Comparison between **Experimental** & **Computational** Results

For The Transition to Turbulence in Flat Plate Channel Flow

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

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 environment applications. For corrosion protection, heat exchangers with glass-enameling protection are available. A typical example, with the dimensions of 6 x 5 x 8 m 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 individual unit is designed according to the customer's specifications in terms of pressure drop, heat transfer performance and space constraints.

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 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.

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 has to be gas-tight; the two gas streams may not mix. A modular design is shown in Figure 4.

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 mm. Other geometrical parameters are: 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 these variables result in

> "We at APEX-Research successfully used FloEFD for many years and now we have the increased confidence in its results through our own modeling experiment
>
> **Mircea Dinulescu, APEX Group Founder**

Figure 1. Heat Exchanger for combustion air preheating

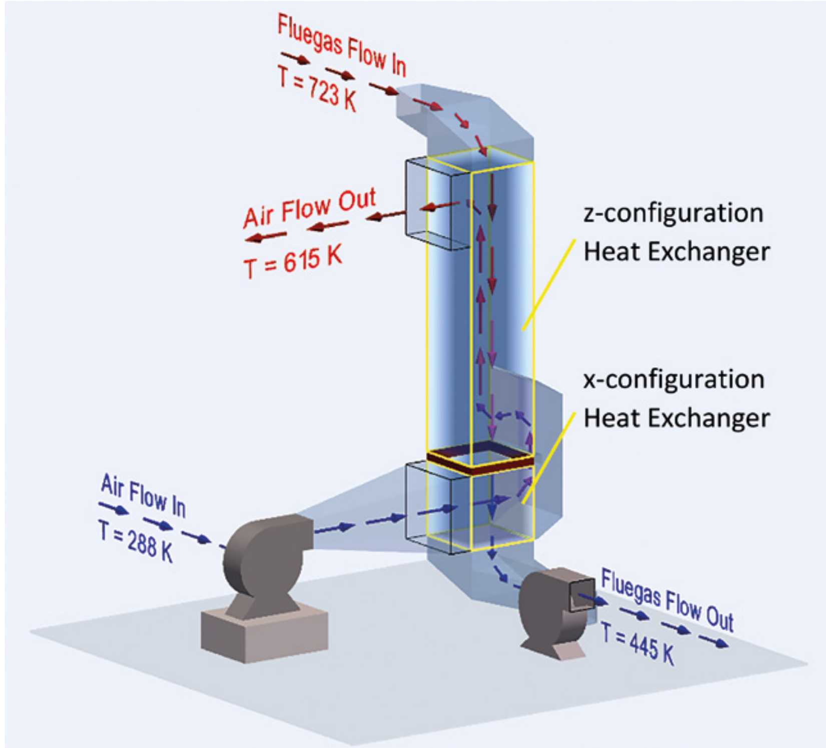

Figure 2. Basic design overview

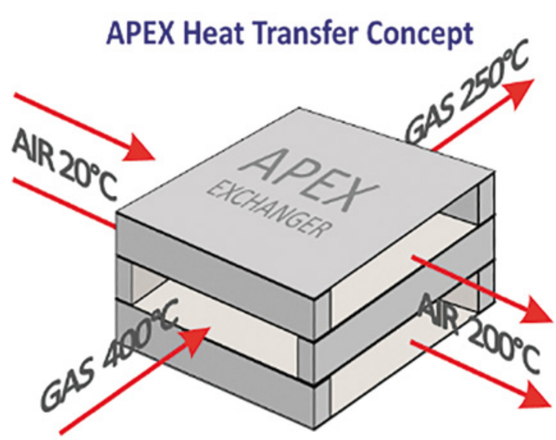

Figure 3. Principle heat exchange metal plates

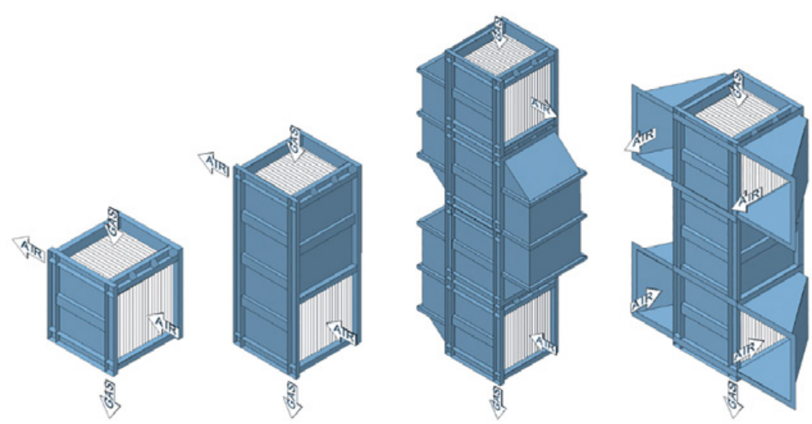

Figure 4. Modular design

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 to the velocity. Therefore, it is important to find the appropriate design compromises.

> I appreciate the straightforward and fast simulations with FloEFD
>
> **Dr.-Ing. Jens Kitzhofer, R&D Manager**
> **APEX-Research B.V.**

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.

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 appeared at the first bend. The flow through the z-configuration channel is associated with a certain pressure drop. On the other side of

