FACULTY OF MECHANICAL ENGINEERING
UNIVERSITY OF WEST BOHEMIA

University of West Bohemia » Department of Power System Engineering

# NUMERICAL INVESTIGATION OF THE FLOW BEHAVIOR INTO THE INLET GUIDE VANE SYSTEM (IGV)

**Publication was supported by project: „Budování excelentního vědeckého týmu pro experimentální a numerické modelování v mechanice tekutin a termodynamice“**

**Project registration number: CZ.1.07/2.3.00/20.0139**

Ing. Roman GÁŠPÁR

Ph.D. candidate

Univerzitní 22, 306 14, Pilsen

gaspar@kke.zcu.cz

evropský sociální fond v ČR
EVROPSKÁ UNIE
MINISTERSTVO ŠKOLSTVÍ, MLÁDEŽE A TĚLOVÝCHOVY
OP Vzdělávání pro konkurenceschopnost
INVESTICE DO ROZVOJE VZDĚLÁVÁNÍ

# TABLE OF CONTENTS

# FIGURES

# TABLES

# 1 CFD – COMPUTATIONAL FLUID DYNAMICS

Computational Fluid Dynamics (CFD) is a method for simulating behavior of the fluid flow or the system involving heat transfer, radiation, and so on. This method uses computers for solving special equations over a region of interest with known conditions on the boundaries.

The first mentions about CFD were on around 1910; more interest in CFD began to show after the Second World War, however, more practical use came up with the expansion of computer technology at the end of the 80s and early 90s. Since these times CFD has been more or less on a theoretical level.

Nowadays, the CFD in one of the basic design tools helping to reduce the design time and to increase effectivity of engineering work. CFD is widely used on fields of power and energy, engine industry, aerodynamics, thermodynamic, aeronautics etc.

## 1.1 MATHEMATICAL PRINCIPLE OF CFD

It is necessary to solve several equations, which describe processes of momentum, heat end mass transfer etc., to describe and predict a flow behavior. These equations are known as Navier-Stokes equations. These equations are defined from the mathematical point of view as partial differential equations. There were derived in the early nineteenth century. These equations can be discretized and solved numerically.

There are a lot of different methods to solve these equations and there are several codes. The solution presented here has been achieved by the commercial software ANSYS and its particular parts. The main solver, which solves equations mentioned above, is called CFX and it's based on the finite volume technique.

In case of the finite volume technique the investigated region is divided to small regions (volumes). The equations mentioned above are discretized and solved iteratively for each of these small volumes. Nowadays complicated industrial computations include 10, 20 or more than 50 mil. volumes, depending on the case.

## 1.2 SOLVING A CFD PROBLEM

From the global point of view there are 5 steps to reach data and solution from CFD simulation:

1) Defining the geometry of region of interest – In this step it is necessary to determine the region of interest. Modeling big regions or volumes is very time consuming. Before the start it must be clear, which effects are significant on the investigated volume and which effects are not. It is necessary to have basic knowledge and vision about the flow behavior before starting the simulation. Based on this knowledge it determines necessary simplifications. After the volume creation it creates surface boundary names.

2) Setting and generating mesh – As described above, to solve partial differential equations in the investigated region it must be divided in small control volumes (also called cells). A region composed by these volumes is called MESH. There are a lot of qualitative or quantitative criteria of mesh evaluation, but the mesh generation process is more or less automatic. A user just sets the mesh generation settings criteria, which are necessary from the solved problem point of view, e.g. inflation layer, max number of cells, type of surface mesh, type of volume mesh, maximum skewness etc.
3) Solver setting – During this process the mesh is imported into the solver pre-processor. After the import the physics of the model are defined, e.g. type of the fluid, material properties, thermodynamic properties of the fluid, boundary conditions, e.g. physical properties of the fluid at inlet and outlet, wall properties, reference pressure, and so on. In this step there are defined domain conditions for calculation as well, e.g. rotational/stationary domain, gravity, initial conditions, etc. Very important step during pre-processing is a solver control. During this process a user sets what kind of equation wishes to use, convergence control, convergence criteria or solution back-up control.
4) Solving the case – CFD solver applies partial differential equations and integrates them over all the control volumes. Integral equations are converted to a system of algebraic equations and solved iteratively.
5) Visualizing the results – Habitually used a post-processing software. This software helps to interpret the solution. Main goals of this software are:
    a. Visualize calculated region and mesh (include information about mesh)
    b. Enable to create control points, lines, surfaces, lines, iso-surfaces etc.…
    c. Enable to evaluate values of interest into the entire discretized domain
    d. Visualize contours of interested values (pressure, velocity, temperature field)
    e. Visualize vectors of interested vector variables, e.g. velocity
    f. Enable to visualize volume rendering of selected value
    g. Animations
    h. Export to report

