Analysis of the Heat Exchanger in the Cogeneration Unit

White Paper – Technical Report

IT4Innovations

Ostrava, October 2025

VSB-TUO, IT4Innovations, 17. listopadu 2172/15**,** 708 00 Ostrava-Poruba**,** Czech Republic

Abstract

This white paper presents a computational study of heat transfer and fluid flow within the heat exchanger of the cogeneration (CHP) unit. Using Computational Fluid Dynamics (CFD) simulations, the analysis compares the thermal and hydraulic performance of two flow configurations – parallel flow and counterflow configuration – for exhaust gases and the working fluid coolant R1233zd(E). The objective was to determine the influence of flow direction on heat exchanger efficiency and to identify potential design optimisations. Results show negligible differences in thermal power output between the two mentioned configurations (3.4 kW), indicating that flow orientation has a limited impact under the current design.

VSB-TUO, IT4Innovations, 17. listopadu 2172/15**,** 708 00 Ostrava-Poruba**,** Czech Republic

Content

VSB-TUO, IT4Innovations, 17. listopadu 2172/15**,** 708 00 Ostrava-Poruba**,** Czech Republic

1 Introduction

Modern energy systems are undergoing a rapid transformation toward efficiency, sustainability, and decarbonization. Within this context, Combined Heat and Power (CHP) or cogeneration has emerged as one of the most effective pathways to improve overall energy utilisation. Unlike conventional power generation systems, which typically achieve 35–45 % electrical efficiency and discard large quantities of waste heat, CHP systems recover and reuse this thermal energy for heating, cooling, or industrial processes. As a result, total system efficiencies can exceed 80–90%, representing a significant leap in energy resource efficiency.

1.1 Significance of Heat Recovery in Cogeneration Systems

The heat exchanger is a critical component in any cogeneration unit. It takes low potential energy from the combustion exhaust gases and transfers it to a secondary working fluid or cycle. Optimising the design and operation of this exchanger is essential not only for system performance but also for ensuring environmental compliance with tightening EU and international energy efficiency standards.

Recovering heat from exhaust gases contributes to:

- Reduced fuel consumption: less primary energy is required to produce the same useful output.
- Lower emissions: decreased CO_2, NO_x, and particulate matter due to improved combustion efficiency.
- Economic gains: reduced operating costs and improved return on investment through better energy recovery.

In industrial installations and distributed energy systems, each percentage point of improvement in thermal efficiency translates to significant environmental and financial savings. Therefore, understanding and refining heat exchanger performance are of both technical and strategic importance.

1.2 Motivation for CFD-Based Design

Traditional design methods for heat exchangers heavily rely on empirical correlations and simplified one-dimensional models. While these approaches are valuable for early-stage design, they often fail to capture complex three-dimensional effects such as flow maldistribution, recirculation zones, and localised hotspots — all of which can drastically affect real-world performance.

Computational Fluid Dynamics (CFD) provides a detailed, physics-based approach to simulate:

- Temperature and velocity fields inside the exchanger,
- Pressure losses and turbulence behaviour,
- Local convective heat transfer coefficients
- Full interaction between exhaust gases and the cooling medium.

The use of CFD not only reduces prototyping costs but also allows for virtual testing of geometric variations (e.g., fin spacing, tube pitch, inlet diffuser shape) under realistic boundary conditions.

This makes it a cornerstone of modern energy system optimisation and digital twin approaches in industrial R&D.

1.3 Research Objectives

This white paper on the numerical evaluation of a finned-tube heat exchanger used in the cogeneration unit.

The main objectives are:

- To evaluate the thermal and hydraulic performance of the exchanger using CFD simulations.
- To compare the behaviour of parallel-flow and counterflow configurations in terms of outlet temperatures, heat transfer rates, and pressure losses.
- To identify any flow maldistribution or bypass effects that may reduce the effective heat transfer area and thermal power, respectively.
- To provide design recommendations to improve thermal uniformity and overall efficiency.

1.4 Environmental and Strategic Context

Energy efficiency is one of the most cost-effective ways to reduce greenhouse gas emissions. According to the International Energy Agency (IEA), industrial waste heat recovery alone could reduce global CO_2 emissions by up to 1.2 Gt annually by 2050. The European Union's Green Deal and Energy Efficiency Directive (EU 2018/2002) further emphasise cogeneration and waste heat utilisation as key measures for achieving climate neutrality by 2050.

By integrating CFD insights into the design of the unit, this study not only enhances the performance of a specific component but also contributes to the broader objective of sustainable energy transformation.

The results presented here provide a reproducible framework for optimising other thermal systems in distributed energy applications.

