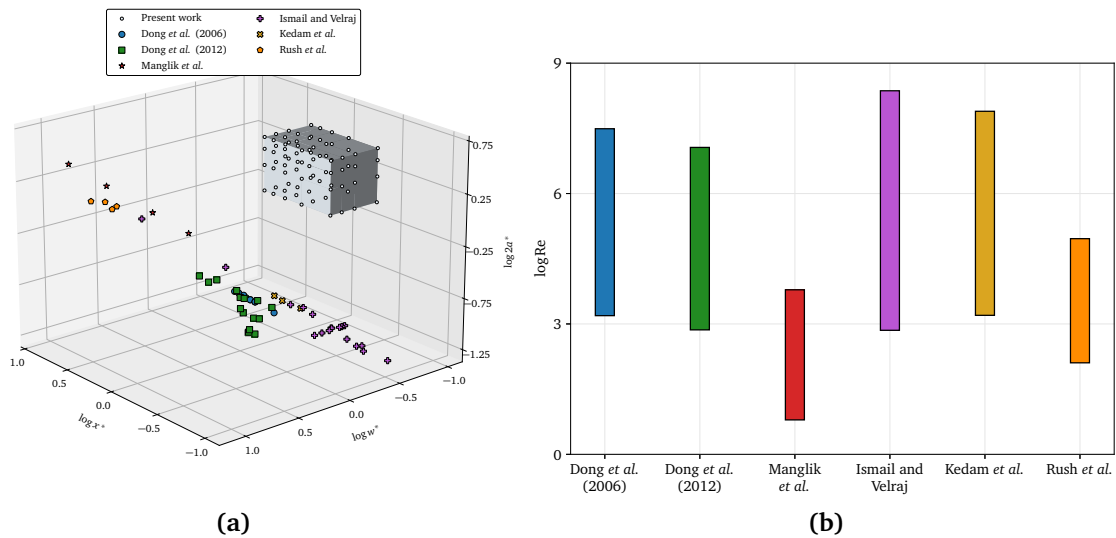


Figure 6.4. Comparison between the dimensional (a) and flow (b) parameters considered in this Chapter compared with those analysed in the literature.

densely confined in narrow regions associated with small deviations of the geometric parameters. As a result, none of the empirical models developed on the basis of these data sets can be considered universal, and their applicability ranges do not appear as well-defined. Furthermore, none of the considered contributions from the literature address the fin configurations which are most relevant for the design of heat exchangers such as those examined in Chapter 4.

These shortcomings are addressed by the present parametric analysis, which allows for the study of configurations of specific technical interest, and for the clear identification of the applicability ranges of the resulting response surfaces, which are represented by the box in Figure 6.4a.

Figure 6.4b reports the fin-height-based Reynolds number ranges considered in each study from the literature, showing that most of the fin configurations were tested in the range $10^3 \leq Re \leq 10^6$. In this case the values of the Prandtl number related to each study are not reported in the plot, as all the considered publications focused on air flows at $Pr = 0.7$.

The present study is still at a very early stage of development, therefore only a single Reynolds number value has been analysed thus far. The preliminary numerical results are presented and discussed in the following.

Table 6.1. Selected levels of the geometric parameters involved in the DOE analysis of wavy fins.

x^*	0.4	0.8	1.2	1.6	2.0
w^*	0.9	1.6	2.3	3.0	
$2a^*$	0.1	0.17	0.23	0.3	

6.5. Preliminary numerical results

Numerical flow field result are present by means of contour plots of dimensionless velocity magnitude and temperature difference, obtained on longitudinal planes positioned at half the height of the fins. As usual the dimensionless temperature difference is given by:

$$\vartheta = \frac{\bar{T}_w - T}{\bar{T}_w - \bar{T}_b}, \quad (6.7)$$

where $\bar{T}_w$ is the integral mean temperature over the heat transfer area A_{ref} , while $\bar{T}_b$ is computed as the mean of the mass flow weighted averages of temperature on the inlet and outlet surfaces. Figure 6.5 shows an example of these contour plots related to a wavy fin defined by $x^* = 1.2$, $w^* = 1.6$, and $2a^* = 0.167$ and to an air flow at $\text{Re} = 5000$. The velocity and temperature distributions closely resemble those obtained in Chapter 4 for wavy fins, with flow recirculation regions in correspondence of the troughs of the fins and velocity peaks in proximity of the crests.

Integral results on head losses and heat transfer rates are expressed by the equivalent Darcy friction factor and Nusselt number. The former is retrieved from the Darcy-Weisbach formula:

$$f = \frac{2 \gamma (\bar{p}_i - \bar{p}_o)}{w \rho \bar{u}_i^2}, \quad (6.8)$$

where overbars represent integral mean values over inlet (i) and outlet (o) sections. The equivalent Nusselt number is defined according to Equation (5.13), which is reported in Section 5.5. Figure 6.6 shows four-dimensional scatter plots of f and Nu derived from numerical results at $\text{Re} = 5000$ and $\text{Pr} = 0.71$, where each point marker identifies a single geometric configuration while the dependent variables are represented by linear colour distributions. As expected, both friction losses and heat transfer effectiveness tend to increase with the corrugation ratio of the channels, i.e. as the amplitude of the fins increases and the wavelength decreases. In addition, both f and Nu decrease with growing fin pitch x^* due to the increase of the hydraulic diameter as in straight channels [58].

6.6. Derivation of preliminary response surfaces

The preliminary numerical results obtained thus far are used as a basis for the derivation of response surfaces for the equivalent Darcy friction factor and Colburn factor. These response surfaces are obtained using the same approach which was extensively discussed in Section 5.6. However, this time the formulation of the equations is much more straightforward due to the very limited base data set. The response surfaces for the Darcy friction factor and Colburn factor are respectively:

$$\log f = b_1 (x^*)^{b_2} + b_3 (w^*)^{b_4} + b_5 (2a^*)^{b_6} + b_7 \left(\frac{2a^*}{w^*} \right), \quad (6.9)$$

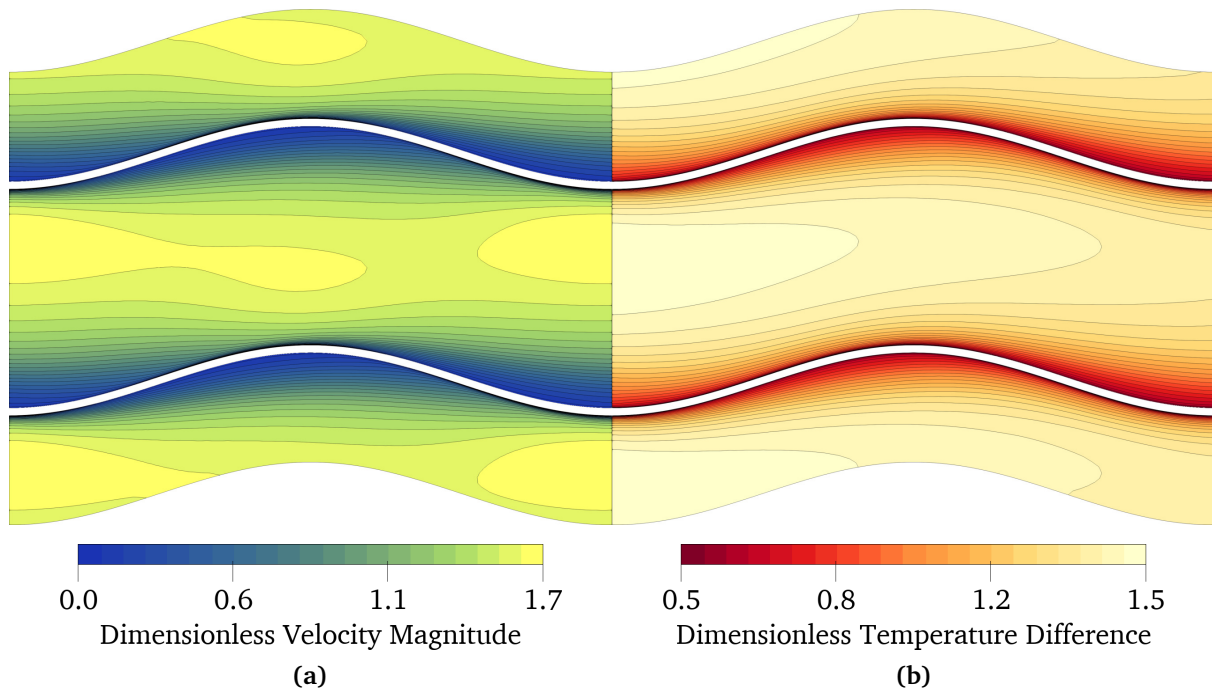


Figure 6.5. Contour plots of dimensionless velocity magnitude (a) and temperature difference (b), obtained on longitudinal planes located at half the height of the fins. These plots are associated with an air flow at $Re = 5000$ and $Pr = 0.71$.

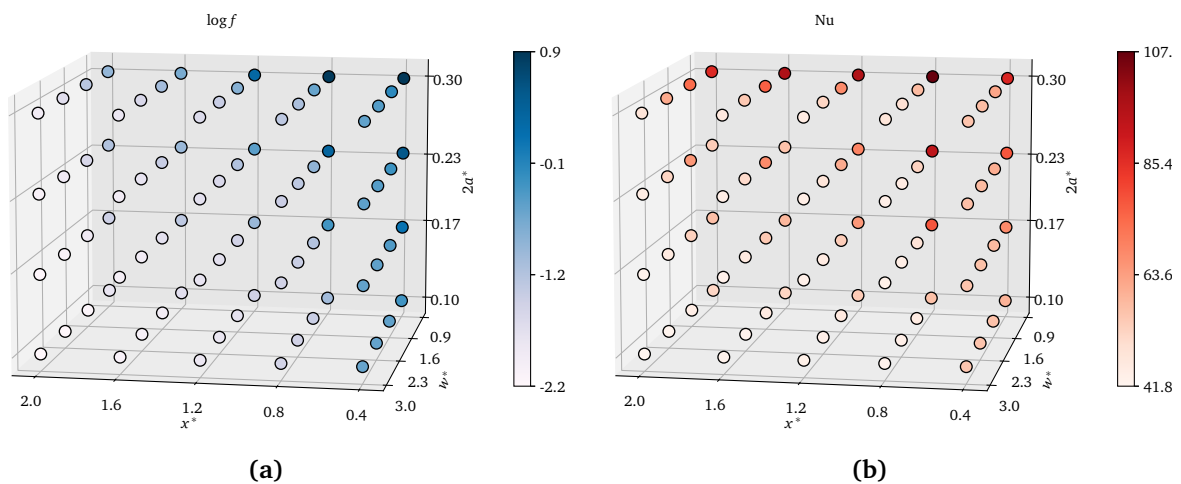


Figure 6.6. Four-dimensional scatter plots of the equivalent Darcy friction factor (a) and Nusselt number (b), derived from CFD results at $Re = 5000$ and $Pr = 0.71$. In these plots each round marker refers to one of the geometric configurations, while the dependent variable is represented by means of a colour distribution.