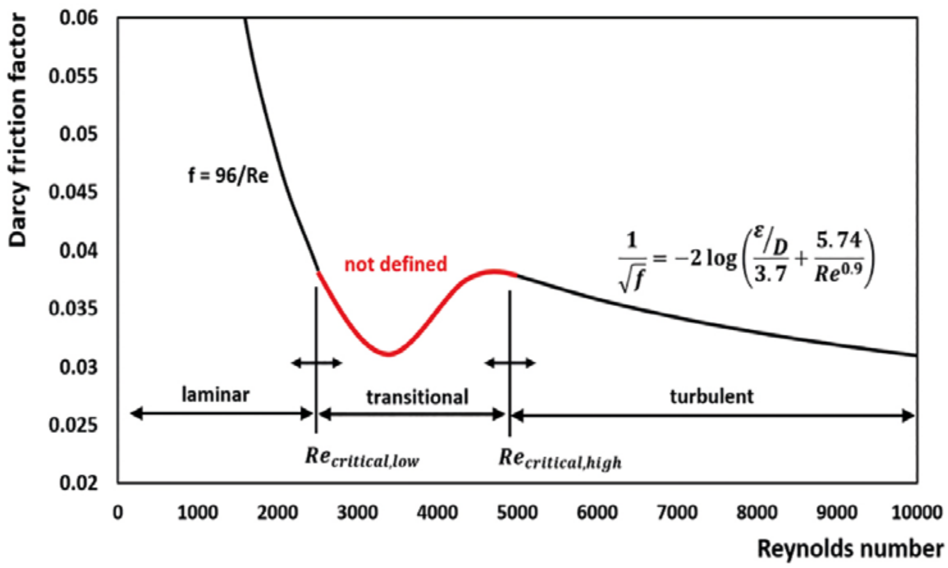

Figure 5. Re number ranges

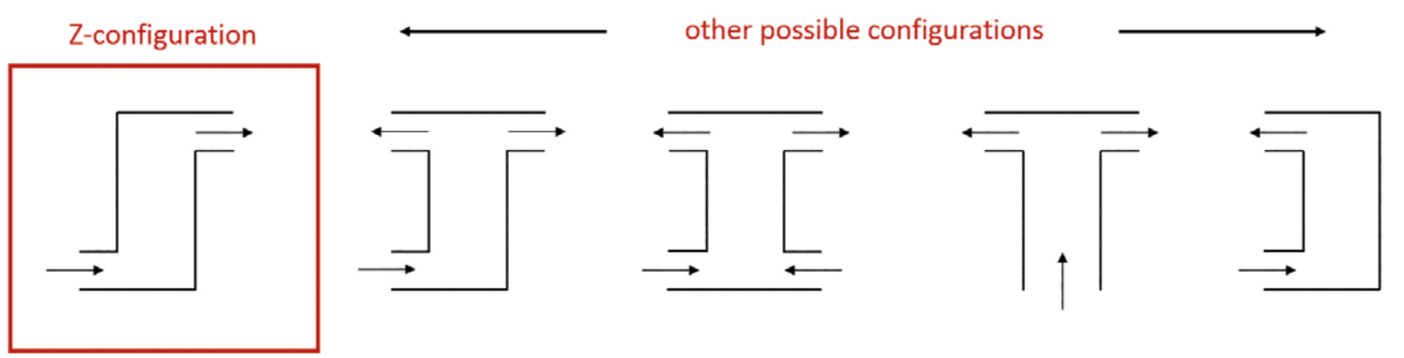

Figure 6. Possible configurations for gas flow arrangement