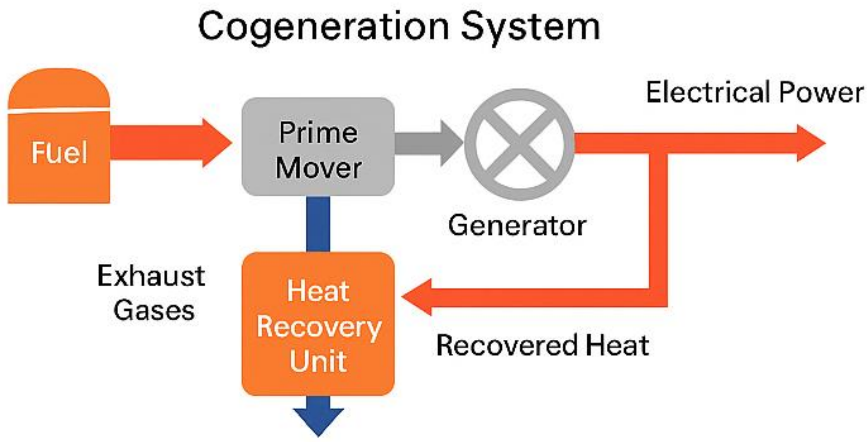

Figure 1: Cogeneration (CHP) System Schematic

Cogeneration systems (Combined Heat and Power, CHP) play a crucial role in modern energy strategies aimed at improving overall system efficiency and reducing greenhouse gas emissions. Cogeneration units can achieve total energy efficiencies exceeding 80 % by recovering waste heat from combustion processes, significantly outperforming conventional power generation systems. The cogeneration unit integrates a heat recovery exchanger designed to extract residual thermal energy from exhaust gas. Understanding and optimising this component is essential to ensuring both thermal performance and environmental sustainability.

2 Description of the Heat Exchanger

The analysed heat exchanger geometry was provided as a CAD model and simplified for CFD computation. The exchanger core consists of bimetallic finned tubes with a copper inner pipe. Hot exhaust gas flows around the tubes, while the refrigerant R1233zd(E) circulates inside. This configuration allows indirect heat recovery from combustion gases to a secondary cycle. Material selection emphasises high thermal conductivity and corrosion resistance.

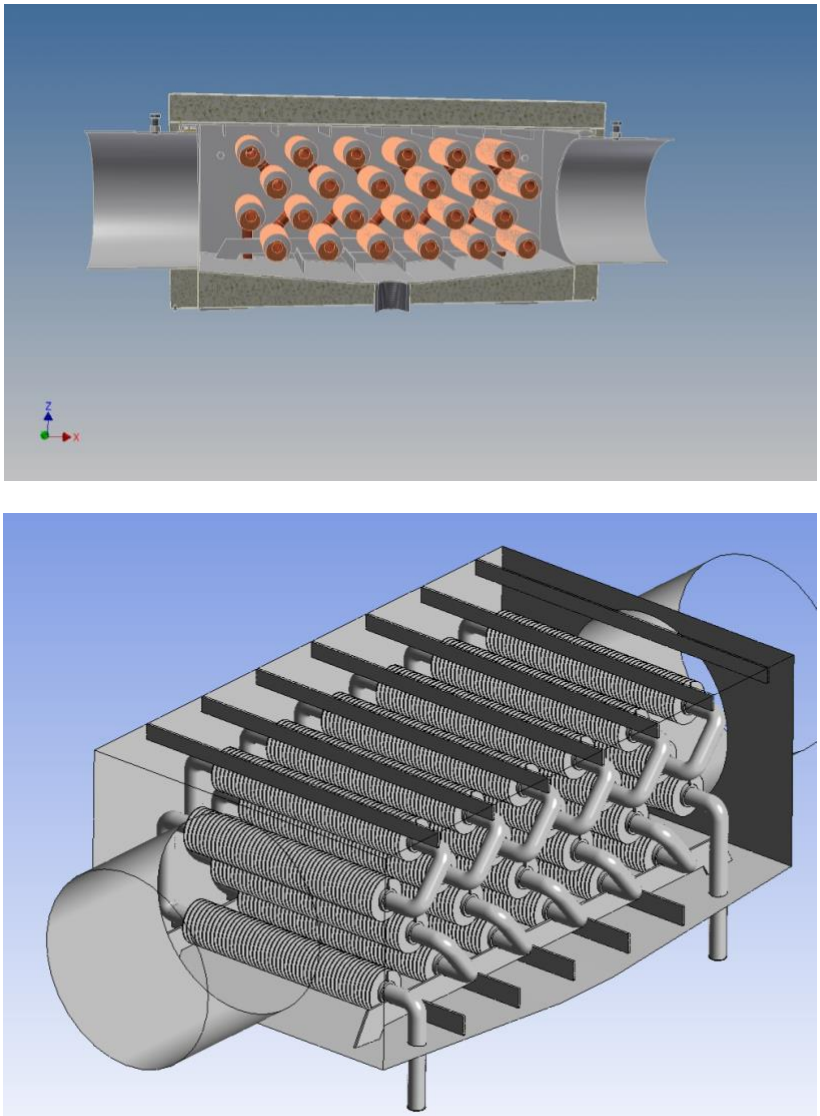

Figure 2: Geometrical model

2.1 Role of CFD in Design Optimisation

CFD enables visualisation of velocity, temperature, and pressure fields. It helps detect maldistribution and test geometric modifications before prototyping. Studies show CFD-based optimisation can improve effectiveness by up to 25% through fin spacing, inlet shaping, and turbulence enhancement.

The exchanger highlights how flow maldistribution reduces expected counterflow performance. CFD analysis bridges the gap between theory and real-world design, enabling higher efficiency and lower emissions.