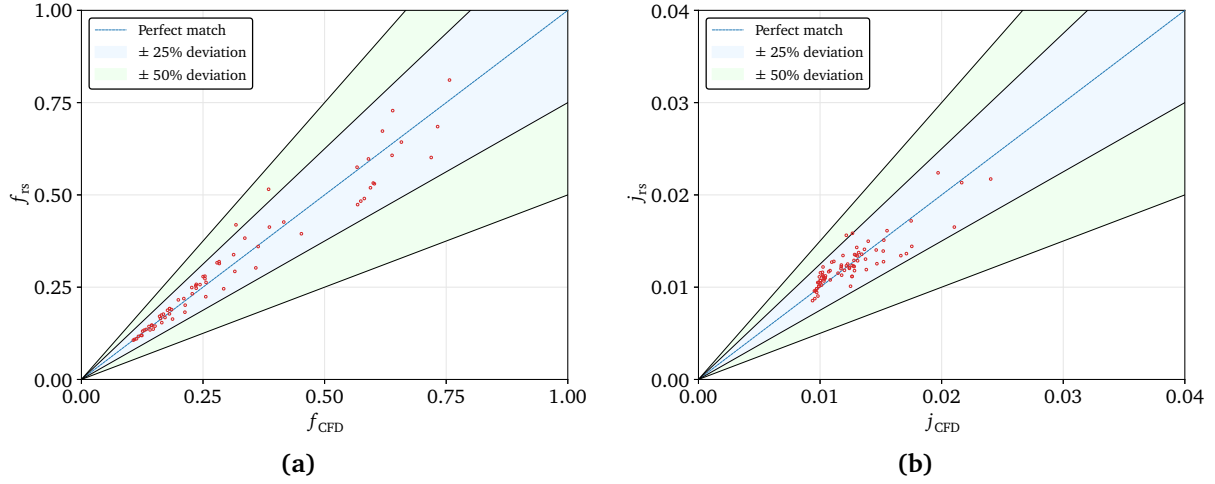


Figure 6.7. Deviations between numerical results and estimations from the derived response surfaces for the equivalent Darcy friction factor (a) and Colburn factor (b).

$$j = d_1 (x^*)^{d_2} + d_3 (w^*)^{d_4} + d_5 (2a^*)^{d_6} + \left(\frac{2a^*}{w^*} \right)^{d_7}, \quad (6.10)$$

where the b_m and d_m coefficients are listed in Table 6.2. The deviations between the outcomes of Equations (6.9) and (6.10) and the underlying CFD data points are shown in the plots of Figure 6.7.

Equations (6.9) and (6.10) hold only for $Re = 5000$, however, their validity can approximately be extended to other Re -values by introducing the scaling laws derived by Stasiek *et al.* [280]:

$$f = f|_{Re=5000} \left(\frac{Re}{5000} \right)^{-1/2}, \quad (6.11)$$

$$Nu = Nu|_{Re=5000} \left(\frac{Re}{5000} \right)^{2/3}. \quad (6.12)$$

The validity of this approach is assessed by comparing numerical results obtained from the above-presented response surfaces with the numerical and experimental results presented in Section 4.5 for the WF4 extended surface and the G94 and G113 heat exchangers. Figure 6.8 shows the distributions of equivalent Darcy friction factor and mean Nusselt number derived from Equations (6.9) to (6.12) with the CFD results and associated correlations obtained from the dedicated single fin CFD model of the WF4 extended surface.

Table 6.2. Values of b_m and d_m coefficients in Equations (6.9) and (6.10), respectively.

m	1	2	3	4	5	6	7
b_m	-1138.82	3.5382×10^{-4}	1295.79	8.7618×10^{-5}	-158.24	8.8538×10^{-4}	3.817
d_m	0.392	0.00247	-0.0024	0.7765	0.4115	0.00495	4.5636

At $Re = 5000$ there is a very good agreement between the outcomes of the response surfaces and the CFD results from Section 4.5 for both f and Nu . This confirms the validity of the present parametric analysis as the response surfaces return estimates on the thermal-hydraulic performance that are effectively equivalent to those obtained from dedicated CFD models, even though the WF4 configuration was not studied directly in the present DOE. In addition, the scaling laws proposed by Stasiek *et al.* [280] seem to correctly represents the scaling of f and Nu with respect to the Reynolds number, as the correlations stemming from the numerical results from Section 4.5 are well-matched by the above-described response surfaces, particularly in case of the mean Nusselt number.

The validation of the response surfaces is expanded by comparing the numerical and experimental results on external head losses and heat transfer rates of the G94 and G113 heat exchangers, which were presented in Section 4.5, with those obtained by substituting the correlations for f and Nu of WF4 derived in Section 4.5 with the generalised response surfaces developed in this Chapter. This comparison is shown in Figure 6.9, where the agreement between numerical results obtained from the present response surfaces and the experimental data is similar to that observed for the numerical results of Section 4.5, confirming the overall validity of the DOE analysis performed in this Chapter. In particular the average relative deviations between numerical and experimental results on heat transfer and head losses are of $\pm 12.1\%$ and $\pm 10.3\%$, respectively. The mismatch between the two sets of numerical results of Figure 6.9 tends to increase with growing Δp^* and $\dot{Q}^*$, i.e. as the flow rate and Reynolds number increase. These discrepancies can be traced back to the deviations between the f and Nu values provided by the response surfaces and those obtained from the dedicated correlations developed for WF4 in Section 4.5, which tend to grow with increasing Re . This shortcoming stems from the limited data set used as a basis for the derivation of the response surfaces, which will be expanded during future research activities.

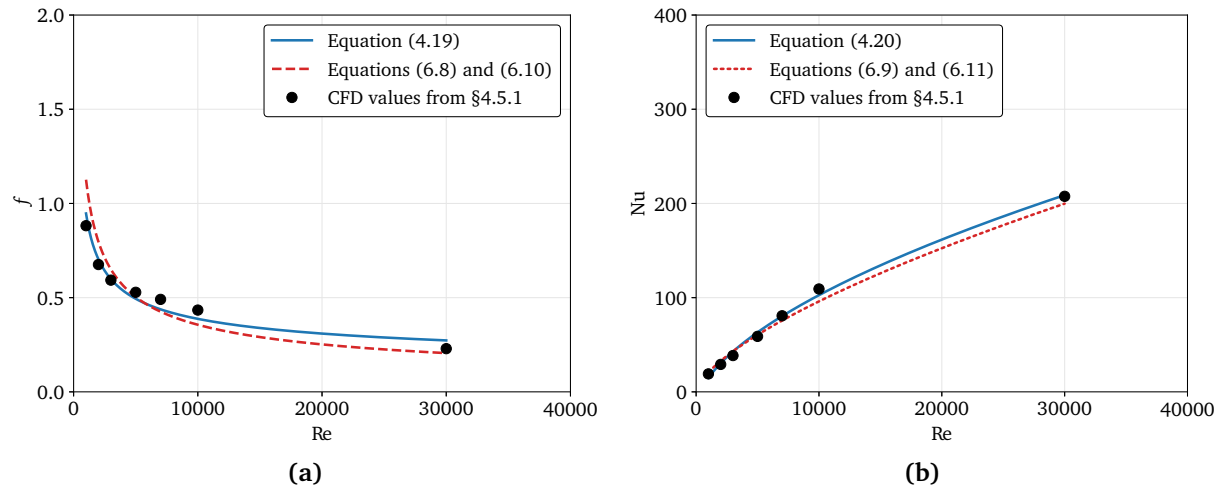


Figure 6.8. Comparison between the values of equivalent Darcy friction factor (a) and mean Nusselt number (b) obtained from the response surfaces derived in this Chapter with those obtained from the dedicated CFD model of WF4, which was developed in Chapter 4.

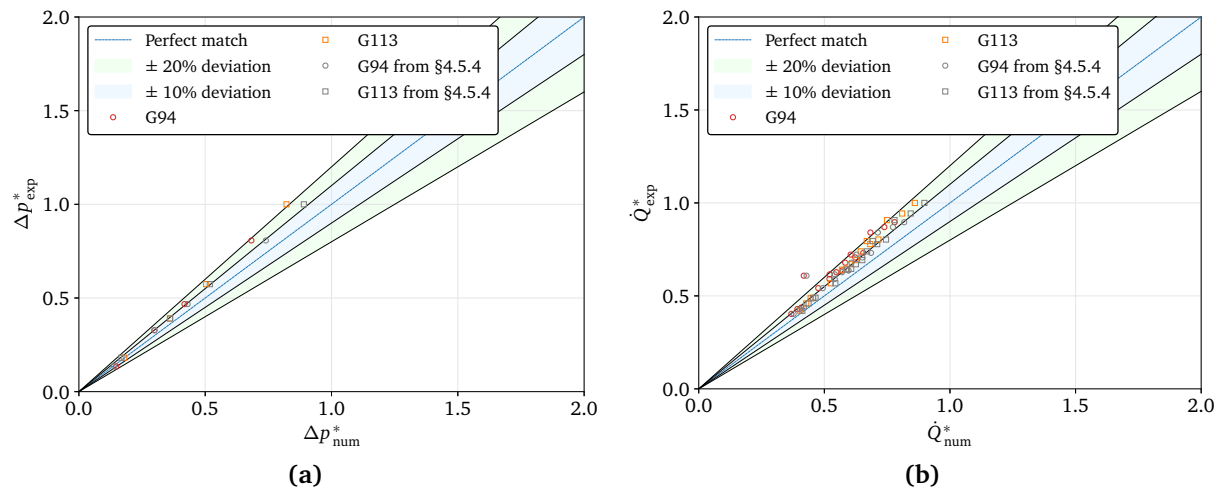


Figure 6.9. Comparison between the numerically obtained values of external head losses (b) and heat transfer rates (a) with the numerical end experimental results presented in Section 4.5.4. In these plots all numerical and experimental values are scaled by the maximum experimental entry.

6.7. Concluding remarks

This Chapter presented the first steps and preliminary results of a parametric analysis on the thermal-hydraulic performance of wavy or corrugated fins. The dependence of geometric parameters on the equivalent Darcy friction factor f and Colburn factor j was evaluated by means of a Design Of Experiment approach, which was carried out by using numerical methods. Each configuration was studied by developing a CFD model of a unitary fin module under periodic flow conditions.

Preliminary numerical results on heat transfer and head losses were employed for the derivation of basic response surfaces for f and j , where the dependency on the Reynolds number was modelled by using scaling laws available in the literature. Despite the limited data set used for the derivation of the response surfaces, these proved be quite accurate when their outcomes were compared with the numerical and experimental data obtained for single fins and complete heat exchangers presented in [Chapter 4](#).

Future research efforts will be aimed at expanding and completing the present parametric analysis by studying several other flow regimes. Additional numerical results will be integrated into the data set used for the derivation of the response surface in order to improve their accuracy.

Conclusions

This thesis presented the development of computational methods for the thermal analysis of powertrain and cooling components of sustainable vehicles. These approaches were discussed through the analysis of case studies relevant to the design and operation of electric vehicles.

Chapter 1 presented a comprehensive overview on electric and hybrid vehicles, along with a selection of relevant powertrain components as electric motors, power electronics systems, and compact heat exchangers. The operating principles and architectures of these components were described, along with the principal methods for heat extraction from the heat-generating parts.

Chapter 2 introduced a generalised methodology for the development and analysis of lumped parameter models, which are used extensively in engineering workflows, ranging from the analysis of simple components to detailed studies of complex physical systems. This approach was based on three main concepts: the use of the graph theory for the representation of the computational domain, an advanced system for the management of units of measurements, and a generalised linear formulation for source and sink terms. These core components were used as a base for the development of a solver for one-dimensional flows within networks of pipes and a solver for networks of thermal resistances. This toolbox was implemented in a Python 3 environment as an expansion package based on existing libraries implementing sparse linear algebra algorithms.

The solver developed for networks of thermal resistances was used in Chapter 3 to study the thermal management of an integrated motor drive composed of a six-phase synchronous motor and its power converter, which featured a switching cell array architecture. The integrated assembly was analysed by developing three lumped parameter thermal networks with varying degree of complexity and number of degrees of freedom. In these models, the losses generated in the motor were computed using numerical results from prior Finite

Element analyses, and considering temperature scaling laws available in the literature. The heat transfer coefficients associated with the cooling jacket were determined by developing a dedicated CFD model.

A comprehensive sensitivity analysis was performed to identify the most relevant parameters included in the thermal models, and to investigate the changes in thermal performance given by changing the constitutive materials of the integrated assembly. The most relevant parameters for the thermal management of the motor were the heat transfer coefficient associated with coolant flow, the thermal conductivity of the iron laminations, the interface gap between stator and case, and the heat transfer coefficients in the air gap. In case of the power converter the most relevant parameters to the thermal performance were the heat transfer coefficient associated with coolant flow, the thermal conductivity of the thermal pad between the cold plates and the switches, and the junction thermal resistance. Furthermore, this analysis highlighted possible improvements to the thermal performance achievable through the use of insulating enamels and resins with higher thermal conductivities. In addition, the developed thermal models were used to compute hot-spot temperatures in the most relevant components of the integrated assembly, to assess the feasibility of the proposed architecture from the thermal point of view.

