SIEMENS
Ingenuity for life



Siemens Digital Industries Software

Comparing experimental and computational results

Simcenter FLOEFD predicts transition to turbulence in flat plate channel flow

Executive summary

The combination of computational fluid dynamics (CFD) simulations and laboratory measurements is essential for the successful development of cutting-edge equipment. APEX-Research uses Simcenter™ FLOEFD™ CFD software in the development of high-performance heat exchangers. The company also operates a modern laboratory with state-of-the-art measurement equipment. This paper describes how APEX-Research used data from laboratory measurements to validate and calibrate the results of Simcenter FLOEFD simulations.

By Mircea Dinulescu, Founder
Jens Kitzhofer, R&D Manager
APEX Group

Comparing experimental and computational results

APEX Group specializes in developing, marketing and manufacturing high-performance heavy-duty equipment for heat recovery and gas handling projects. APEX Group's main products are plate-type and tubular heat exchangers for large volumes of gases, in petrochemical, power, metallurgical, pulp and paper, cement, DeNOx and general environmental applications. For corrosion protection, heat exchangers with glass-enameling protection are available. A typical example with the dimensions of 6 x 5 x 8 meters is shown in figure 1 for a combustion air preheating system. The use of these heat recovery systems is mandatory for plants with long operating times and high thermal output. Each unit is designed according to the customer's specifications in terms of pressure drop, heat transfer performance and space constraints.



Figure 1: Heat exchanger for combustion air preheating.

APEX-Research B.V. in the Netherlands has a modern laboratory equipped with state-of-the-art measurement equipment such as advanced laser equipment (particle image velocimetry and laser doppler anemometry). The combination of CFD simulations and laboratory measurements is essential for the successful development of cutting-edge equipment. One component of the applied software package is the CFD software Simcenter FLOEFD.

A basic design of a plate-type heat exchanger system is shown in Figure 2, which consists of one pass in z-configuration and one pass in x-configuration. The cold air first enters the x-configuration heat exchanger and is redirected into the z-configuration heat exchanger. The air travels through the heat exchanger from the bottom to the top, where it leaves the heat exchanger preheated. The hot flue gas (for example from a combustion process) is directed in the opposite direction. The inlet is at the top of the z-configuration heat exchanger, the flow direction is from top to bottom and the outlet of the cooled flue gas is at the bottom of the x-configuration heat exchanger, which is frequently described as the cold end.

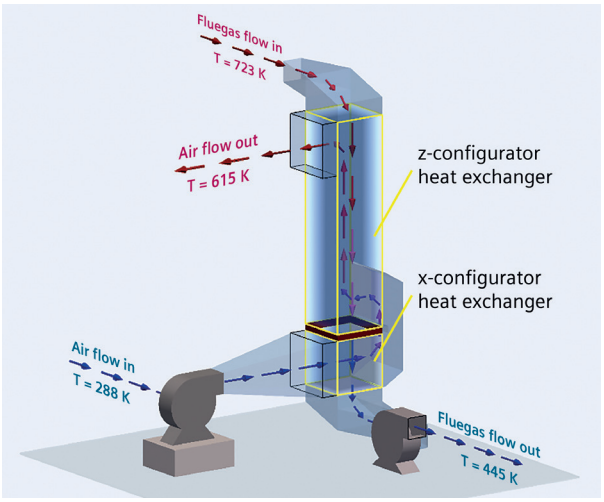


Figure 2: Basic design overview.

The heat is exchanged between the hot flue gas and the cold fresh air through metal plates of different thicknesses (figure 3) depending on the application. The system must be gas-tight; the two gas streams may not mix. A modular design is shown in figure 4.

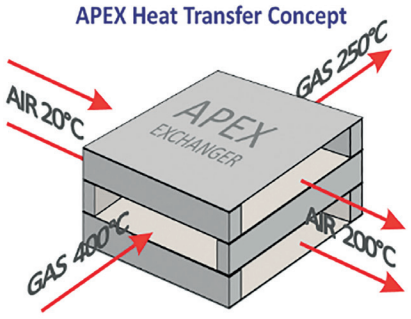


Figure 3: Principle heat exchange metal plates.