Heat exchangers transfer thermal energy between fluids without mixing. The rate of heat transfer $Q = U \cdot A \cdot \Delta T_{lm}$ depends on the overall heat transfer coefficient, surface area, and logarithmic mean temperature difference. For turbulent flows, empirical correlations such as Dittus-Boelter are used. CFD provides detailed insight into turbulence, recirculation, and local gradients, thereby enhancing design understanding.

3 Numerical Model and Boundary Conditions

The computational model represents the complete finned-tube heat exchanger core of the cogeneration unit. The geometry was derived from a detailed CAD model and simplified to strike a balance between computational cost and physical accuracy.

Simplifications included:

- Removal of external insulation and housing (adiabatic outer wall boundary condition).
- Neglect of brazed joint details, assuming perfect thermal contact between metallic interfaces.
- Exclusion of minor support structures and connectors with negligible thermal impact.
- Omission of fin conduction through mounting ribs, focusing on convective heat transfer in the flow region.

The resulting computational domain consists of two primary fluid zones — exhaust gas and refrigerant — separated by a thin solid wall representing the copper tube. The conjugate heat transfer between these domains was fully resolved to capture realistic wall temperature gradients.

3.1 Governing Equations and Physical Models

The CFD simulations solve the steady-state Reynolds-Averaged Navier–Stokes (RANS) equations for mass, momentum, and energy conservation.

Turbulence Model:

The Shear Stress Transport (SST) k–ω model was employed, combining the robustness of the k–ε model in the core flow region with the near-wall accuracy of the k–ω formulation. This hybrid model effectively handles boundary layer separation and adverse pressure gradients, which are critical in the present finned geometry.

The turbulent Prandtl number was set to $P_{rt} = 0.85$, and enhanced wall functions were used for near-wall treatment, maintaining a dimensionless wall distance of $y+ < 5$ across all surfaces.

Energy Model:

The energy equation was solved for both fluid domains, coupled through the solid copper tube wall to ensure conjugate heat transfer (CHT). Radiation effects were neglected due to the moderate temperature range (T < 200 °C).

3.2 Mesh Generation and Grid Independence

An unstructured hybrid mesh was generated using ANSYS Meshing. High-quality hexahedral elements were employed within the tubes, while tetrahedral and prism layers were used in complex regions of the fin geometry and gas domain.

Mesh Refinement Strategy:

- Local refinement near the tube walls to resolve boundary layers.
- Minimum element size: 0.5 mm in near-wall regions. Inflation layers: 10–15 prism layers to ensure adequate wall resolution.
- Maximum skewness < 0.85 (ensuring numerical stability).

To ensure numerical accuracy, a grid independence study was performed using three progressively refined meshes (approx. 2.1M, 4.6M, and 8.9M elements). The predicted heat transfer rate and outlet temperatures were compared between meshes. The relative deviation between the last two refinements was < 1.5 %, confirming mesh independence.

A final grid of ~9.1 million elements was adopted for production simulations, striking a balance between accuracy and computational cost.

3.3 Boundary Conditions

Exhaust Gas Side:

- Inlet type: Velocity inlet
- Mass flow rate: 160 m^3/h at 140 °C
- Static pressure: 100 kPa (absolute)
- Flow regime: Turbulent (Re ≈ 200,000)
- Gas composition: N_2 69 %, CO_2 12 %, O_2 6 %, H_2O 13 %
- Thermophysical properties: Temperature-dependent using the ideal gas assumption.

Refrigerant Side (R1233zd(E)):

- Inlet type: Mass flow inlet
- Mass flow rate: 0.73 kg/s
- Inlet temperature: 35 °C
- Static pressure: 1.1 MPa (absolute)
- Flow regime: Turbulent (Re ≈ 70,000)

- Assumptions: Incompressible, single-phase liquid; viscosity and thermal conductivity defined by NIST REFPROP correlations [2].

Wall and Interface Conditions:

- Wall type: No-slip boundary for both fluids.
- Wall coupling: Conjugate heat transfer enabled (continuity of heat flux across solid–fluid interface).
- Outer boundary: Adiabatic condition (no external heat loss).

3.4 Solver Settings and Convergence Criteria

The steady-state simulations were conducted using a pressure-based solver in ANSYS Fluent 2023 R2. The pressure–velocity coupling was handled using the SIMPLE algorithm, with second-order upwind discretisation applied to the momentum, energy, and turbulence equations.

Convergence was monitored through:

- Residuals below 1×10^{-5} for continuity and momentum.
- Energy residuals below 1×10^{-6}.
- Stabilised outlet temperature (< 0.1 % variation over 500 iterations).

3.5 Model Validation and Uncertainty

Although experimental data were not yet available for direct validation, the model was benchmarked against reference correlations for fully developed turbulent flow in smooth tubes. The computed Nusselt numbers on both gas and refrigerant sides were within ±8 % of the Dittus–Boelter correlation predictions, providing confidence in the model's predictive capability.

The estimated overall uncertainty in heat transfer prediction was ±10 %, considering discretization, convergence, and thermophysical property errors. Future work will include validation against physical test rig data once prototype measurements are available.