Chapter 4 was focused on the thermal analysis of compact heat exchangers, whose effectiveness and energy efficiency play a critical role in the overall performance and range of electric vehicles. A multiscale approach, based on detailed CFD models for the extended surface and manifolds and reduced-order models for the evaluation of the overall performance of the heat exchangers, was presented through the analysis of a case study including several configurations of plate-fin heat exchangers. Numerical results on extended surfaces were compared with data available in the literature, underlining the need to develop dedicated numerical models to obtain accurate results for fin configurations used in actual industrial applications. The good agreement between numerical and experimental results on the overall thermal-hydraulic performance of the heat exchangers corroborated the validity of the presented approach. In addition, numerical results on internal head losses underscored the importance of considering the contributions given by the inlet and outlet manifolds, which are dominant for flows at high Reynolds numbers.

The study of compact heat exchangers was expanded in Chapters 5 and 6 by performing a parametric analysis on the thermal-hydraulic characteristics of offset-strip fins and wavy extended surfaces, which were carried out by mean of a Design Of Experiment procedure based on numerical techniques. Numerical results were used to derive continuous multivariate response surfaces with clearly defined applicability ranges, which are suitable to performed optimisation procedures. These included a novel modelling scheme for the Prandtl number scaling coefficient of offset-strip fins. The obtained response surfaces were validated against the experimental data presented in Chapter 4.

In future research activities, the computational toolbox developed in Chapter 2 will be expanded with the addition of generalised multi-physics solver and part of the source code will be compiled to reduce the computational effort required for the solution of large models. The methodology developed in Chapter 3 for the thermal analysis of electric motors and their power converters will be employed for the study of other relevant powertrain components of electric and hybrid vehicles as battery packs, and DC/DC power converters.

In the future, the approach presented in Chapter 4 for compact heat exchangers will be employed in conjunction with the response surfaces derived in Chapters 5 and 6 to carry out optimisation procedures on complete heat exchangers with respect to their thermal-hydraulic performance, volume, and cost.

Acknowledgements

PhD thesis co-funded within the framework of the National Operational Programme Research and Innovation 2014-2020 (CCI 2014IT16M2OP005), ESF REACT-EU resources, Action IV.4 "Doctorates and research contracts on innovation topics" and Action IV.5 "Doctorates on Green topics".

The study presented in Chapter 3 received additional funding by the European Union's Horizon Europe research and innovation program under grant agreement No. 101056781. Views and opinions expressed are however those of the author only and do not necessarily reflect those of the European Union or CINEA. Neither the European Union nor the granting authority can be held responsible for them.

APPENDIX A

Numerical methods

A.1. Upwind differencing

The upwind differencing scheme is employed to transfer node fields to branches: the values of pressure, density, temperature, and specific heat related to branches are obtained from their respective upwind node.

Let us consider grid points spaced apart by Δx and a generic regular function φ . The value of φ at the point $x + \Delta x$ can be evaluated through a Taylor expansion:

$$\varphi(x + \Delta x) = \varphi(x) + \frac{d}{dx} \varphi(x) \Delta x + \frac{1}{2} \frac{d^2}{dx^2} \varphi(x) \Delta x^2 + \dots \quad (\text{A.1})$$

The truncation error that arises by assuming $\varphi(x + \Delta x) = \varphi(x)$ is:

$$E = \frac{d}{dx} \varphi(x) \Delta x + \frac{1}{2} \frac{d^2}{dx^2} \varphi(x) \Delta x^2 + \dots = o(\Delta x), \quad (\text{A.2})$$

meaning that upwind differencing is a first order accurate numerical method.

A.2. Midpoint rule

The midpoint rule is the simplest second order method for approximating definite integrals, it is also valid for surface and volume integrals. Let us begin by considering a generic definite integral:

$$\int_a^b f(x) dx, \quad (\text{A.3})$$

then let c be the midpoint of the interval of integration: $c = a + b/2$. The Taylor expansion of $f(x)$ around c reads:

$$f(x) = f(c) + \frac{d}{dx} f(c)(x - c) + \frac{1}{2} \frac{d^2}{dx^2} f(c)(x - c)^2 + \dots \quad (\text{A.4})$$

The midpoint rule is obtained by introducing Equation (A.4) into Equation (A.3) and solving the integrals:

$$\int_a^b f(x) dx = f(c)(b - a) + \frac{1}{24} \frac{d^2}{dx^2} f(c)(b - a)^3 + \dots \quad (\text{A.5})$$

The discretisation error is given by:

$$E = \frac{1}{b - a} \int_a^b f(x) dx - f(c) = \frac{1}{24} \frac{d^2}{dx^2} f(c)(b - a)^2, \quad (\text{A.6})$$

which means that the difference between the integral average of a function and its midpoint value is a second order term.

Equation (A.5) can be used to approximate surface and volume integrals, assuming the integral average of the considered function equal to its value sampled at the surface or volume centre point.

A.3. Implicit Euler method

Let us consider the following Cauchy problem [288]:

$$\begin{cases} \frac{d\phi(t)}{dt} = f(\phi(t), t) & \text{for } t > t_0 \\ \phi(t_0) = \phi_0 \end{cases} \quad (\text{A.7})$$

Equation (A.7) is solved between two time steps: t^n and t^{n+1} spaced apart by a time interval Δt . Equation (A.7) written for the time step t^{n+1} reads:

$$\frac{d\phi(t^{n+1})}{dt} = f(\phi(t^{n+1}), t^{n+1}) \quad (\text{A.8})$$

the time derivative on the left side of Equation (A.8) is evaluated by using a backwards Taylor expansion:

$$\frac{d\phi(t^{n+1})}{dt} = \frac{\phi(t^{n+1}) - \phi(t^n)}{\Delta t} + o(\Delta t). \quad (\text{A.9})$$

Combining Equations (A.8) and (A.9), and rearranging the terms leads to:

$$\phi(t^{n+1}) = \phi(t^n) + f(\phi(t^{n+1}), t^{n+1}) \Delta t + o(\Delta t^2). \quad (\text{A.10})$$

Let ϕ^{n+1} be the approximated value of $\phi(t^{n+1})$, obtained by discarding the term $o(\Delta t^2)$. The same reasoning can be applied at the previous time step, leading to $\phi^n = \phi(t^n)$. By introducing the previous considerations in Equation (A.10) the implicit Euler method is finally obtained:

$$\phi^{n+1} = \phi^n + f(\phi^{n+1}, t^{n+1}) \Delta t. \quad (\text{A.11})$$

The local truncation error at time step $n + 1$ is defined by:

$$E^{n+1} = \frac{\phi(t^{n+1}) - \phi(t^n)}{\Delta t} - f(\phi(t^{n+1}), t^{n+1}). \quad (\text{A.12})$$

By considering the Taylor expansion of the first right-hand side term of Equation (A.12) and by using Equation (A.8) the error becomes:

$$E^{n+1} = \frac{d}{dt} \phi(t^{n+1}) - \frac{\Delta t}{2} \frac{d^2}{dt^2} \phi(t^{n+1}) - \frac{d}{dt} \phi(t^{n+1}) = -\frac{\Delta t}{2} \frac{d^2}{dt^2} \phi(t^{n+1}). \quad (\text{A.13})$$

Finally, the modulus of error is:

$$|E^{n+1}| \leq \frac{\Delta t}{2} \max \left(\frac{d^2}{dt^2} \phi(t) \right) \quad \text{for } t_0 \leq t \leq t_F. \quad (\text{A.14})$$

From Equation (A.14) it is clear that the implicit Euler method is of first order accuracy.

APPENDIX B

Offset-strip fins

B.1. Implementation of the non-dimensional framework

```

/*-----*- C++ -*-----*\
| ===== |
| \ \      / F ield      | OpenFOAM: The Open Source CFD Toolbox |
| \ \      / O peration  | Version: 7.0 |
| \ \      / A nd        | Web: www.OpenFOAM.org |
| \ \      / M anipulation |
| \ \      /              |
\*-----*-*/
FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    location     "constant/fluid-cells";
    object       thermophysicalProperties;
}
// * * * * *

#include "../initialConditions"

thermoType
{
    type          heRhoThermo;
    mixture        pureMixture;
    transport      const;
    thermo         hConst;
    equationOfState rhoConst;
    specie         specie;
    energy         sensibleEnthalpy;
}

mixture
{
    specie
    {
        nMoles      1;  // placeholder
        molWeight    1;  // placeholder
    }
    thermodynamics
    {
        Cp          #calc "$Re*$Pr";
        Hf          0;
    }
    transport
    {
        mu          #calc "1.0/$Re";
        Pr          $Pr;
    }
    equationOfState
    {
        rho         1;
    }
}

// * * * * *

```

Bibliography

- [1] E.E. Agency. Greenhouse gas emissions from transport in europe. *Technical Report, Europe Environment Agency*, 2023.
- [2] E.E. Agency. Greenhouse gas emissions from transport in europe. *Technical Report, Europe Environment Agency*, 2018.
- [3] T.F. Pignolet. A policy framework for climate and energy in the period from 2020 up to 2030: Impact analysis, 2015.
- [4] J.R. IPCC. *Global warming of 1.5° C: IPCC special report on impacts of global warming of 1.5° C above pre-industrial levels in context of strengthening response to climate change, sustainable development, and efforts to eradicate poverty*. Cambridge University Press Cambridge, UK, 2022.
- [5] K. Shahzad and I. Cheema. Low-carbon technologies in automotive industry and decarbonizing transport. *Journal of Power Sources*, 591:233888, 2024.
- [6] H. Shi, J. Chai, Q. Lu, J. Zheng, and S. Wang. The impact of china's low-carbon transition on economy, society and energy in 2030 based on co2 emissions drivers. *Energy*, 239:122336, 2022.
- [7] A. Albatayneh, M. Assaf, D. Alterman, and M. Jaradat. Comparison of the overall energy efficiency for internal combustion engine vehicles and electric vehicles. *Rigas Tehniskas Universitates Zinatniskie Raksti*, 24(1):669–680, 2020.
- [8] K. Poornesh, K.P. Nivya, and K. Sireesha. A comparative study on electric vehicle and internal combustion engine vehicles. In *2020 International Conference on Smart Electronics and Communication (ICOSEC)*, pages 1179–1183, 2020.

- [9] Range of full electric vehicles - electric vehicle database. <https://ev-database.org/cheatsheet/range-electric-car>. Accessed: 2024-12-14.
- [10] M. Ehsani, Y. Gao, S. Longo, and K. Ebrahimi. *Modern electric, hybrid electric, and fuel cell vehicles*. CRC press, 2018.
- [11] K.C. Prajapati, R. Patel, and R. Sagar. Hybrid vehicle: A study on technology. *Power (kW)*, 2:10–20, 2014.
- [12] M. Barcaro, N. Bianchi, and F. Magnussen. Pm motors for hybrid electric vehicles. In *2008 43rd International Universities Power Engineering Conference*, pages 1–5, 2008.
- [13] J.A. Sanguesa, V. Torres-Sanz, P. Garrido, F.J. Martinez, and J.M. Marquez-Barja. A review on electric vehicles: Technologies and challenges. *Smart Cities*, 4(1):372–404, 2021.
- [14] P. Nikolaidis and A. Poullikkas. A comparative overview of hydrogen production processes. *Renewable and sustainable energy reviews*, 67:597–611, 2017.
- [15] M.S. Patil and S. Dhamal. A detailed motor selection for electric vehicle traction system. In *2019 Third International conference on I-SMAC (IoT in Social, Mobile, Analytics and Cloud) (I-SMAC)*, pages 679–684, 2019.
- [16] A. Hughes. *Electric Motors and Drives*. Elsevier, 3rd Edition - December 14, 2005.
- [17] A. El-Refaie. Motors/generators for traction/propulsion applications: A review. *IEEE Vehicular Technology Magazine*, 8(1):90–99, 2013.
- [18] H. El Hadraoui, M. Zegrari, A. Chebak, O. Laayati, and N. Guennouni. A multi-criteria analysis and trends of electric motors for electric vehicles. *World Electric Vehicle Journal*, 13(4):65, 2022.
- [19] S. Rind, Y. Ren, Y. Hu, J. Wang, and L. Jiang. Configurations and control of traction motors for electric vehicles: A review. *Chinese Journal of Electrical Engineering*, 3(3):1–17, 2017.
- [20] P. Famouri and W. Cooley. Design of dc traction motor drives for high efficiency under accelerating conditions. *IEEE Transactions on Industry Applications*, 30(4):1134–1138, 1994.
- [21] C.S. Goli, M. Manjrekar, S. Essakiappan, P. Sahu, and N. Shah. Landscaping and review of traction motors for electric vehicle applications. In *2021 IEEE Transportation Electrification Conference & Expo (ITEC)*, pages 162–168, 2021.