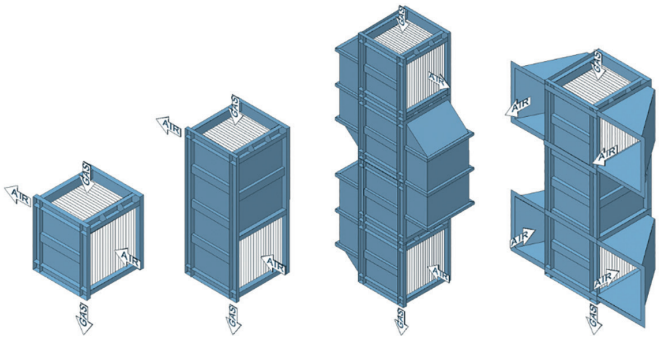


Figure 4: Modular design.

Depending on the project specifications, there are numerous geometrical and fluid-mechanical parameters that influence each other. The channel height can vary, for example between 6 and 20 millimeters. Other geometrical parameters include cross-section area, number of channels or flow configurations. Fluid- and thermo-mechanical properties are volume flow rate, Reynolds number, gas compositions or inlet and outlet temperatures. All of these variables result in a variety of possible design configurations that make it almost impossible to establish experimental or empirical data for each individual parameter variation.

A good pressure drop to heat transfer relation is crucial for these applications to save energy or to fit into existing plant configurations.

The design criterion often demands that the flow must be in the lower turbulent Reynolds number range (figure 5). A higher velocity allows a higher heat transfer, but at the same time, the pressure drop increases rapidly proportional to the square of the velocity. Therefore, it is important to find the appropriate design compromises.

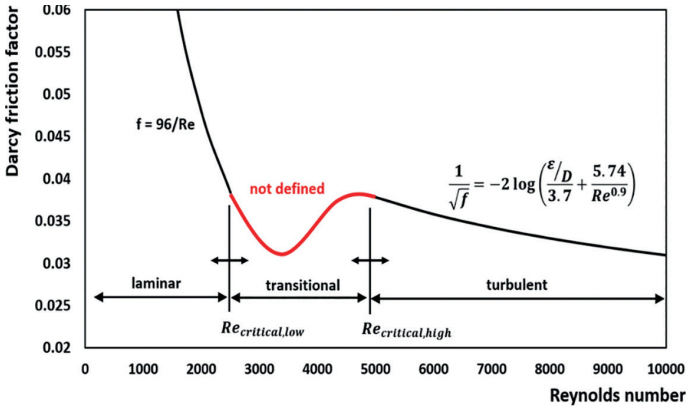


Figure 5: Reynolds number ranges.

A large variety of possible configurations (figure 6) for the gas flow arrangement must be investigated with regard to the pressure drop, heat transfer performance and recirculation zones. For one of these configurations, the highlighted z-configuration, APEX-Research B.V., a company of APEX Group, conducted a calibration between simulation and measurement.

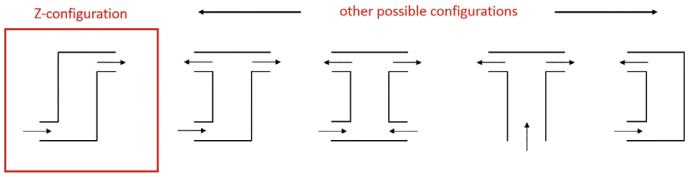


Figure 6: Possible configurations for gas flow arrangement.

A detailed description of the flow conditions in a z-configuration channel is shown in figure 7. The blue arrow indicates the air inlet flow. Two recirculation areas are generated at the first bend downstream of the inlet, followed by a specific velocity profile and finally by the same recirculation areas in the second bend as at the first bend. The flow through the z-configuration channel is associated with a certain pressure drop. On the other side of the channel (below and above, not shown in the figure) flows the hot flue gas. The highest heat output is achieved with straight counter-flow, and as such, cross-flow areas should be minimized.

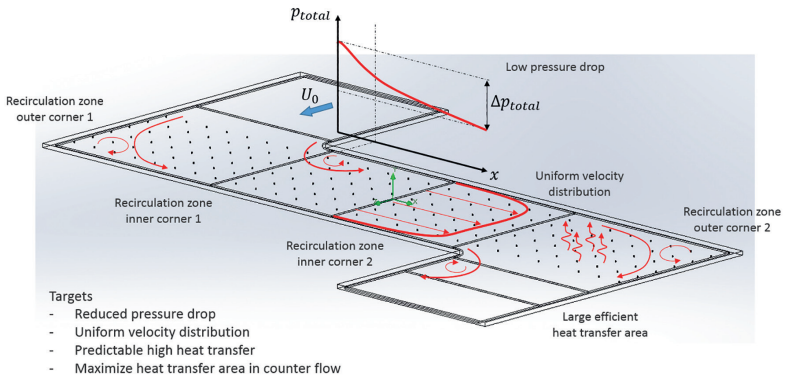


Figure 7: Flow conditions in a channel.