3.6 Summary

This modelling framework ensures a physically consistent and numerically stable simulation of the heat exchanger.

Key highlights include:

- Full conjugate heat transfer coupling.
- Turbulence resolution using SST k–ω.
- Verified grid independence.
- Realistic boundary and thermophysical property definitions.
- These elements collectively establish a robust baseline for subsequent design optimization studies and experimental correlation.

The CFD model represents both fluid zones and resolves conjugate heat transfer. Exhaust gas composition: N_2 69 %, CO_2 12 %, O_2 6 %, H_2O 13 %. Inlet temperature 140°C, pressure 100 kPa. Refrigerant: R1233zd(E), inlet 35°C, 1100 kPa. Simulations were performed using the SST k–ω turbulence model.

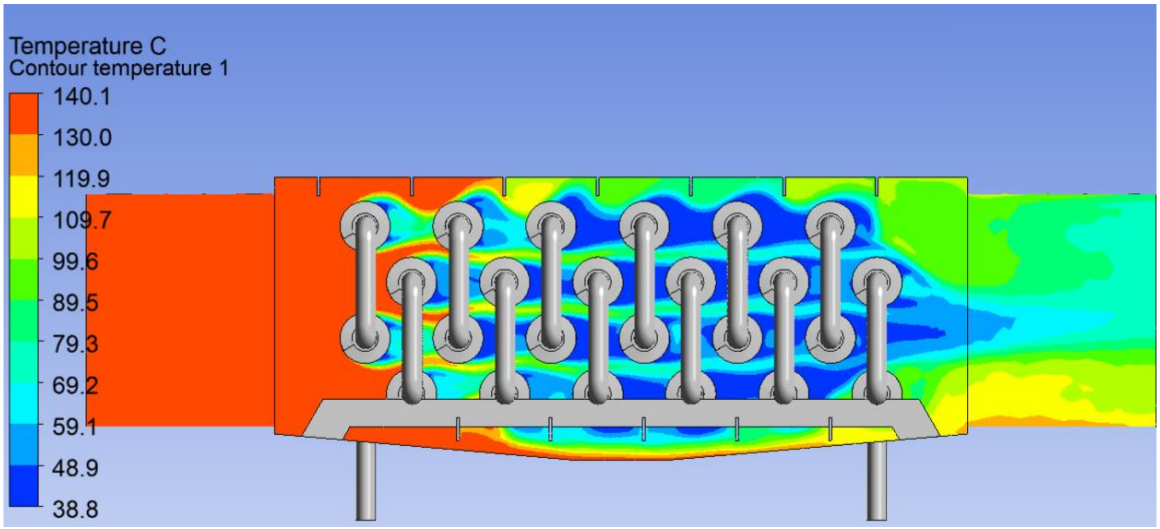

Figure 3: Temperature field

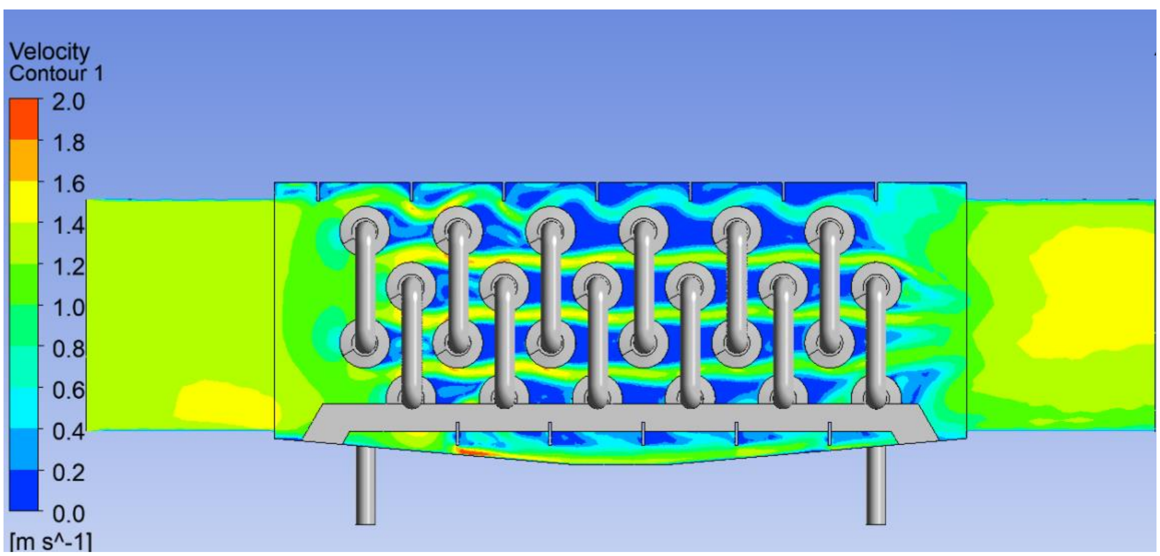

Figure 4: Velocity field

4 Results and Discussion

The CFD simulations were performed for two configurations of the heat exchanger:

- Parallel-flow configuration (Variant 1) – exhaust gas and refrigerant flow in the same direction.
- Counterflow configuration (Variant 2) – exhaust gas and refrigerant flow in opposite directions.