- [22] A. Krings and C. Monissen. Review and trends in electric traction motors for battery electric and hybrid vehicles. In *2020 International Conference on Electrical Machines (ICEM)*, volume 1, pages 1807–1813, 2020.
- [23] J.W. Ahn. Switched reluctance motor. *Torque control*, pages 201–252, 2011.
- [24] Y. Yang et al. Thermal management of electric machines. *IET Electrical Systems in Transportation*, 7(2):104–116, 2017.
- [25] S. Nalakath, M. Preindl, B. Bilgin, B. Cheng, and A. Emadi. Modeling and analysis of ac resistance of a permanent magnet machine for online estimation purposes. In *2015 IEEE Energy Conversion Congress and Exposition (ECCE)*, pages 314–319, 2015.
- [26] M. Pastura, R. Notari, S. Nuzzo, D. Barater, and G. Franceschini. On the ac losses in the end conductors of hairpin windings. In *2022 International Conference on Electrical Machines (ICEM)*, pages 1150–1155, 2022.
- [27] M. Pastura, R. Notari, S. Nuzzo, D. Barater, and G. Franceschini. Ac losses analysis and design guidelines for hairpin windings with segmented conductors. *IEEE Transactions on Transportation Electrification*, 10(1):33–41, 2024.
- [28] D. Deisenroth and M. Ohadi. Thermal management of high-power density electric motors for electrification of aviation and beyond. *Energies*, 12(19), 2019.
- [29] A. Ranjbar and B. Fahimi. Ac machines: Permanent magnet synchronous and induction machines. In *Electric, Hybrid, and Fuel Cell Vehicles*, pages 451–481. Springer, 2021.
- [30] D. Konovalov, I. Tolstorebrov, T.M. Eikevik, H. Kobalava, M. Radchenko, A. Hafner, and A. Radchenko. Recent developments in cooling systems and cooling management for electric motors. *Energies*, 16(19):7006, 2023.
- [31] D. Staton, A. Boglietti, and A. Cavagnino. Solving the more difficult aspects of electric motor thermal analysis in small and medium size industrial induction motors. *IEEE Transactions on Energy Conversion*, 20(3):620–628, 2005.
- [32] A. Anton, K.W. Lienert, and G. Hegemann. Wire enamels—an application for high performance polymers unknown to chemists. *Macromolecular Materials and Engineering*, 293(5):331–339, 2008.
- [33] A. Uchechukwu, E. Anthony, and E. Godspower. Electrical properties of enamel wire insulation. *International Journal of Trend in Scientific Research and Development*, 3(2):803–806, 2019.

- [34] F. Guastavino, A. Ratto, S. Squarcia, and E. Torello. Electro-thermal aging tests on different kinds of enamelled wires. In *2010 10th IEEE International Conference on Solid Dielectrics*, pages 1–4, 2010.
- [35] S. Nategh, A. Boglietti, Y. Liu, D. Barber, R. Brammer, D. Lindberg, and O. Aglen. A review on different aspects of traction motor design for railway applications. *IEEE Transactions on Industry Applications*, 56(3):2148–2157, 2020.
- [36] Y. Sun, S. Zhang, W. Yuan, Y. Tang, J. Li, and K. Tang. Applicability study of the potting material based thermal management strategy for permanent magnet synchronous motors. *Applied Thermal Engineering*, 149:1370–1378, 2019.
- [37] Y. Yao, L. Gu, T. Fan, W. Sun, and J. Luo. Evaluation of heat transfer characteristic of aluminum nitride (aln) potting compound for the end windings of permanent magnet synchronous machines. In *2011 International Conference on Electrical and Control Engineering*, pages 4498–4501, 2011.
- [38] M. Polikarpova, P. Lindh, C. Gerada, M. Rilla, V. Naumanen, and J. Pyrhönen. Thermal effects of stator potting in an axial-flux permanent magnet synchronous generator. *Applied Thermal Engineering*, 75:421–429, 2015.
- [39] V. Madonna, P. Giangrande, L. Lusuardi, A. Cavallini, and M. Galea. Impact of thermal overload on the insulation aging in short duty cycle motors for aerospace. In *2018 IEEE International Conference on Electrical Systems for Aircraft, Railway, Ship Propulsion and Road Vehicles & International Transportation Electrification Conference (ESARS-ITEC)*, pages 1–6, 2018.
- [40] A. Ayachit and M. Kazimierczuk. Thermal effects on inductor winding resistance at high frequencies. *IEEE Magnetics Letters*, 4:0500304–0500304, 2013.
- [41] M. Grabowski, K. Urbaniec, J. Wernik, and K.J. Wołosz. Numerical simulation and experimental verification of heat transfer from a finned housing of an electric motor. *Energy Conversion and Management*, 125:91–96, 2016. Sustainable development of energy, water and environment systems for future energy technologies and concepts.
- [42] M. Jaeger, A. Ruf, K. Hameyer, and T.G.v. Tongeln. Thermal analysis of an electrical traction motor with an air cooled rotor. In *2018 IEEE Transportation Electrification Conference and Expo (ITEC)*, pages 467–470, 2018.
- [43] A.M. Teamah and M.S. Hamed. Cooling of a synchronous electric motor using an axial air flow. *International Journal of Thermofluids*, 20:100525, 2023.

- [44] S. Ulbrich, J. Koppe, and J. Proske. Cooling fin optimization on a tefc electrical machine housing using a 2-d conjugate heat transfer model. *IEEE Transactions on Industrial Electronics*, 65(2):1711 – 1718, 2018. Cited by: 56.
- [45] H.S. Peng and F.H. Lai. Investigation of parameters affecting heat transfer and fluid flow of a tefc electric motor by using taguchi method. In *IOP Conference Series: Materials Science and Engineering*, volume 491, page 012021. IOP Publishing, 2019.
- [46] J. Kuria and P. Hwang. Investigation of thermal performance of electric vehicle bldc motor. *International journal of mechanical engineering*, 1(1):1–17, 2012.
- [47] T. Jercic, D. Zarko, M. Martinovic, M. Kovacic, J. Juric, Z. Hanic, and S. Stipetic. Centrifugal fan design for permanent magnet synchronous motor in a traction application. In *2017 IEEE International Electric Machines and Drives Conference (IEMDC)*, pages 1–7, 2017.
- [48] M. Roffi, F.J.T.E. Ferreira, and A.T. De Almeida. Comparison of different cooling fan designs for electric motors. In *2017 IEEE International Electric Machines and Drives Conference (IEMDC)*, pages 1–7, 2017.
- [49] Y.C. Chen, C.L. Chen, and Q. Dong. Cfd modeling for motor fan system. In *IEEE International Electric Machines and Drives Conference, 2003. IEMDC'03.*, volume 2, pages 764–768 vol.2, 2003.
- [50] C.C. Chang, Y.F. Kuo, J.C. Wang, and S.L. Chen. Air cooling for a large-scale motor. *Applied Thermal Engineering*, 30(11):1360–1368, 2010.
- [51] M.S. Kim, K.S. Lee, and S. Um. Numerical investigation and optimization of the thermal performance of a brushless dc motor. *International Journal of Heat and Mass Transfer*, 52(5):1589–1599, 2009.
- [52] E. Galloni, P. Parisi, F. Marignetti, and G. Volpe. Cfd analyses of a radial fan for electric motor cooling. *Thermal Science and Engineering Progress*, 8:470–476, 2018.
- [53] M. Satrústegui, M. Martinez-Iturralde, J.C. Ramos, P. Gonzalez, G. Astarbe, and I. Elosegui. Design criteria for water cooled systems of induction machines. *Applied Thermal Engineering*, 114:1018–1028, 2017.
- [54] A.J. Bourgault, P. Roy, E. Ghosh, and N.C. Kar. A survey of different cooling methods for traction motor application. In *2019 IEEE Canadian Conference of Electrical and Computer Engineering (CCECE)*, pages 1–4, 2019.
- [55] C. Yang, H. Wang, X. Niu, J. Zhang, and Y. Yan. Design and analysis of cycling oil cooling in driving motors for electric vehicle application. In *2016 IEEE Vehicle Power and Propulsion Conference (VPPC)*, pages 1–6, 2016.

- [56] P.S. Wu, M.F. Hsieh, W.L. Cai, J.H. Liu, Y.T. Huang, J.F. Caceres, and S.W. Chang. Heat transfer and thermal management of interior permanent magnet synchronous electric motor. *Inventions*, 4(4):69, 2019.
- [57] Z. Rehman and K. Seong. Three-d numerical thermal analysis of electric motor with cooling jacket. *Energies*, 11(1):92, 2018.
- [58] F. Incropera, D. DeWitt, T. Bergman, A. Lavine, et al. *Fundamentals of heat and mass transfer*, volume 6. Wiley New York, 1996.
- [59] Z. Huang, S. Nategh, V. Lassila, M. Alaküla, and J. Yuan. Direct oil cooling of traction motors in hybrid drives. In *2012 IEEE International Electric Vehicle Conference*, pages 1–8, 2012.
- [60] J. Li, Y. Lu, Y.H. Cho, and R. Qu. Design, analysis, and prototyping of a water-cooled axial-flux permanent-magnet machine for large-power direct-driven applications. *IEEE Transactions on Industry Applications*, 55(4):3555–3565, 2019.
- [61] A. Boglietti, A. Cavagnino, and D. Staton. Tefc induction motors thermal models: a parameter sensitivity analysis. In *Conference Record of the 2004 IEEE Industry Applications Conference, 2004. 39th IAS Annual Meeting.*, volume 4, pages 2469–2476 vol.4, 2004.
- [62] K. Bennion and J. Cousineau. Sensitivity analysis of traction drive motor cooling. In *2012 IEEE Transportation Electrification Conference and Expo (ITEC)*, pages 1–6, 2012.
- [63] C. Wohlers, P. Juris, S. Kabelac, and B. Ponick. Design and direct liquid cooling of tooth-coil windings. *Electrical Engineering*, 100:2299–2308, 2018.
- [64] M. Schiefer and M. Doppelbauer. Indirect slot cooling for high-power-density machines with concentrated winding. In *2015 IEEE International Electric Machines & Drives Conference (IEMDC)*, pages 1820–1825, 2015.
- [65] V. Madonna, P. Giangrande, A. Walker, and M. Galea. On the effects of advanced end-winding cooling on the design and performance of electrical machines. In *2018 XIII International Conference on Electrical Machines (ICEM)*, pages 311–317, 2018.
- [66] V. Madonna, A. Walker, P. Giangrande, G. Serra, C. Gerada, and M. Galea. Improved thermal management and analysis for stator end-windings of electrical machines. *IEEE Transactions on Industrial Electronics*, 66(7):5057–5069, 2019.
- [67] F. Marcolini, G.D. Donato, and F. Caricchi. Direct oil cooling of end-windings in torus-type axial-flux permanent-magnet machines. In *2019 IEEE Energy Conversion Congress and Exposition (ECCE)*, pages 5645–5651, 2019.