APEX-Research established an experimental setup to validate the simulation results and to calibrate the CFD software for the application of plate-type flow configurations. The experimental set up is shown in figure 8. The scaled 1:1 model is made of plexiglass for optical access. A centrifugal fan is placed at the inlet. The flow is guided through the diffuser duct and the distribution duct to enter into the z-configuration channel. The

measurements for volume flow rate, velocity and turbulence distribution are carried out by laser doppler anemometry (LDA) and pressure drop measurements are collected by static pressure sensors.



Figure 8: Experimental setup.

For the Simcenter FLOEFD simulations, a 1:1 scaled z-configuration example was used, with symmetric conditions and the use of half of the model (figure 9). Approximately 4.3 million cells were created by the automatic mesh generation to have a high spatial resolution in the computational domain. The boundary conditions for the simulation were set with respect to the measured values in the physical values.

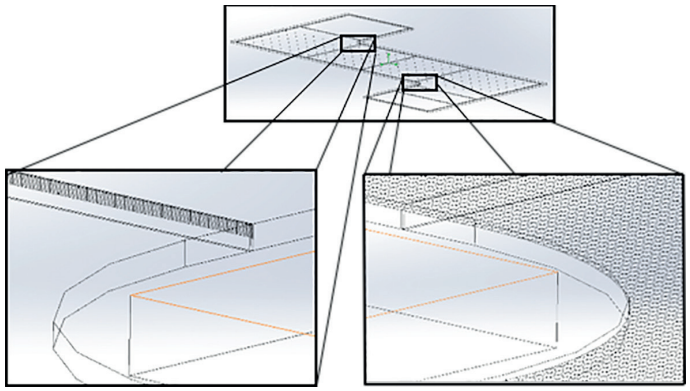


Figure 9: Simcenter FLOEFD simulation model.

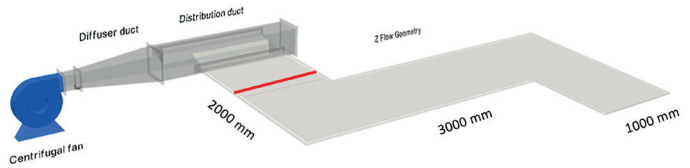


Figure 10: Physical flow model.

The volume flow rate, the static pressure distribution, and the velocity as well as the turbulence intensity profile were measured in the physical flow model, see figure 10. Figure 11 (a) shows the measured streamwise velocity in the center of the channel for three different Reynolds numbers (lower turbulent Reynolds number range) as a function of width (along the red line in figure 10). Figure 11 (b) shows the measured turbulence intensities in the center of the channel as a function of width. Figure 11 (c) shows the velocity profile as a function of channel height. The integration of the velocity along the height, and following integration over the width, yields the volume flow rate.

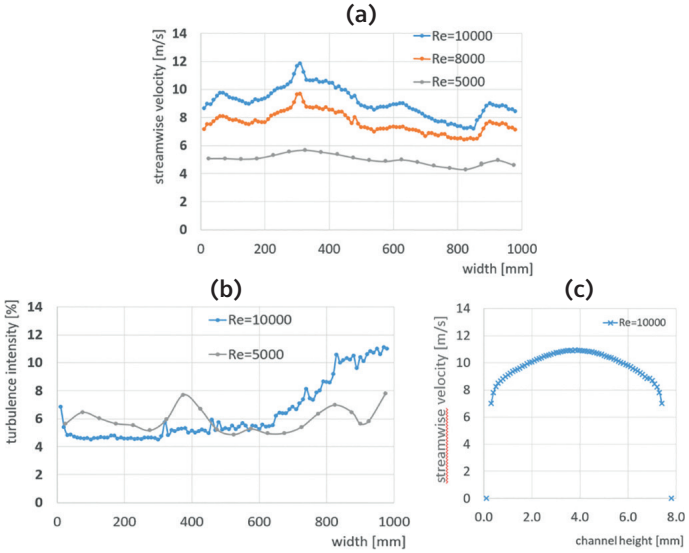


Figure 11 (a,b,c): Measurement results.