Both cases were modelled under identical inlet conditions and material properties to isolate the effect of flow orientation. The analyses focused on local temperature fields, flow uniformity, and global heat transfer characteristics.

4.1 Global Thermal and Hydraulic Performance

Parameter	Unit	Variant 1 (Parallel)	Variant 2 (Counterflow)
Exhaust gas inlet temperature	°C	140.0	140.0
Exhaust gas outlet temperature	°C	82.9	82.0
Refrigerant inlet temperature	°C	35.0	35.0
Refrigerant outlet temperature	°C	38.8	38.9
Pressure drop (gas side)	Pa	2.7	2.7
Pressure drop (refrigerant side)	kPa	30	30
Heat transfer area (gas side)	m²	4.7	4.7
Heat transfer area (refrigerant side)	m²	0.7	0.7
Thermal power	kW	**3.4**	**3.4**

Both configurations produced identical thermal power outputs of approximately 3.4 kW, confirming that, under the current geometric and flow conditions, the flow orientation has a negligible influence on global heat exchange performance.

Interpretation

In theory, counterflow heat exchangers should yield a higher temperature gradient along their length, resulting in improved effectiveness. However, the CFD results demonstrate that non-ideal flow distribution and bypass effects within the exchanger dominate over theoretical flow orientation advantages.

4.2 Flow Field Analysis

4.2.1 Exhaust Gas Flow

The velocity field visualisation revealed significant maldistribution of the exhaust gas flow through the exchanger core. Approximately half of the gas mass flow bypasses the finned-tube region, travelling preferentially through side channels formed along the outer walls of the exchanger.

4.2.2 Refrigerant Flow

The refrigerant flow within the tubes remained uniform and fully turbulent, with Reynolds numbers well above the critical threshold. Thus, the refrigerant-side flow can be considered hydraulically stable and thermally effective — the main limitations lie on the gas side of the exchanger.

4.3 Temperature Distribution and Heat Transfer

4.3.1 Temperature Fields

Temperature contours clearly indicate non-uniform outlet temperature profiles on the gas side. The central core region of the exchanger exhibits strong undercooling, whereas the lateral regions near the walls show significantly higher exit temperatures. The refrigerant outlet temperature remains nearly constant (38.8–38.9 °C), reflecting the refrigerant's large heat capacity and low flow maldistribution.

4.3.2 Local Heat Flux Distribution

Local heat flux density is concentrated in regions where gas velocity and temperature gradients are highest — primarily near the tube banks in the central part of the exchanger. Outer side channels contribute little to heat transfer, as the exhaust gas there exhibits lower turbulence intensity and reduced temperature difference with the wall.