- [68] P.M. Lindh, I. Petrov, R.S. Semken, M. Niemelä, J.J. Pyrhönen, L. Aarniovuori, T. Vaimann, and A. Kallaste. Direct liquid cooling in low-power electrical machines: Proof-of-concept. *IEEE Transactions on Energy Conversion*, 31(4):1257–1266, 2016.
- [69] P. Lindh, I. Petrov, J. Pyrhönen, E. Scherman, M. Niemelä, and P. Immonen. Direct liquid cooling method verified with a permanent-magnet traction motor in a bus. *IEEE Transactions on Industry Applications*, 55(4):4183–4191, 2019.
- [70] Y. Alexandrova, R.S. Semken, and J. Pyrhönen. Permanent magnet synchronous generator design solution for large direct-drive wind turbines: Thermal behavior of the lc dd-pmsg. *Applied Thermal Engineering*, 65(1):554–563, 2014.
- [71] X. Wang, B. Li, D. Gerada, K. Huang, I. Stone, S. Worrall, and Y. Yan. A critical review on thermal management technologies for motors in electric cars. *Applied Thermal Engineering*, 201:117758, 2022.
- [72] X. Chen, J. Wang, A. Griffo, and A. Spagnolo. Thermal modeling of hollow conductors for direct cooling of electrical machines. *IEEE Transactions on Industrial Electronics*, 67(2):895–905, 2020.
- [73] C. Liu, Z. Xu, D. Gerada, J. Li, C. Gerada, Y.C. Chong, M. Popescu, J. Goss, D. Staton, and H. Zhang. Experimental investigation on oil spray cooling with hairpin windings. *IEEE Transactions on Industrial Electronics*, 67(9):7343–7353, 2020.
- [74] R.D. Plant, J. Friedman, and M.Z. Saghir. A review of jet impingement cooling. *International Journal of Thermofluids*, 17:100312, 2023.
- [75] L. Ye, F. Tao, L. Qi, and W. Xuhui. Experimental investigation on heat transfer of directly-oil-cooled permanent magnet motor. In *2016 19th International Conference on Electrical Machines and Systems (ICEMS)*, pages 1–4, 2016.
- [76] F. Zhang, D. Gerada, Z. Xu, C. Liu, H. Zhang, T. Zou, Y.C. Chong, and C. Gerada. A thermal modeling approach and experimental validation for an oil spray-cooled hairpin winding machine. *IEEE Transactions on Transportation Electrification*, 7(4):2914–2926, 2021.
- [77] T. Davin, J. Pellé, S. Harmand, and R. Yu. Experimental study of oil cooling systems for electric motors. *Applied Thermal Engineering*, 75:1–13, 2015.
- [78] H. Chuan, R. Burke, and Z. Wu. A comparative study on different cooling topologies for axial flux permanent magnet machine. In *2019 IEEE Vehicle Power and Propulsion Conference (VPPC)*, pages 1–6, 2019.

- [79] A.M. EL-Refaie, J.P. Alexander, S. Galimoto, P.B. Reddy, K.K. Huh, P. de Bock, and X. Shen. Advanced high-power-density interior permanent magnet motor for traction applications. *IEEE Transactions on Industry Applications*, 50(5):3235–3248, 2014.
- [80] Y. Gai, M. Kimiabeigi, J.D. Widmer, Y.C. Chong, J. Goss, U. SanAndres, and D.A. Statton. Shaft cooling and the influence on the electromagnetic performance of traction motors. In *2017 IEEE International Electric Machines and Drives Conference (IEMDC)*, pages 1–6, 2017.
- [81] A.S. Sudhan, K. Ramachandran, A.B. Solomon, and C. Jawahar. Research progress on performance enhancement of heat pipes: a review. *Journal of Thermal Analysis and Calorimetry*, pages 1–37, 2021.
- [82] A. Faghri. *Heat pipe science and technology*. Taylor & Francis, 1995.
- [83] M. Groll, H. Krahling, and W. Munzel. Heat pipes for cooling of an electric motor. *Journal of Energy*, 2(6):363–367, 1978.
- [84] N. Putra and B. Ariantara. Electric motor thermal management system using l-shaped flat heat pipes. *Applied Thermal Engineering*, 126:1156–1163, 2017.
- [85] G. Fang, W. Yuan, Z. Yan, Y. Sun, and Y. Tang. Thermal management integrated with three-dimensional heat pipes for air-cooled permanent magnet synchronous motor. *Applied Thermal Engineering*, 152:594–604, 2019.
- [86] M. Galea, C. Gerada, T. Raminosa, and P. Wheeler. A thermal improvement technique for the phase windings of electrical machines. *IEEE Transactions on Industry Applications*, 48(1):79–87, 2012.
- [87] J.X. Wang, Y.Z. Li, S.N. Wang, H.S. Zhang, X. Ning, and W. Guo. Experimental investigation of the thermal control effects of phase change material based packaging strategy for on-board permanent magnet synchronous motors. *Energy Conversion and Management*, 123:232–242, 2016.
- [88] S. Wang, Y.Z. Li, Y. Liu, H. Zhou, Y. Li, W. Guo, and X. Xiao. Temperature control of permanent-magnet synchronous motor using phase change material. In *2015 IEEE International Conference on Advanced Intelligent Mechatronics (AIM)*, pages 1635–1640, 2015.
- [89] S. Wang, Y. Li, Y.Z. Li, J. Wang, X. Xiao, and W. Guo. Transient cooling effect analyses for a permanent-magnet synchronous motor with phase-change-material packaging. *Applied Thermal Engineering*, 109:251–260, 2016.

- [90] S. Ayat, C. Serghine, T. Klonowski, S. Yon, A. Mutabazi, and S. McDaniel. The use of phase change material for the cooling of electric machine windings formed with hollow conductors. In *2019 IEEE International Electric Machines & Drives Conference (IEMDC)*, pages 1195–1201, 2019.
- [91] R. Rafiezadeh, S. Busquets-Monge, and S. Alepuz. Optimized modular design of neutral-point-clamped traction inverters for multiple electric vehicles. In *2023 AEIT International Conference on Electrical and Electronic Technologies for Automotive (AEIT AUTOMOTIVE)*, pages 1–6, 2023.
- [92] The authoritative dictionary of iee standards terms, seventh edition, 2000.
- [93] M.H. Rashid. *Power electronics handbook*. Butterworth-heinemann, 2017.
- [94] S. Busquets-Monge and L. Caballero. Switching-cell arrays—an alternative design approach in power conversion. *IEEE Transactions on Industrial Electronics*, 66(1):25–36, 2019.
- [95] J. Reimers, L. Dorn-Gomba, C. Mak, and A. Emadi. Automotive traction inverters: Current status and future trends. *IEEE Transactions on Vehicular Technology*, 68(4):3337–3350, 2019.
- [96] X. Li, L. Zhang, S. Guo, Y. Lei, A.Q. Huang, and B. Zhang. Understanding switching losses in sic mosfet: Toward lossless switching. In *2015 IEEE 3rd Workshop on Wide Bandgap Power Devices and Applications (WiPDA)*, pages 257–262, 2015.
- [97] W.T. Franke and F.W. Fuchs. Comparison of switching and conducting performance of sic-jfet and sic-bjt with a state of the art igbt. In *2009 13th European Conference on Power Electronics and Applications*, pages 1–10, 2009.
- [98] K. Suzuki, K. Yuki, and M. Mochizuki. Application of boiling heat transfer to high-heat-flux cooling technology in power electronics. *Transactions of The Japan Institute of Electronics Packaging*, 4(1):127–133, 2011.
- [99] M. Pedram and S. Nazarian. Thermal modeling, analysis, and management in vlsi circuits: Principles and methods. *Proceedings of the IEEE*, 94(8):1487–1501, 2006.
- [100] J. Black. Electromigration—a brief survey and some recent results. *IEEE Transactions on Electron Devices*, 16(4):338–347, 1969.
- [101] M. Anwar, M. Hayes, A. Tata, M. Teimorzadeh, and T. Achatz. Power dense and robust traction power inverter for the second-generation chevrolet volt extended-range ev. *SAE International Journal of Alternative Powertrains*, 4(1), 2015.

- [102] M. Anwar, S.N. Hasan, M. Teimor, M. Korich, and M.B. Hayes. Development of a power dense and environmentally robust traction power inverter for the second-generation chevrolet volt extended-range ev. In *2015 IEEE Energy Conversion Congress and Exposition (ECCE)*, pages 6006–6013. IEEE, 2015.
- [103] M. Anwar, M. Teimor, P. Savagian, R. Saito, and T. Matsuo. Compact and high power inverter for the cadillac ct6 rear wheel drive phev. In *2016 IEEE Energy Conversion Congress and Exposition (ECCE)*, pages 1–7, 2016.
- [104] A. Khan, T. Grewe, J. Liu, M. Anwar, A. Holmes, and R. Balsley. The gm rwd phev propulsion system for the cadillac ct6 luxury sedan. Technical report, SAE Technical Paper, 2016.
- [105] J. Reimers, L. Dorn-Gomba, C. Mak, and A. Emadi. Automotive traction inverters: Current status and future trends. *IEEE Transactions on Vehicular Technology*, 68(4):3337–3350, 2019.
- [106] R.J. Ramm, W. Liu, , and C. Campbell. Heatsink with internal cavity for liquid cooling, U.S. Patent US14286670, 2014.
- [107] N. Nozawa, T. Maekawa, S. Nozawa, and K. Asakura. Development of power control unit for compact-class vehicle. *SAE International Journal of Passenger Cars-Electronic and Electrical Systems*, 2(2009-01-1310):376–382, 2009.
- [108] O. Kitazawa, T. Kikuchi, M. Nakashima, Y. Tomita, H. Kosugi, and T. Kaneko. Development of power control unit for compact-class vehicle. *SAE International Journal of Alternative Powertrains*, 5(2):278–285, 2016.
- [109] S. Nozawa, T. Maekawa, E. Yagi, Y. Terao, and H. Kohno. Development of new power control unit for compact-class vehicle. In *2010 22nd International Symposium on Power Semiconductor Devices & IC's (ISPSD)*, pages 43–45, 2010.
- [110] G. Moreno, K. Bennion, C. King, and S. Narumanchi. Evaluation of performance and opportunities for improvements in automotive power electronics systems. In *2016 15th IEEE Intersociety Conference on Thermal and Thermomechanical Phenomena in Electronic Systems (ITherm)*, pages 185–192. IEEE, 2016.
- [111] Y. Sato, S. Ishikawa, T. Okubo, M. Abe, and K. Tamai. Development of high response motor and inverter system for the nissan leaf electric vehicle. Technical report, SAE Technical Paper, 2011.
- [112] M. Ehsani, Y. Gao, S. Gay, and A. Emadi. *Modern Electric, Hybrid Electric, and Fuel Cell Vehicles: Fundamentals, Theory and Design*. CRC Press, 2005.

- [113] K. Liang, M. Wang, C. Gao, B. Dong, C. Feng, X. Zhou, and J. Liu. Advances and challenges of integrated thermal management technologies for pure electric vehicles. *Sustainable Energy Technologies and Assessments*, 46:101319, 2021.
- [114] D. Howey, R. Martinez-Botas, B. Cussons, and L. Lytton. Comparative measurements of the energy consumption of 51 electric, hybrid and internal combustion engine vehicles. *Transportation Research Part D: Transport and Environment*, 16(6):459–464, 2011.
- [115] G. Leoncini, R. Mothier, B. Michel, and M. Clausse. A review on challenges concerning thermal management system design for medium duty electric vehicles. *Applied Thermal Engineering*, 236:121464, 2024.
- [116] Y. Park, J. Kim, J. Oh, U. Han, and H. Lee. Multi-objective optimization of an offset strip fin heat exchanger for waste heat recovery in electric vehicles. *Applied Thermal Engineering*, 228:120533, 2023.
- [117] V. Pomme. Reversible heat pump system for an electrical vehicle. Technical report, SAE Technical Paper, 1997.
- [118] T. Wang, W. Luan, W. Wang, and S.T. Tu. Waste heat recovery through plate heat exchanger based thermoelectric generator system. *Applied Energy*, 136:860–865, 2014.
- [119] W.M. Rohsenow. Heat exchangers – theory and practice, 1984.
- [120] X. Zheng and Z. Qi. A comprehensive review of offset strip fin and its applications. *Applied Thermal Engineering*, 139:61–75, 2018.
- [121] J. Dong, J. Chen, W. Zhang, and J. Hu. Experimental and numerical investigation of thermal -hydraulic performance in wavy fin-and-flat tube heat exchangers. *Applied Thermal Engineering*, 30(11):1377–1386, 2010.
- [122] S. Documentation. Simulation and model-based design, 2020.
- [123] GT-SUITE. Integrated multi-physics systems simulation.
- [124] G. Ramchandran and J. Harrison. A thermodynamic chamber modelling approach for oil free and oil injected twin screw compressors. In *IOP Conference Series: Materials Science and Engineering*, volume 1180, page 012002. IOP Publishing, 2021.
- [125] D. Brück, H. Elmqvist, S. Mattsson, and H. Olsson. Dymola for multi-engineering modeling and simulation. In *Proceedings of modelica*, volume 2002. Citeseer, 2002.