"We at APEX-Research successfully used Simcenter FLOEFD for many years and now we have the increased confidence in its results through our own modeling experiment."

Mircea Dinulescu
Founder
APEX

The Simcenter FLOEFD results were compared with the experimental data. One of the main findings was the importance of the correct boundary condition definition. The centrifugal fan and intake geometry generate an inhomogeneous velocity profile at the inlet. The definition of the real velocity profile and turbulence intensity, measured in the laboratory, significantly increased the simulation results accuracy. Figure 12 shows the comparison with respect to overall total pressure drop. The deviation decreased from approximately 10 percent to 2 percent, only on the basis of the realistic boundary conditions definition.

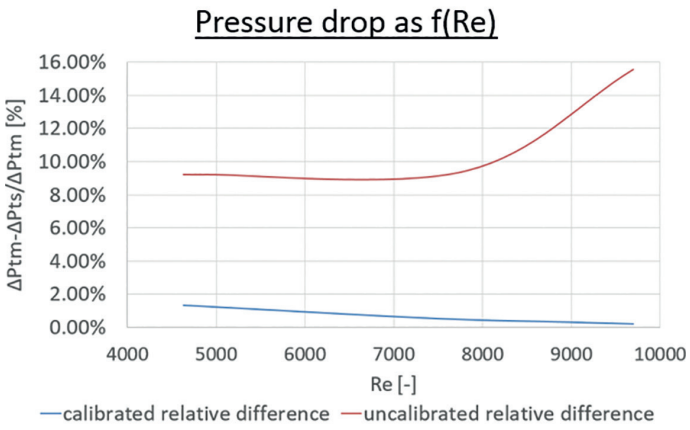


Figure 12: Deviation comparison for different boundary condition definitions.

In figure 13, some comparisons of the test results with the simulation at different measuring positions are presented for the overall static pressure distribution. The good agreement between simulation and experiment can be seen by the identical color coding.

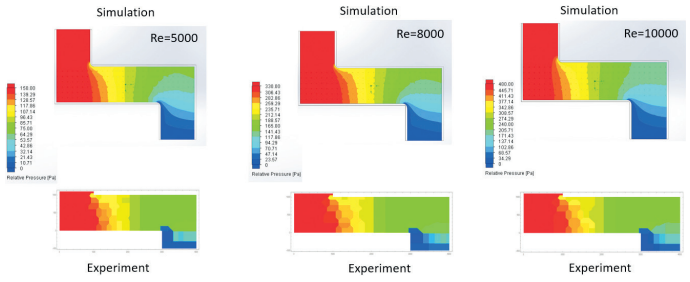


Figure 13: Comparison of test results with simulation results.

Also, the comparison of exact values of the static pressures at identical positions (red line in figure 14) shows a good agreement between simulation and experiment.

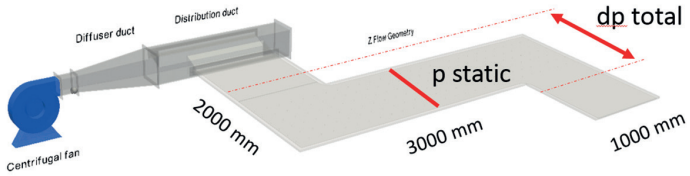


Figure 14: Position for static pressure measurement.

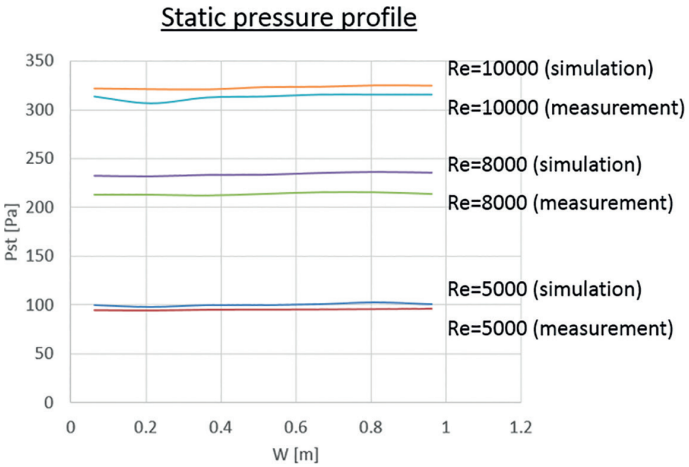


Figure 15: Comparison of measurement and simulation.