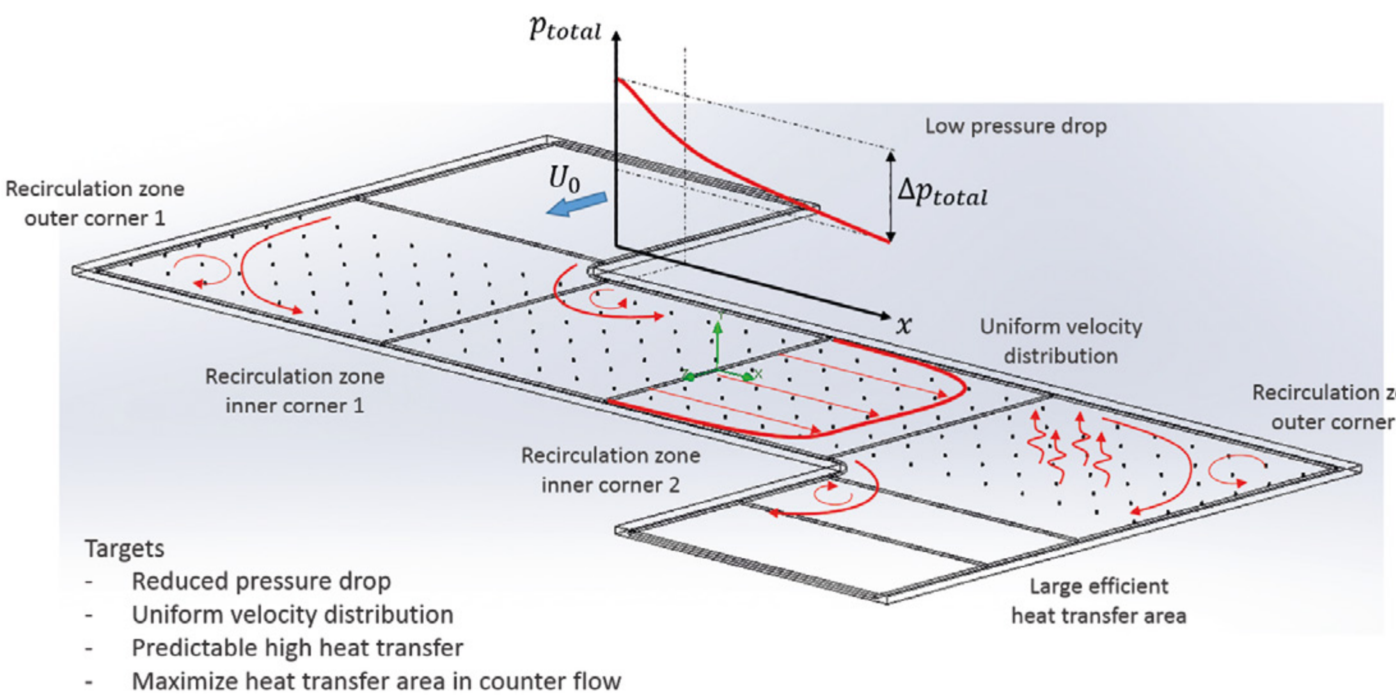

Figure 7. Flow conditions in a channel

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.