- [126] M. Tiller. *Introduction to physical modeling with Modelica*. Springer Science & Business Media, 2001.
- [127] Modelica Association. *Modelica™ - A Unified Object-Oriented Language for Systems Modeling*, 2023.
- [128] L. Petzold. Dassl. a differential/algebraic system solver. Technical report, Lawrence Livermore National Lab.(LLNL), Livermore, CA (United States), 1982.
- [129] L. Petzold. Description of dassl: a differential/algebraic system solver. Technical report, Sandia National Labs., Livermore, CA (USA), 1982.
- [130] S. Campbell, J. Chancelier, R. Nikoukhah, S. Campbell, and R. Nikoukhah. *Modeling and Simulation in SCILAB*. Springer, 2010.
- [131] G. Van Rossum, F. Drake, et al. *Python reference manual*, volume 111. Centrum voor Wiskunde en Informatica Amsterdam, 1995.
- [132] G. Van Rossum and F. Drake. *Python 3 Reference Manual*. CreateSpace, Scotts Valley, CA, 2009.
- [133] T. Oliphant. Python for scientific computing. *Computing in Science & Engineering*, 9(3):10–20, 2007.
- [134] F. Perez, B. Granger, and J. Hunter. Python: an ecosystem for scientific computing. *Computing in Science & Engineering*, 13(2):13–21, 2010.
- [135] C. Harris et al. Array programming with NumPy. *Nature*, 585(7825):357–362, September 2020.
- [136] P. Virtanen et al. SciPy 1.0: Fundamental Algorithms for Scientific Computing in Python. *Nature Methods*, 17:261–272, 2020.
- [137] F. Perez and B. Granger. Ipython: A system for interactive scientific computing. *Computing in Science & Engineering*, 9(3):21–29, 2007.
- [138] T. Brown, J. Hörsch, and D. Schlachtberger. Pypsa: Python for power system analysis. *arXiv preprint arXiv:1707.09913*, 2017.
- [139] F. Witte and I. Tuschy. TESPpy: Thermal Engineering Systems in Python. *Journal of Open Source Software*, 5(49):2178, 2020.
- [140] I. Bell, J. Wronski, S. Quoilin, and V. Lemort. Pure and pseudo-pure fluid thermophysical property evaluation and the open-source thermophysical property library coolprop. *Industrial & Engineering Chemistry Research*, 53(6):2498–2508, 2014.

- [141] M. Hänninen and J. Ylijoki. *The one-dimensional separate two-phase flow model of APROS*. Citeseer, 2008.
- [142] R. Salko et al. Ctf: A modernized, production-level, thermal hydraulic solver for the solution of industry-relevant challenge problems in pressurized water reactors. *Nuclear Engineering and Design*, 397:111927, 2022.
- [143] S. Patankar and D. Spalding. A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. In *Numerical prediction of flow, heat transfer, turbulence and combustion*, pages 54–73. Elsevier, 1983.
- [144] C. Fletcher and R. Schultz. Relap5/mod3 code manual. Technical report, Nuclear Regulatory Commission, Washington, DC (United States), 1992.
- [145] F. Barre and M. Bernard. The cathare code strategy and assessment. *Nuclear Engineering and Design*, 124(3):257–284, 1990.
- [146] D. Bestion. The physical closure laws in the cathare code. *Nuclear Engineering and Design*, 124(3):229–245, 1990.
- [147] G. Geffraye, O. Antoni, M. Farvacque, D. Kadri, G. Lavialle, B. Rameau, and A. Ruby. Cathare 2 v2. 5_2: A single version for various applications. *Nuclear Engineering and Design*, 241(11):4456–4463, 2011.
- [148] J. Scott and M. Tuuma. *Algorithms for sparse linear systems*. Springer Nature, 2023.
- [149] M.I. Board. Mars climate orbiter mishap investigation board phase i report november 10, 1999, 1999.
- [150] G. Cantore. *Macchine: appunti tratti dalle lezioni del Prof. Cantore*. Esculapio, 1999.
- [151] E. Stalio. Aerodynamics: lecture notes for the advanced automotive engineering course, 2019.
- [152] J. Anderson. *Computational Fluid Dynamics*. Computational Fluid Dynamics: The Basics with Applications. McGraw-Hill Education, 1995.
- [153] D.T. P.H. Mellor, D. Roberts. Lumped parameter thermal model for electrical machines of tefc design. *IEE Proceedings B (Electric Power Applications)*, 138:205–218(13), September 1991.
- [154] F. Masana. A new approach to the dynamic thermal modelling of semiconductor packages. *Microelectronics Reliability*, 41(6):901–912, 2001.

- [155] X. Li, F. He, and L. Ma. Thermal management of cylindrical batteries investigated using wind tunnel testing and computational fluid dynamics simulation. *Journal of Power Sources*, 238:395–402, 2013.
- [156] R.L. Kotnik. Equivalent thermal circuit for nonventilated motors. *Electrical Engineering*, 74(3):213–213, 1955.
- [157] S. Chowdhury and P. Baski. A simple lumped parameter thermal model for electrical machine of tefc design. In *2010 Joint International Conference on Power Electronics, Drives and Energy Systems 2010 Power India*, pages 1–7, 2010.
- [158] C. Kral, A. Haumer, and T. Bauml. Thermal model and behavior of a totally-enclosed-water-cooled squirrel-cage induction machine for traction applications. *IEEE Transactions on Industrial Electronics*, 55(10):3555–3565, 2008.
- [159] T. Bauml, C. Kral, A. Haumer, and H. Kapeller. Enhanced thermal model of a totally enclosed fan cooled squirrel cage induction machine. In *2007 IEEE International Electric Machines Drives Conference*, volume 2, pages 1054–1058, 2007.
- [160] O. Aglen and A. Andersson. Thermal analysis of a high-speed generator. In *38th IAS Annual Meeting on Conference Record of the Industry Applications Conference, 2003.*, volume 1, pages 547–554 vol.1, 2003.
- [161] F. Ahmed, E. Ghosh, and N. Kar. Transient thermal analysis of a copper rotor induction motor using a lumped parameter temperature network model. In *2016 IEEE Transportation Electrification Conference and Expo (ITEC)*, pages 1–6, 2016.
- [162] A. EL-Refaie, N. Harris, T. Jahns, and K. Rahman. Thermal analysis of multibarrier interior pm synchronous machine using lumped parameter model. *IEEE Transactions on Energy Conversion*, 19(2):303–309, 2004.
- [163] P. Ponomarev, M. Polikarpova, and J. Pyrhönen. Thermal modeling of directly-oil-cooled permanent magnet synchronous machine. In *2012 XXth International Conference on Electrical Machines*, pages 1882–1887, 2012.
- [164] A. Boglietti, A. Cavagnino, M. Lazzari, and A. Pastorelli. A simplified thermal model for variable speed self cooled industrial induction motor. In *Conference Record of the 2002 IEEE Industry Applications Conference. 37th IAS Annual Meeting (Cat. No.02CH37344)*, volume 2, pages 723–730 vol.2, 2002.
- [165] G. Henneberger, K. Yahia, and M. Schmitz. Calculation and identification of a thermal equivalent circuit of a water cooled induction motor for electric vehicle applications. In *1995 Seventh International Conference on Electrical Machines and Drives (Conf. Publ. No. 412)*, pages 6–10, 1995.

- [166] O. Wallscheid and J. Böcker. Global identification of a low-order lumped-parameter thermal network for permanent magnet synchronous motors. *IEEE Transactions on Energy Conversion*, 31(1):354–365, 2016.
- [167] C. Sciascera, P. Giangrande, L. Papini, C. Gerada, and M. Galea. Analytical thermal model for fast stator winding temperature prediction. *IEEE Transactions on Industrial Electronics*, 64(8):6116–6126, 2017.
- [168] A. Boglietti, A. Cavagnino, and D. Staton. Thermal sensitivity analysis for tefc induction motors. In *Second International Conference on Power Electronics, Machines and Drives (PEMD 2004)*., volume 1, pages 160–165 Vol.1, 2004.
- [169] K. Bennion and J. Cousineau. Sensitivity analysis of traction drive motor cooling. In *2012 IEEE Transportation Electrification Conference and Expo (ITEC)*, pages 1–6, 2012.
- [170] B. Assaad, K. El kadri Benkara, S. Vivier, G. Friedrich, and A. Michon. Thermal design optimization of electric machines using a global sensitivity analysis. *IEEE Transactions on Industry Applications*, 53(6):5365–5372, 2017.
- [171] B. Assaad, K. El kadri Benkara, G. Friedrich, S. Vivier, and A. Michon. Reducing the complexity of thermal models for electric machines via sensitivity analyses. In *2017 IEEE Energy Conversion Congress and Exposition (ECCE)*, pages 4658–4665, 2017.
- [172] M. Cavazzuti, G. Gaspari, S. Pasquale, and E. Stalio. Thermal management of a formula e electric motor: Analysis and optimization. *Applied Thermal Engineering*, 157:113733, 2019.
- [173] S. Yamazaki, Y. Kurosaki, and T. Wakisaka. Anisotropic thermal conductivity of lamination stacks of non-oriented electrical steel. *Tetsu-to-Hagane*, 107(2):121–127, 2021.
- [174] M. Seilmayer and V. Katepally. Thermal conductivity survey of different manufactured insulation systems of rectangular copper wires. *AIMS Electronics and Electrical Engineering*, 2(1):27–36, 2018.
- [175] L. Marackova, V. Melcova, J. Samek, and O. Zmeskal. Thermal properties of electro insulating paper. In *Chemistry and Life*, volume 955 of *Materials Science Forum*, pages 25–30. Trans Tech Publications Ltd, 6 2019.
- [176] C. Luo, Z. Zhao, D. Zhang, C. Li, C. Liu, Y. Li, and J. Zhang. Temperature effect on the thermal properties of insulation paper. *Electrical Engineering*, 103:1083–1091, 2021.