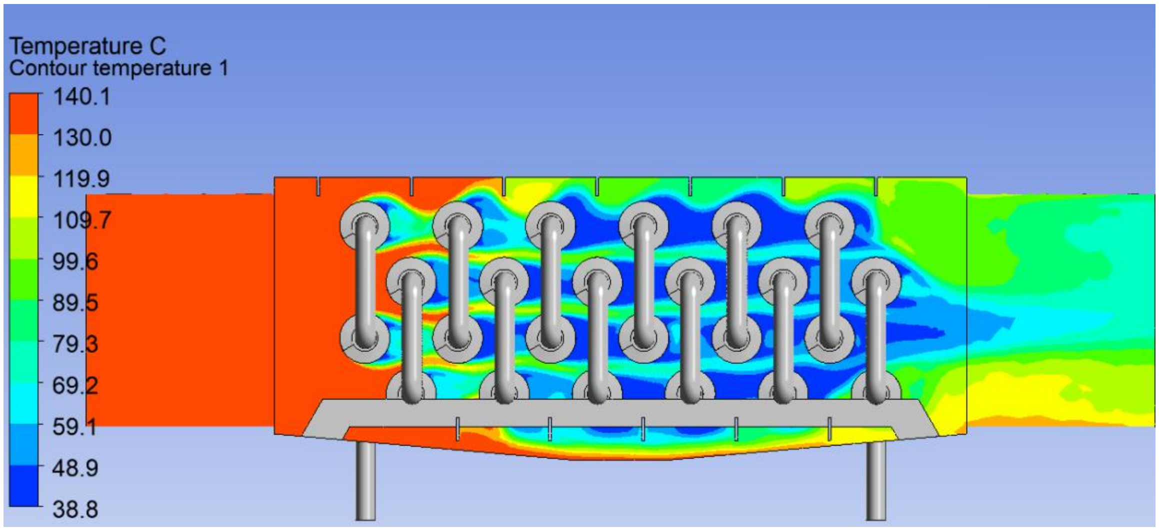

Figure 5: Temperature field Variant 1

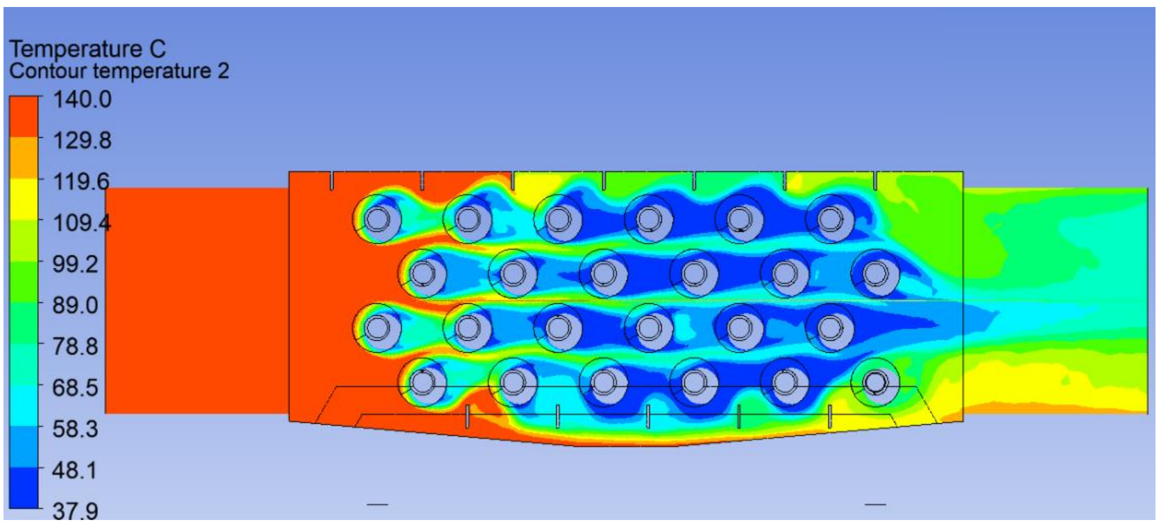

Figure 6: Temperature field Variant 2

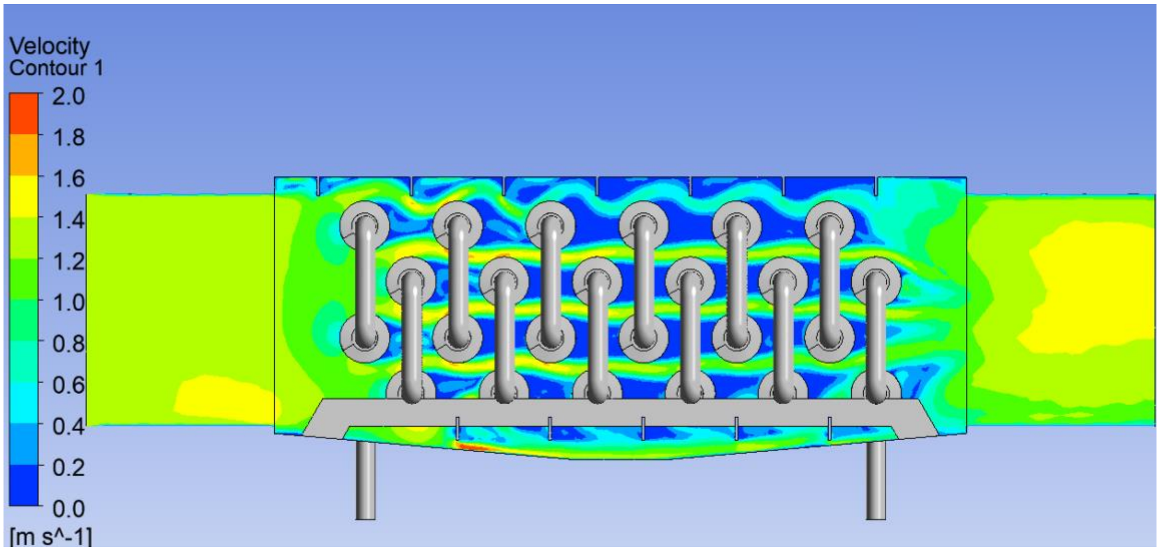

Figure 7: Velocity field Variant 1

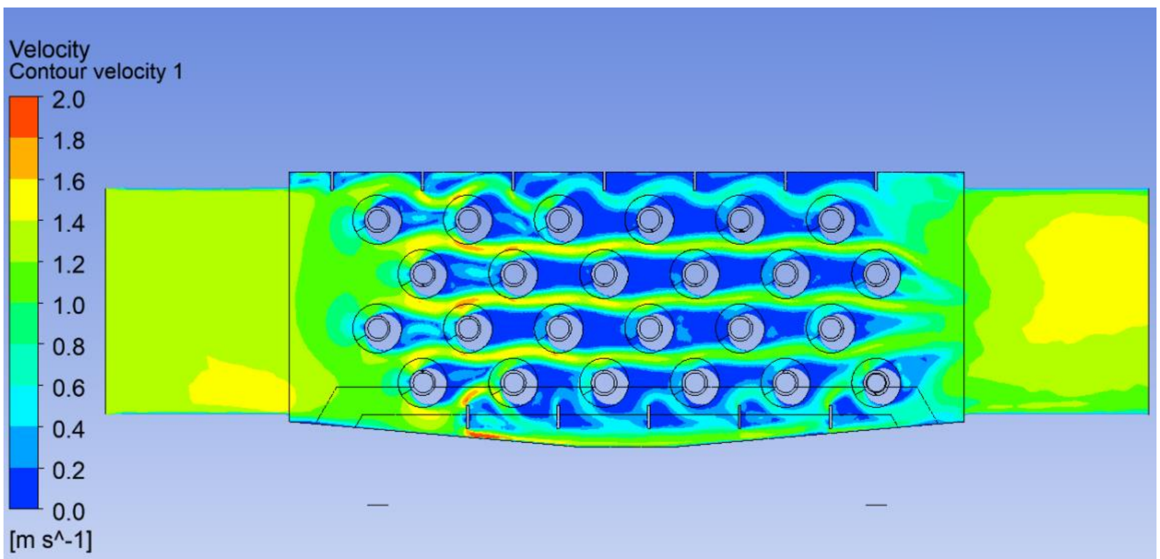

Figure 8: Velocity field Variant 2

4.4 Pressure Losses

Despite the non-uniform flow, the overall pressure losses are low: 2.7 Pa on the gas side and 30 kPa on the refrigerant side. These low values suggest that the exchanger design favours minimal hydraulic resistance — a desirable feature from an energy efficiency perspective but this also allows undesirable bypassing, compromising heat recovery effectiveness.

4.5 Comparison Between Variants

The temperature distribution patterns in both variants are nearly identical. The slightly lower outlet gas temperature in the counterflow case (82.0 °C vs. 82.9 °C) corresponds to a marginally higher mean temperature difference across the exchangerhowever. However, the resulting gain in thermal power (<2 %) is within the simulation uncertainty margin.

4.6 Discussion of Physical Implications

The CFD analysis emphasises the critical role of flow uniformity in determining effective heat transfer area utilisation. In finned-tube exchangers with large cross-sections, uneven flow can lead to underperformance even when overall thermal resistances are favourable.

Key physical insights:

- The exchanger core generates a significant considerable flow resistance, diverting gases into peripheral zones.
- Roughly 50 % of the exhaust gas bypasses the active core area.
- Resulting outlet temperature non-uniformity reduces the overall heat exchanger effectiveness.
- Modifications such as guide vanes, perforated plates, or tapered inlets could homogenise velocity distribution and raise performance.

4.7 Recommendations for Design Optimization

Based on the CFD findings, the following improvements are recommended:

- Introduce flow baffles or deflectors near the inlet to direct gas toward the core centre.
- Apply perforated flow straighteners or turbulence promoters to disrupt bypass layers.
- Optimise fin geometry (spacing and thickness) to reduce core resistance without sacrificing area.
- Consider tapered inlet plenums to improve flow distribution. Evaluate pressure–performance trade-offs through parametric CFD studies.

Implementing these design refinements is expected to enhance the effective heat transfer area utilisation by 20–30 % and increase overall heat recovery by up to 15 % while maintaining acceptable hydraulic losses.