APEX-Research established an experimental set-up 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 Plexiglas 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.

For the FloEFD simulations, a 1:1 scaled z-configuration example was used, with symmetric conditions and the use of half of the model, see 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.

The volume flow rate, the static pressure distribution, and the velocity as well as the turbulence intensity profile were measured

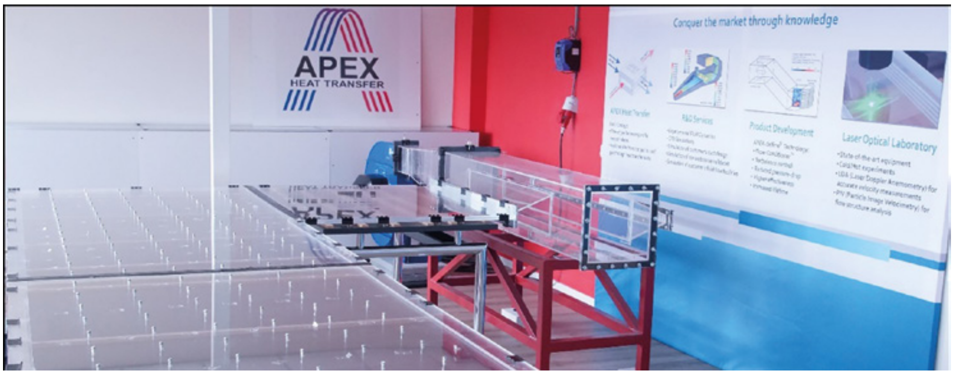

Figure 8. Experimental set-up

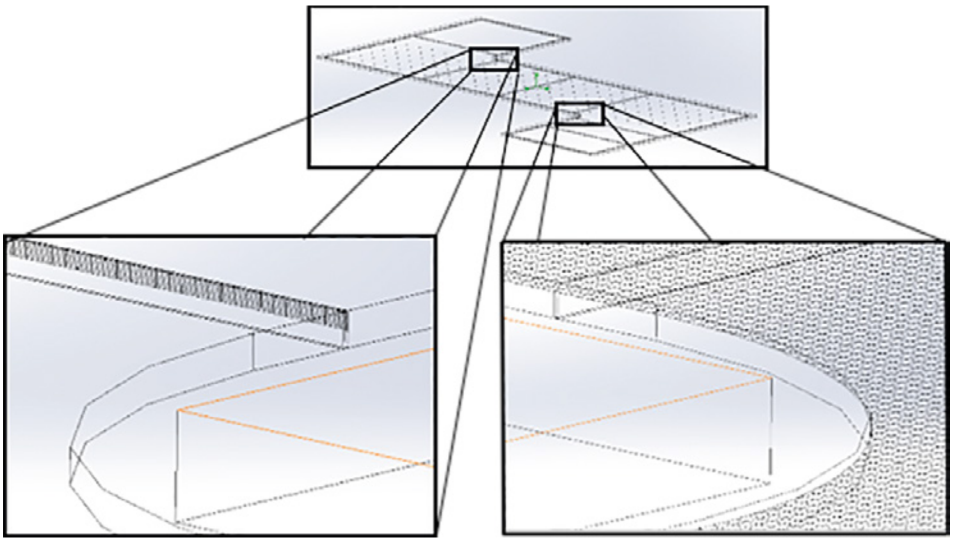

Figure 9. FloEFD simulation model

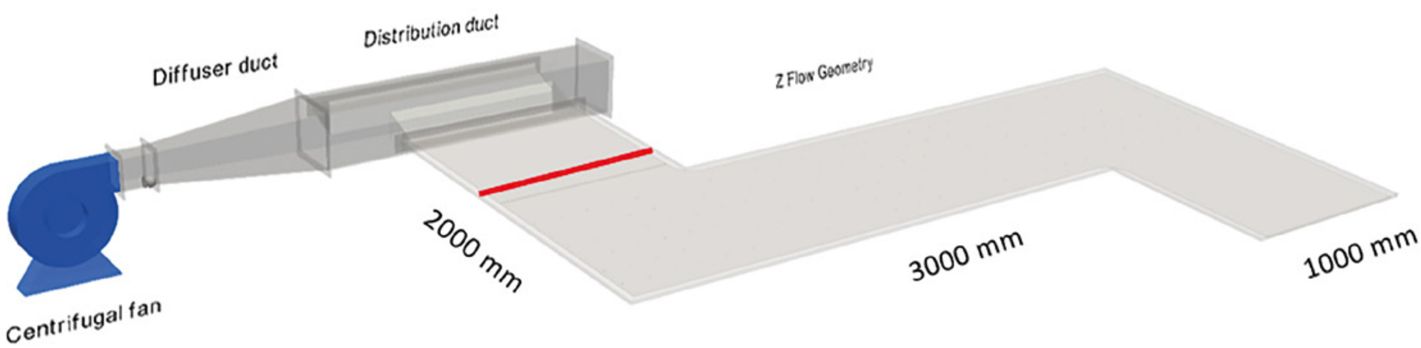

Figure 10. Physical flow model

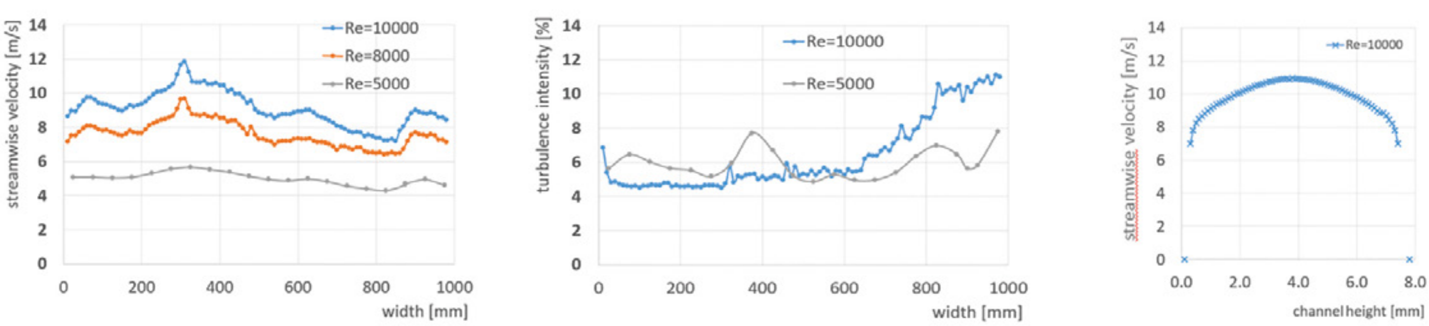

Figure 11a, b, c. Measurement results in the physical flow model, see Figure 10. Figure 11a shows the measured streamwise velocity in the center of the channel for three different Reynolds numbers (lower turbulent Reynolds number range) as a function of width (along the red line in Figure 10). Figure 11b shows the measured turbulence intensities in the center of the channel as a function of width and Figure 11c 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.

The 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 % to 2 %, only on the basis of the realistic boundary conditions definition.

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.

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.

The comparison of the velocity profile is shown in Figure 16 for different Re numbers at the position of the red line, which crosses the recirculation area in the inner corner. The simulation 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 the plate-type heat exchangers with the calibrated simulation.