- [177] S. Nategh, Z. Huang, A. Krings, O. Wallmark, and M. Leksell. Thermal modeling of directly cooled electric machines using lumped parameter and limited cfd analysis. *IEEE Transactions on Energy Conversion*, 28(4):979–990, 2013.
- [178] F. Ahmed, P. Roy, M. Towhidi, G. Feng, and N.C. Kar. Cfd and lptn hybrid technique to determine convection coefficient in end-winding of tefc induction motor with copper rotor. In *IECON 2019 - 45th Annual Conference of the IEEE Industrial Electronics Society*, volume 1, pages 939–944, 2019.
- [179] B. Guo, Y. Huang, Y. Guo, and J. Zhu. Thermal analysis of the conical rotor motor using lptn with accurate heat transfer coefficients. *IEEE Transactions on Applied Superconductivity*, 26(7):1–7, 2016.
- [180] M. Park and S. Kim. Development and validation of lumped parameter thermal network model on rotational oil spray cooled motor for electric vehicles. *Applied Thermal Engineering*, 225:120176, 2023.
- [181] S. Mezani, N. Takorabet, and B. Laporte. A combined electromagnetic and thermal analysis of induction motors. *IEEE Transactions on Magnetics*, 41(5):1572–1575, 2005.
- [182] O. Okoro. Steady and transient states thermal analysis of a 7.5-kw squirrel-cage induction machine at rated-load operation. *IEEE Transactions on Energy Conversion*, 20(4):730–736, 2005.
- [183] J. Fan, C. Zhang, Z. Wang, Y. Dong, C.E. Nino, A.R. Tariq, and E.G. Strangas. Thermal analysis of permanent magnet motor for the electric vehicle application considering driving duty cycle. *IEEE Transactions on Magnetics*, 46(6):2493–2496, 2010.
- [184] L. Alberti and N. Bianchi. A coupled thermal-electromagnetic analysis for a rapid and accurate prediction of im performance. *IEEE Transactions on Industrial Electronics*, 55(10):3575–3582, 2008.
- [185] G. Dajaku and D. Gerling. An improved lumped parameter thermal model for electrical machines. In *17th International conference on electrical machines (ICEM2006)*, pages 1–6. Citeseer, 2006.
- [186] D. Gerling and G. Dajaku. Thermal calculation of systems with distributed heat generation. In *Thermal and Thermomechanical Proceedings 10th Intersociety Conference on Phenomena in Electronics Systems, 2006. IThERM 2006.*, pages 8 pp.–652, 2006.
- [187] D. Gerling and G. Dajaku. Novel lumped-parameter thermal model for electrical systems. In *2005 European Conference on Power Electronics and Applications*, pages 10 pp.–P10, 2005.

- [188] R. Wrobel, A. Griffo, and P.H. Mellor. Scaling of ac copper loss in thermal modeling of electrical machines. In *2012 XXth International Conference on Electrical Machines*, pages 1424–1429, 2012.
- [189] M. Pastura, R. Notari, S. Nuzzo, D. Barater, and G. Franceschini. Ac losses analysis and design guidelines for hairpin windings with segmented conductors. *IEEE Transactions on Transportation Electrification*, pages 1–1, 2023.
- [190] M. Yovanovich. A general expression for predicting conduction shape factors. *11th Aerospace Sciences Meeting*, 1973.
- [191] I.S. Bjorklund and W.M. Kays. Heat transfer between concentric rotating cylinders. *Journal of Heat Transfer*, 81(3):175–183, 08 1959.
- [192] M. Pastura. Hairpin windings for high reliability and high power density electrical machines. *PhD thesis - Alma Mater Studiorum - University of Bologna*, 2023.
- [193] M. Raya, E. Solà, M. Vellvehi, X. Perpiñà, P. Lasserre, S. Busquets-Monge, and X. Jordà. Sic mosfet chip embedded switching-cell for multilevel converters. In *IEEE 37th International Symposium on Power Semiconductor Devices and ICs (ISPSD)*, 2025.
- [194] R. Rafieezadeh, S. Busquets-Monge, and S. Alepuz. Thermo-electrical modeling of multilevel switching-cell-array-based power converters. In *2023 IEEE 32nd International Symposium on Industrial Electronics (ISIE)*, pages 1–8, 2023.
- [195] F. Menter. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, 32(8):1598 – 1605, 1994. Cited by: 13968; All Open Access, Green Open Access.
- [196] H. Weller, G. Tabor, H. Jasak, and C. Fureby. A tensorial approach to computational continuum mechanics using object-oriented techniques. *Computers in Physics*, 12(6):620–631, 1998.
- [197] P.J. Roache. Quantification of uncertainty in computational fluid dynamics. *Annual review of fluid Mechanics*, 29(1):123–160, 1997.
- [198] M. Seilmayer and V.K. K. Thermal conductivity survey of different manufactured insulation systems of rectangular copper wires. *AIMS Electronics and Electrical Engineering*, 2(1):27, 2018.
- [199] S. Nategh, A. Krings, O. Wallmark, and M. Leksell. Evaluation of impregnation materials for thermal management of liquid-cooled electric machines. *IEEE Transactions on Industrial Electronics*, 61(11):5956–5965, 2014.

- [200] S. Yamazaki, Y. Kurosaki, and T. Wakisaka. Anisotropic thermal conductivity of lamination stacks of non-oriented electrical steel. *Tetsu To Hagane - Journal Of The Iron And Steel Institute Of Japan*, 107(2):121–127, 2021.
- [201] R.W.A. LLC. Physical properties of permanent magnet materials. <https://www.allianceorg.com/pdfs/PhysicalPropertiesofMagnets.pdf>, 2021. [Online; accessed 14-September-2024].
- [202] W. Kays and A. London. *Compact Heat Exchangers*. 3rd ed. McGraw-Hill, 1984.
- [203] A. London and R. Shah. Offset Rectangular Plate-Fin Surfaces–Heat Transfer and Flow Friction Characteristics. *Journal of Engineering for Gas Turbines and Power*, 90(3):218–228, 1968.
- [204] A. Wieting. Empirical Correlations for Heat Transfer and Flow Friction Characteristics of Rectangular Offset-Fin Plate-Fin Heat Exchangers. *Journal of Heat Transfer*, 97(3):488–490, 1975.
- [205] W.Y. S. Mochizuki, Y. Yagi. Transport phenomena in stacks of interrupted parallel-plate surfaces. *Experimental Heat Transfer*, 1(2):127–140, 1987.
- [206] R. Manglik and A. Bergles. Heat transfer and pressure drop correlations for the rectangular offset strip fin compact heat exchanger. *Exp. Therm. Fluid Sci.*, 10(2):171–180, 1995. *Aerospace Heat Exchanger Technology*.
- [207] E. Dubrovsky and V. Vasiliev. Enhancement of convective heat transfer in rectangular ducts of interrupted surfaces. *International Journal of Heat and Mass Transfer*, 31(4):807–818, 1988.
- [208] H. Joshi and R. Webb. Heat transfer and friction in the offset stripfin heat exchanger. *International Journal of Heat and Mass Transfer*, 30(1):69–84, 1987.
- [209] Y. Yang and Y. Li. General prediction of the thermal hydraulic performance for plate-fin heat exchanger with offset strip fins. *International Journal of Heat and Mass Transfer*, 78:860–870, 2014.
- [210] J. Wen, H. Yang, X. Tong, K. Li, S. Wang, and Y. Li. Optimization investigation on configuration parameters of serrated fin in plate-fin heat exchanger using genetic algorithm. *International Journal of Thermal Sciences*, 101:116–125, 2016.
- [211] X. Zheng and Z. Qi. A comprehensive review of offset strip fin and its applications. *Applied Thermal Engineering*, 139:61–75, 2018.

- [212] L. Li, J. Lu, H. Fang, Z. Yin, T. Wang, R. Wang, X. Fan, L. Zhao, D. Tan, and Y. Wan. Lattice boltzmann method for fluid-thermal systems: Status, hotspots, trends and outlook. *IEEE Access*, 8:27649–27675, 2020.
- [213] M. Kim, J. Lee, S. Yook, and K. Lee. Correlations and optimization of a heat exchanger with offset-strip fins. *International Journal of Heat and Mass Transfer*, 54(9):2073–2079, 2011.
- [214] R. Manglik, J. Zhang, and A. Muley. Low reynolds number forced convection in three-dimensional wavy-plate-fin compact channels: fin density effects. *International Journal of Heat and Mass Transfer*, 48(8):1439–1449, 2005.
- [215] E.S. L. Goldstein, Jr. Heat/mass transfer characteristics for flow in a corrugated wall channel. *Journal of Heat Transfer*, 99(2):187–195, 1977.
- [216] T. Rush, T. Newell, and A. Jacobi. An experimental study of flow and heat transfer in sinusoidal wavy passages. *International Journal of Heat and Mass Transfer*, 42(9):1541–1553, 1999.
- [217] D. Junqi, C. Jiangping, C. Zhijiu, Z. Yimin, and Z. Wenfeng. Heat transfer and pressure drop correlations for the wavy fin and flat tube heat exchangers. *Applied Thermal Engineering*, 27(11):2066–2073, 2007.
- [218] Y. Asako and M. Faghri. Finite-volume solutions for laminar flow and heat transfer in a corrugated duct. *IEEE Transactions on Magnetics*, 109(3):627–634, 1987.
- [219] J. Zhang, J. Kundu, and R. Manglik. Effect of fin waviness and spacing on the lateral vortex structure and laminar heat transfer in wavy-plate-fin cores. *International Journal of Heat and Mass Transfer*, 47(8-9):1719–1730, 2004.
- [220] H. Metwally and R. Manglik. Enhanced heat transfer due to curvature-induced lateral vortices in laminar flows in sinusoidal corrugated-plate channels. *International Journal of Heat and Mass Transfer*, 47(10-11):2283–2292, 2004.
- [221] M. Aliabadi, F. Hormozi, and E. Rad. New correlations for wavy plate-fin heat exchangers: different working fluids. *International Journal of Numerical Methods for Heat & Fluid Flow*, 24(5):1086–1108, 2012.
- [222] Y. Tao, Y. He, Z. Wu, and W. Tao. Numerical design of an efficient wavy fin surface based on the local heat transfer coefficient study. *Journal Of Enhanced Heat Transfer*, 14(4):315–332, 2007.
- [223] M. Pham, F. Plourde, and S. Doan. Turbulent heat and mass transfer in sinusoidal wavy channels. *International Journal of Heat and Fluid Flow*, 29(5):1240–1257, 2008.

- [224] E.H. de Vasconcelos Segundo, V.C. Mariani, and L.S. Coelho. Design of heat exchangers using falcon optimization algorithm. *Applied Thermal Engineering*, 156:119–144, 2019.
- [225] H. Zarea, F.M. Kashkooli, A.M. Mehryan, M. Saffarian, and E.N. Beherghani. Optimal design of plate-fin heat exchangers by a bees algorithm. *Applied Thermal Engineering*, 69(1):267–277, 2014.
- [226] M. Yousefi, R. Enayatifar, and A. Darus. Optimal design of plate-fin heat exchangers by a hybrid evolutionary algorithm. *International Communications in Heat and Mass Transfer*, 39(2):258–263, 2012.
- [227] M. Yousefi, R. Enayatifar, A. Darus, and A. Abdullah. Optimization of plate-fin heat exchangers by an improved harmony search algorithm. *Applied Thermal Engineering*, 50(1):877–885, 2013.
- [228] J. Hao, Q. Chen, J. Ren, M. Zhang, and J. Ai. An experimental study on the offset-strip fin geometry optimization of a plate-fin heat exchanger based on the heat current model. *Applied Thermal Engineering*, 154:111–119, 2019.
- [229] J. Hao, Q. Chen, X. Li, and T. Zhao. A correction factor-based general thermal resistance formula for heat exchanger design and performance analysis. *International Journal of Thermal Sciences*, 30(3):892–901, May 2021.
- [230] Q. Chen, J. Hao, and T. Zhao. An alternative energy flow model for analysis and optimization of heat transfer systems. *International Journal of Heat and Mass Transfer*, 108:712–720, 2017.
- [231] M. Zhang, Q. Chen, W. Shao, X. Chen, and J. Hao. An integral identification method of characteristic parameters and optimization of parallel connection heat transfer systems based on the power flow method. *Applied Thermal Engineering*, 143:1057–1067, 2018.
- [232] M. Awad and Y. Muzychka. Models for pressure drop and heat transfer in air cooled compact wavy fin heat exchangers. *Journal Of Enhanced Heat Transfer*, 18(3):191–207, 2011.
- [233] S.V. Patankar and D.B. Spalding. Paper 5 - a calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. In S. Patankar, A. Pollard, A. Singhal, and S. Vanka, editors, *Numerical Prediction of Flow, Heat Transfer, Turbulence and Combustion*, pages 54–73. Pergamon, 1983.
- [234] P. Forchheimer. Wasserbewegung durch boden. *Zeitschrift des Vereines Deutscher Ingenieure*, 45:1782–1788, 1901.