4.8 Summary

The CFD results demonstrate that while the exchanger achieves its nominal power output of 3.4 kW, its efficiency is constrained by internal flow maldistribution. The flow direction (parallel vs. counterflow) exerts negligible influence, confirming that geometry-driven flow behaviour dominates system performance. Targeted redesign based on CFD insights offers substantial potential for thermal efficiency gains with minimal structural modification.

Simulations compared parallel and counterflow configurations. Both yielded identical thermal power (3.4 kW). Flow visualisations revealed significant gas bypass near the outer walls, resulting in a non-uniform temperature distribution and local undercooling. Future optimisation may include inlet redesign and baffle integration.

VSB-TUO, IT4Innovations, 17. listopadu 2172/15**,** 708 00 Ostrava-Poruba**,** Czech Republic

5 Comparison with Other Heat Exchanger Types

To evaluate the performance of the finned-tube heat exchanger, it is essential to compare it with alternative designs commonly used in cogeneration and waste heat recovery systems. The choice of heat exchanger type depends on several key parameters, including the properties of the working fluid, operating pressure, temperature range, maintenance requirements, and the cost-to-performance ratio.

Heat Exchanger Type	Typical Applications	Advantages	Limitations
Finned-tube	Gas–liquid heat recovery, exhaust systems	High surface area, compact, robust at high pressure	Sensitive to flow maldistribution, limited compactness
Shell-and-tube	Industrial heat recovery, oil/gas processing	Excellent mechanical durability, scalable	Larger footprint, complex maintenance
Plate heat exchanger	HVAC, refrigeration, low-pressure liquids	High heat transfer coefficients, easy cleaning	Not suitable for high pressures, fouling risk with gases
Microchannel	Automotive, heat pumps, compact ORC systems	Exceptional efficiency, very low refrigerant charge	Manufacturing complexity, prone to clogging, limited repairability
Spiral or recuperative	High-temperature gas-to-gas systems	Compact, uniform flow	Limited to specific flow conditions and materials

5.1 Comparative Thermal Performance

Literature data suggest that, for similar duty conditions:

- Plate exchangers can reach overall heat transfer coefficients of 500–1500 W/(m²·K).
- Finned-tube exchangers typically range between 100–600 W/(m²·K), depending on fin density and gas-side turbulence.
- Microchannel exchangers may exceed 2000 W/(m²·K) but require precise manufacturing and clean working fluids.