The comparison of the velocity profile is shown in figure 16 for different Reynolds numbers at the position of the red line, which crosses the recirculation area in the inner corner. The simulation results of the flow fields are very close to the experimental results. This allows the reliable prediction of flow field characteristics like pressure drop or flow uniformity in plate-type heat exchangers with the calibrated simulation.

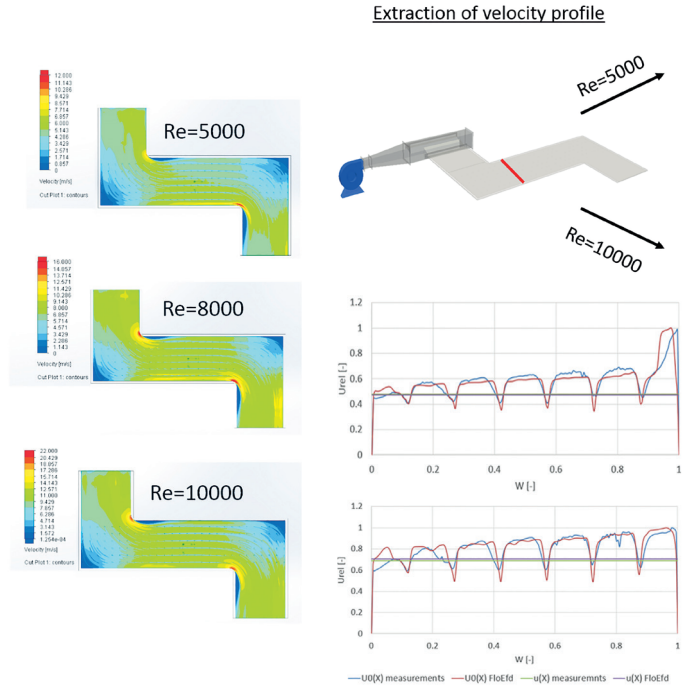


Figure 16: Comparison of the velocity profile.

The simulation results are confirmed by the measurement results, using state-of-the-art measurement technology such as laser doppler anemometry, particle image velocimetry (PIV) and static pressure measurements. As a result of the confidence achieved by the calibration of simulation with experimental data, numerous configurations can be simulated. With Simcenter FLOEFD many different geometries can be investigated according to specific project specifications, such as heat transfer output, pressure drop limitations and space requirements. The optimized configuration can be designed for production with the collaboration of simulation and experimental measurements.



"I appreciate the straightforward and fast simulations with Simcenter FLOEFD."

Dr.-Ing. Jens Kitzhofer
R&D Manager
APEX-Research B.V.

References

1. www.apexgroup.eu
2. https://www.apexgroup.eu/admin/uploads/_pdf/Short_Note_CFD.pdf
3. https://www.dantecdynamics.com/laser-doppler-anemometry

Siemens Digital Industries Software

Headquarters
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 972 987 3000

Americas
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 314 264 8499

Europe
Stephenson House
Sir William Siemens Square
Frimley, Camberley
Surrey, GU16 8QD
+44 (0) 1276 413200

Asia-Pacific
Unit 901-902, 9/F
Tower B, Manulife Financial Centre
223-231 Wai Yip Street, Kwun Tong
Kowloon, Hong Kong
+852 2230 3333

About Siemens Digital Industries Software

Siemens Digital Industries Software is driving transformation to enable a digital enterprise where engineering, manufacturing and electronics design meet tomorrow. Our solutions help companies of all sizes create and leverage digital twins that provide organizations with new insights, opportunities and levels of automation to drive innovation. For more information on Siemens Digital Industries Software products and services, visit siemens.com/software or follow us on LinkedIn, Twitter, Facebook and Instagram. Siemens Digital Industries Software – Where today meets tomorrow.

siemens.com/software

© 2020 Siemens. A list of relevant Siemens trademarks can be found here.
Other trademarks belong to their respective owners.

81717-C3 4/20 H