- [235] D. Marquardt. An algorithm for least-squares estimation of nonlinear parameters. *Journal of the Society for Industrial and Applied Mathematics*, 11(2):431–441, 1963.
- [236] P. Thomas and J. Middlecoff. Direct control of the grid point distribution in meshes generated by elliptic equations. *AIAA journal*, 18(6):652–656, 1980.
- [237] P. Virtanen et al. SciPy 1.0: Fundamental Algorithms for Scientific Computing in Python. *Nature Methods*, 17:261–272, 2020.
- [238] M. Grespan, A. Leonforte, M. Cavazzuti, L. Calò, and D. Angeli. Accurate reduced-order modelling of plate-fin heat exchangers. In *International Heat Transfer Conference Digital Library*. Begel House Inc., 2023.
- [239] O. Fadodun, O. Fadodun, and A. Kaood. Hydrothermal performance and entropy production rate of rgo-co₃o₄/h₂o hybrid nanofluid in corrugated-converging pipes. *International Journal of Thermal Sciences*, 199:108911, 2024.
- [240] M. Hassan and A. Kaood. Multi-criteria assessment of enhanced radiant ceiling panels using internal longitudinal fins. *Building and Environment*, 224:109554, 2022.
- [241] S. Manson. Correlations of heat-transfer data and of friction data for interrupted plane fins staggered in successive rows. Technical report, NASA, 1950.
- [242] S. Roesler, B. Kerler, J. Kuehndel, and S. Forker. Experimental and numerical investigation of flow instabilities in offset strip fin heat exchanger geometries. In *International Heat Transfer Conference Digital Library*. Begel House Inc., 2023.
- [243] J. Dong, J. Chen, Z. Chen, and Y. Zhou. Air-side thermal hydraulic performance of offset strip fin aluminum heat exchangers. *Applied Thermal Engineering*, 27:306–313, 2007.
- [244] M. Kim and K. Lee. The thermoflow characteristics of an oscillatory flow in offset-strip fins. *Numerical Heat Transfer, Part A: Applications An International Journal of Computation and Methodology*, 58(11):835–851, 2010.
- [245] Y. Yang, Y. Li, B. Si, and J. Zheng. Performance evaluation of heat transfer enhancement in plate-fin heat exchangers with offset strip fins. *Phys. Procedia*, 67:543–550, 2015. Proceedings of the 25th International Cryogenic Engineering Conference and International Cryogenic Materials Conference 2014.
- [246] Y. Yang, Y. Li, B. Si, J. Zheng, and R. Kang. Analysis of the fin performance of offset strip fins used in plate-fin heat exchangers. *Journal of Heat Transfer*, 138(10):101801, 2016.

- [247] H. Peng, X. Ling, and J. Li. Performance investigation of an innovative offset strip fin arrays in compact heat exchangers. *Energy Conversion and Management*, 80:287–297, 2014.
- [248] G. Michna, A. Jacobi, and R. Burton. An experimental study of the friction factor and mass transfer performance of an offset-strip fin array at very high reynolds numbers. *Journal of Heat Transfer*, 2007.
- [249] Y. Muzychka and M. Yovanovich. Modeling the f and j characteristics for transverse flow through an offset strip fin at low reynolds number. *Journal Of Enhanced Heat Transfer*, 8(4):243–259, 2001.
- [250] F. Tinaut, A. Melgar, and A. Ali. Correlations for heat transfer and flow friction characteristics of compact plate-type heat exchangers. *International Journal of Heat and Mass Transfer*, 35(7):1659–1665, 1992.
- [251] K.H. S. Hu. Prandtl number effect on offset fin heat exchanger performance: predictive model for heat transfer and pressure drop. *International Journal of Heat and Mass Transfer*, 38:1043–1051, 1995.
- [252] S. Hu and K. Herold. Prandtl number effect on offset fin heat exchanger performance: experimental results. *International Journal of Heat and Mass Transfer*, 38(6):1053–1061, 1995.
- [253] L. Guo, J. Chen, F. Qin, Z. Chen, and W. Zhang. Empirical correlations for lubricant side heat transfer and friction characteristics of the hpd type steel offset strip fins. *International Communications in Heat and Mass Transfer*, 35:251–262, 2008.
- [254] L. Guo, F. Qin, J. Chen, Z. Chen, and Y. Zhou. Influence of geometrical factors and pressing mould wear on thermal-hydraulic characteristics for steel offset strip fins at low reynolds number. *International Journal of Thermal Sciences*, 46(12):1285–1296, 2007.
- [255] L. Guo, F. Qin, J. Chen, and Z. Chen. Lubricant side thermal-hydraulic characteristics of steel offset strip fins with different flow angles. *Applied Thermal Engineering*, 28:907–914, 2008.
- [256] H. Xiang, W. Du, and J. Zhao. Numerical investigation on the thermal-hydraulic performance of high prandtl number fluid in offset strip fin with different offset. *International Journal of Heat and Mass Transfer*, 220:124915, 2024.
- [257] J. Du, M. Yang, and S. Yang. Correlations and optimization of a heat exchanger with offset fins by genetic algorithm combining orthogonal design. *Applied Thermal Engineering*, 107:1091–1103, 2016.

- [258] N. Kedam, D. Uglanov, E. Blagin, A. Gorshkalev, R. Panshin, and J. Liu. Unified ann model for heat transfer factor (j) and friction factor (f) prediction in offset strip and wavy fin pfhes. *Case Studies in Thermal Engineering*, 53:103845, 2024.
- [259] K. Chandan, K. Nagaraja, F. Gamaoun, T. Smitha, N. Neelima, U. Khan, and A. Hassan. Improving flow efficiency in micro and mini-channels with offset strip fins: A stacking ensemble technique for accurate friction factor prediction in steady periodically developed flow. *Case Studies in Thermal Engineering*, 56:104232, 2024.
- [260] M. Salman, P. Dhamodharan, R. Prabakaran, and S. Kim. Condensation heat transfer performance of r290 in a plate heat exchanger for electric vehicle heat pumps. *International Journal of Thermal Sciences*, 203:109103, 2024.
- [261] J. Kim, Y. Park, H. Choi, J. Oh, and H. Lee. Analyses of the thermal and hydraulic characteristics of a perpendicular-flow offset-strip-fin heat exchanger for electric vehicles. *International Journal of Heat and Mass Transfer*, 226:125520, 2024.
- [262] B. Kim and B. Sohn. An experimental study of flow boiling in a rectangular channel with offset strip fins. *International Journal of Heat and Fluid Flow*, 27(3):514–521, 2006.
- [263] B. Pulvirenti, A. Matalone, and U. Barucca. Boiling heat transfer in narrow channels with offset strip fins: Application to electronic chipsets cooling. *Applied Thermal Engineering*, 30(14):2138–2145, 2010.
- [264] C. Ranganayakulu and S. Kabelac. Boiling of r134a in a plate-fin heat exchanger having offset fins. *Journal of Heat Transfer*, 137(12):121002, 2015.
- [265] R. Prabakaran, M. Salman, P. Kumar, D. Lee, and S. Kim. Heat transfer and pressure drop characteristics of r1234yf during evaporation in a plate heat exchanger with offset strip fins: An experimental study. *International Journal of Heat and Mass Transfer*, 194:123091, 2022.
- [266] R. Prabakaran, M. Salman, D. Lee, and S. Kim. Condensation of r1234yf in a plate heat exchanger with an offset strip fin flow structure for electric vehicle heat pumps. *International Communications in Heat and Mass Transfer*, 143:106699, 2023.
- [267] M. Salman, P. Dhamodharan, R. Prabakaran, and S.C. Kim. Condensation heat transfer characteristics of low gwp r290/r1270 blend in a plate heat exchanger for ev heat pumps. *Korean Society of Automotive Engineers Fall Conference and Exhibition*, pages 1818–1818, 2023.
- [268] P. Dhamodharan, M. Salman, R. Prabakaran, and S. Kim. Comparative analysis of r290 and r1234yf cooling performance in offset strip-fin plate heat exchanger for

- electric-vehicle battery thermal management. *International Communications in Heat and Mass Transfer*, 157:107708, 2024.
- [269] M. Pendar, J. Páscoa, and R. Lima. Numerical investigation of automotive paint oven for improving the thermal efficiency. In *Fluids Engineering Division Summer Meeting*, volume 85840, page V002T05A032. American Society of Mechanical Engineers, 2022.
- [270] F. Cramer. Scientific colour maps, June 2023.
- [271] E. Scolaro, M. Beligoj, M.P. Estevez, L. Alberti, M. Renzi, and M. Mattetti. Electrification of agricultural machinery: A review. *IEEE Access*, 9:164520–164541, 2021.
- [272] S. Wongwises and Y. Chokeman. Effect of fin pitch and number of tube rows on the air side performance of herringbone wavy fin and tube heat exchangers. *Energy Conversion and Management*, 46(13):2216–2231, 2005.
- [273] A. Yutaka, N. Hiroshi, and M. Faghri. Heat transfer and pressure drop characteristics in a corrugated duct with rounded corners. *International Journal of Heat and Mass Transfer*, 31(6):1237–1245, 1988.
- [274] D.R. Sawyers, M. Sen, and H.C. Chang. Heat transfer enhancement in three-dimensional corrugated channel flow. *International Journal of Heat and Mass Transfer*, 41(22):3559–3573, 1998.
- [275] C. Wang, W. Fu, and C. Chang. Heat transfer and friction characteristics of typical wavy fin-and-tube heat exchangers. *Experimental Thermal and Fluid Science*, 14(2):174–186, 1997.
- [276] M. Ali and S. Ramadhyani. Experiments on convective heat transfer in corrugated channels. *EXperimental Heat Transfer an International Journal*, 5(3):175–193, 1992.
- [277] L. Yang, Y. Asako, Y. Yamaguchi, and M. Faghri. Numerical prediction of transitional characteristics of flow and heat transfer in a corrugated duct. *IEEE Transactions on Magnetics*, 1997.
- [278] C.C. Wang. Investigation of wavy fin-and-tube heat exchangers: a contribution to databank. *Experimental Heat Transfer*, 12(1):73–89, 1999.
- [279] Z. Chen and J. Ren. Effect of fin spacing on the heat transfer and pressure drop of a two-row plate fin and tube heat exchanger. *International Journal of Refrigeration*, 11(6):356–360, 1988.

- [280] J. Stasiek, M. Collins, M. Ciofalo, and P. Chew. Investigation of flow and heat transfer in corrugated passages—i. experimental results. *International Journal of Heat and Mass Transfer*, 39(1):149–164, 1996.
- [281] M. Ciofalo, J. Stasiek, and M. Collins. Investigation of flow and heat transfer in corrugated passages—ii. numerical simulations. *International Journal of Heat and Mass Transfer*, 39(1):165–192, 1996.
- [282] B. Launder and D. Spalding. Paper 8 - the numerical computation of turbulent flows. In S.V. Patankar, A. Pollard, A.K. Singhal, and S.P. Vanka, editors, *Numerical Prediction of Flow, Heat Transfer, Turbulence and Combustion*, pages 96–116. Pergamon, 1983.
- [283] J. Samagorinsky. General circulation experiments with the primitive equations, part i: The basic experiments. *Monthly Weather Review*, 91:99–152, 1963.
- [284] D.K. Lilly. On the application of the eddy viscosity concept in the inertial sub-range of turbulence. *NCAR manuscript*, 123, 1966.
- [285] L. Ismail and R. Velraj. Studies on fanning friction (f) and colburn (j) factors of offset and wavy fins compact plate fin heat exchanger—a cfd approach. *Numerical Heat Transfer, Part A: Applications*, 56(12):987–1005, 2009.
- [286] J. Dong, L. Su, Q. Chen, and W. Xu. Experimental study on thermal-hydraulic performance of a wavy fin-and-flat tube aluminum heat exchanger. *Applied Thermal Engineering*, 51(1):32–39, 2013.
- [287] N. Kedam, U. Dmitry A, B. Evgeniy V, and G. Alexey A. Heat transfer factor j and friction factor f correlations for offset strip fin and wavy fin of compact plate-fin heat-exchangers. *Case Studies in Thermal Engineering*, 28:101552, 2021.
- [288] E. Galligani. Dispense del corso di metodi numerici per l'ingegneria, 2019.