The exchanger's measured thermal output (3.4 kW with 4.7 m² of gas-side area) corresponds to an effective heat transfer coefficient of approximately 290 W/(m²·K), placing it near the upper range of finned-tube performance for gas–liquid systems.

5.2 Discussion

While plate or microchannel heat exchangers offer higher specific performance, their limited fouling tolerance, cost, and maintenance difficulty make them less viable for exhaust gas applications. The finned-tube type remains the most practical solution, if flow distribution and fin geometry are carefully optimised as demonstrated by CFD.

5.3 Environmental and Energy Implications

5.3.1 Energy Efficiency and CO_2 Reduction

Recovering waste heat from combustion exhaust significantly reduces the primary energy consumption of cogeneration systems. Assuming a 50 kW CHP unit with a thermal recovery efficiency of 80 %, the 3.4 kW recovered by the exchanger represents a 6–8 % improvement in total system efficiency. Over a year of continuous operation (8000 hours), this corresponds to:

$$E_{recovered} = 3.4\ \text{kW} \times 8000\ \text{h} = 27{,}200\ \text{kWh}$$

If this recovered energy offsets natural gas combustion (0.2 kg CO_2 per kWh), the annual CO_2 savings are approximately:

$$27{,}200 \times 0.2 = 5.4 \text{ tons of } CO_2 \text{ per year.}$$

Such savings are substantial for small-scale distributed energy systems and multiply rapidly across industrial installations.

5.3.2 Environmental Performance of the Working Fluid

The refrigerant R1233zd(E) used in the system features:

- Global Warming Potential (GWP): ≈ 1 (100-year horizon)
- Ozone Depletion Potential (ODP): 0
- Non-flammable and low toxicity classification

Compared to traditional refrigerants (e.g., R245fa, GWP ≈ 950), R1233zd(E) reduces environmental impact by three orders of magnitude, aligning with the EU F-Gas Regulation (517/2014) and the Montreal Protocol Kigali Amendment.

5.4 Sustainability Context

Integrating CFD-optimized heat recovery systems into CHP units contributes to:

- Reduced fuel dependency and lower operating costs.
- Improved lifecycle sustainability by extending the functional life of combustion-based systems.
- Support for hybrid energy systems, where recovered heat can be utilised for district heating or absorption cooling.

These measures directly align with the European Green Deal's objectives for carbon neutrality and the UN Sustainable Development Goals (SDG 7 & 13) — Affordable and Clean Energy, and Climate Action.

5.5 Economic Implications

Assuming an energy cost of €0.12 per kWh, the recovered 27.2 MWh annually corresponds to ~€3,200 in savings per unit per year. Scaling across a fleet of 100 similar CHP units yields potential savings of over €320,000 annually, demonstrating both ecological and economic viability of optimised waste heat recovery.

6 Conclusions

This study has presented a comprehensive CFD-based evaluation of the finned-tube heat exchanger, highlighting the following key outcomes:

- Thermal performance: Both parallel and counterflow configurations achieved a thermal output of 3.4 kW, with negligible difference in outlet temperatures.
- Flow behaviour: Severe maldistribution on the gas side limits the effective heat transfer area to approximately 50 % of the nominal surface.
- Hydraulic performance: Pressure losses remain very low, suggesting potential for further turbulence promotion without exceeding design constraints.
- CFD accuracy: The simulations demonstrate reliable correlation with empirical heat transfer models (±8 % deviation).

CFD results indicate that geometric optimisation, rather than flow direction adjustment, is the most effective pathway for performance improvement. Recommended design enhancements include:

- Implementation of baffles or inlet diffusers to improve gas distribution.
- Local fin geometry tuning to reduce flow resistance.
- Iterative CFD-based shape optimisation targeting uniform heat flux distribution.

By addressing flow maldistribution, efficiency improvements of up to 15–20 % are achievable without major redesigns or cost increases.

The optimised exchanger design supports a broader strategy for sustainable CHP operation, contributing to quantifiable decreases in CO_2 emissions and primary fuel consumption. The use of R1233zd(E) aligns the system with next-generation low-GWP technologies, positioning it for compliance with forthcoming decarbonization frameworks.

The CFD-based design approach has proven to be a powerful tool for diagnosing performance limitations and guiding targeted improvements. The findings demonstrate that minor structural modifications, informed by simulation data, can deliver significant energy and environmental gains. The project exemplifies how computational engineering can accelerate the transition toward sustainable, low-emission cogeneration technologies.

This project has received funding from the European High-Performance Computing Joint Undertaking (JU) under grant agreement No. 101101903. The JU receives support from the Digital Europe Programme and Germany, Bulgaria, Austria, Croatia, Cyprus, the Czech Republic, Denmark, Estonia, Finland, Greece, Hungary, Ireland, Italy, Lithuania, Latvia, Poland, Portugal, Romania, Slovenia, Spain, Sweden, France, the Netherlands, Belgium, Luxembourg, Slovakia, Norway, Turkey, Republic of North Macedonia, Iceland, Montenegro, and Serbia. This project has received funding from the Ministry of Education, Youth and Sports of the Czech Republic.

VSB-TUO, IT4Innovations, 17. listopadu 2172/15**,** 708 00 Ostrava-Poruba**,** Czech Republic