The simulation results are confirmed by the measurement results, using state of

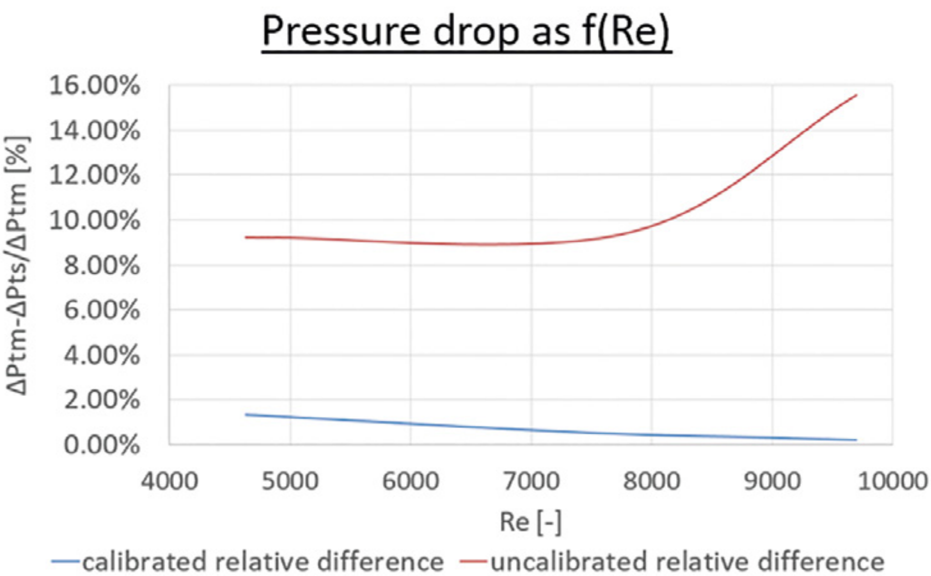

Figure 12. Deviation comparison for different boundary condition definitions

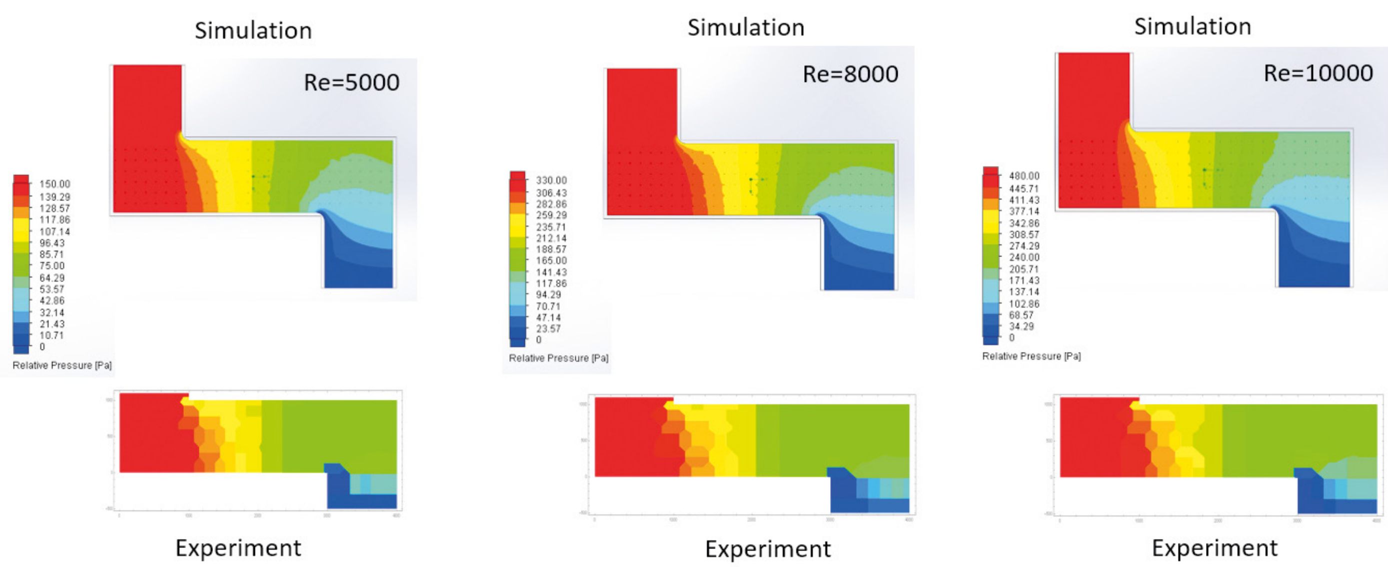

Figure 13. Comparison of test results with simulation results

the art technology for measurement, such as Laser Doppler Anemometry (LDA), 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 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.

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

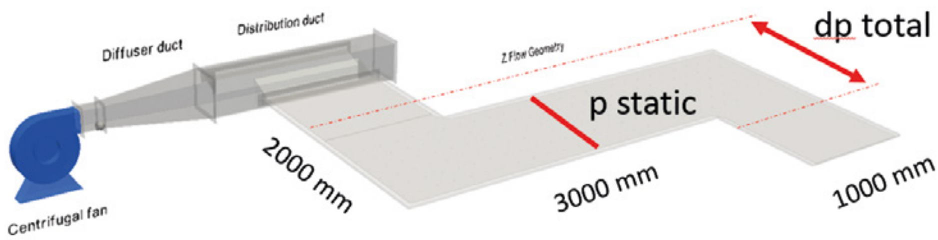

Figure 14. Position for static pressure measurement

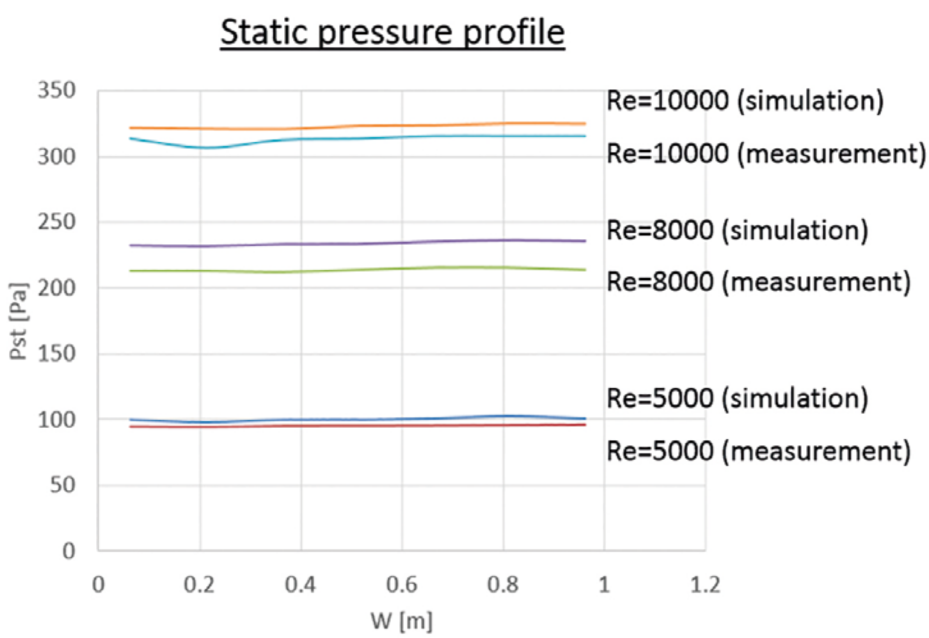

Figure 15. Comparison of measurement and simulation

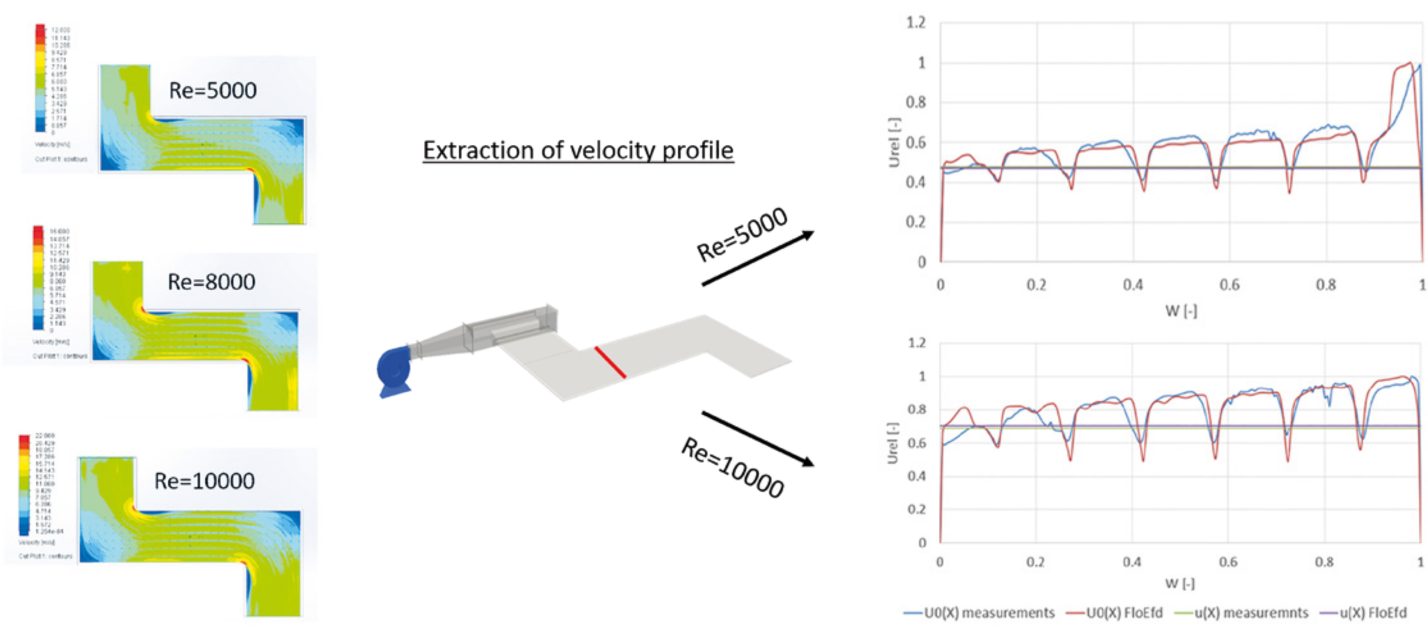

Figure 16. Comparison of the velocity profile

Mentor®
A Siemens Business

ENGINEERING EDGE

Accelerate Innovation with CFD & Thermal Characterization

This article originally appeared in Engineering Edge Vol. 7 Iss. 1

Download the latest issue:
www.mentor.com/products/mechanical/engineering-edge

You might also be interested in...

Thermal Simulation of a SYN TRAC Powerpack

Edge Devices are Hot for IoT

Distributed Chilled Water Pumping AC System

Reliability Evaluation of Sintered Copper Die Bonding Materials

Virtual Reality and Modeling Tools

Ask the GSS Expert

Simulation of Hydraulic Servo Gimbal Control of a Space Launch Vehicle

How to… Convert CAD Geometry into a FloMASTER Sub-System

A Study of Aerothermal Loads

Development of a Turbocharger with FloEFD

Risk Prevention & Energy Saving in Ships

Interview: Takuya Shinoda, Denso Corporation

Three Thermal Simulation & Test Innovations for Electronics Equipment Design

Series Car Brake Cooling

Liquid Cooling Turned on its Side

Thermal Analysis of an ADAS Camera

Thermal Design Leading the Charge

Understanding Die Attach Thermal Performance

Rapid Beverage Cooling Analysis

Bitcoin Mining: A Thermal Perspective

Designing High Efficiency, Low-Cost Synchronous Reluctance Motors

From Measurements to Standardized Multi-Domain Compact Models of LEDs

A Cool Emulator

Achieving Precision in 1D Dynamic Models of Hydraulic Servo Valves

Using FloEFD as an Engineering Tool

Comparison between Experimental & Computational Results

Thermal Interface for Pluggable Optics Modules

Geek Hub: What do Engineers see in Coffee Grounds?