# 2 INTRODUCTION AND MOTIVATION

The hereinafter described CFD investigation is connected with the integrated gas turbine with recycling process. This facility was supervised by the Chair of Thermal Power Machinery and Plants. Besides this investigation at that facility there were several other investigations:

- Performance of power plant process
- Steam injection for high steam mass flow (optimal position, optimum steam parameters, mix with flue gases)
- Impact on flow, cooling, performance and life of the turbine
- Affect behavior and performance of the compressor
- Water treatment (procedures, operating at elevated temperatures)

The aim of the project was CFD (Computational Fluid Dynamics) investigation of The Inlet Guide Vane (IGV) System integrated into the gas turbine. The process of the IGV system numerical simulation is described below step by step. The work started with geometry generation and it follows by generating mesh and setting up calculation. The last part includes important results and conclusions. It also includes a description of the position of the important files, which were forwarded to the host institution.

Before the final calculation described hereinafter, there were done several calculations to improve the accuracy of the calculation. First step was a mesh study. Different mesh sizes were tested to investigate the sensibility of the calculation (Fig. 1). Meshes described in this report are generated according to the mesh, which has 1,737 million of elements. These meshes start with the first boundary layer of the height of 0,0001 [m], and the maximum length of the elements is around 0,0015 [m] in the volumes between blades. This setting represents a good balance between the accuracy and the mesh size.

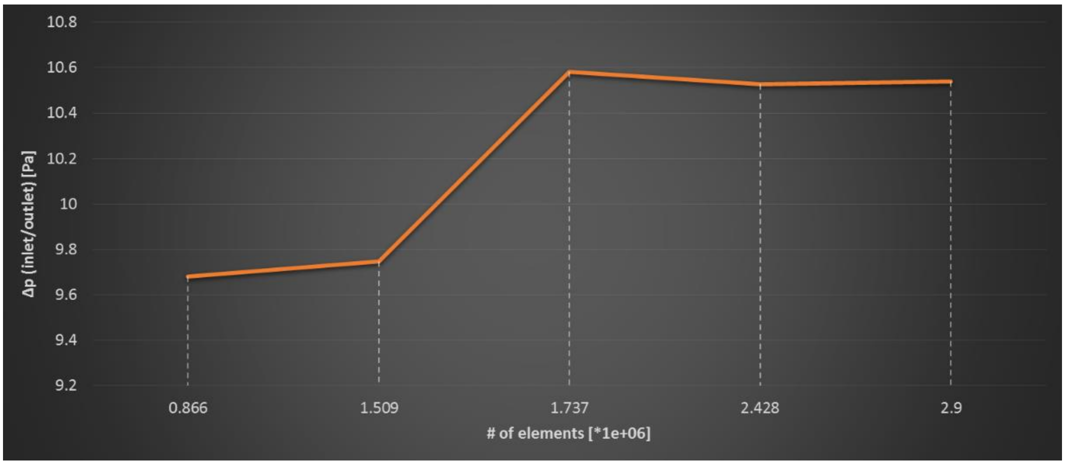


Fig. 1 Results of mesh sensibility study

During the calculation of the entire IGV with the inlet, there was a large separation of the flow at one side of the inlet part observed. The first geometry was generated with sharp edges. These sharp edges at the inlet part were blended with a constant radius of 5 [mm] to prevent a huge separation at the inlet. The effect of this modification is obvious from Fig. 2. All the geometry mentioned below was generated as blended.

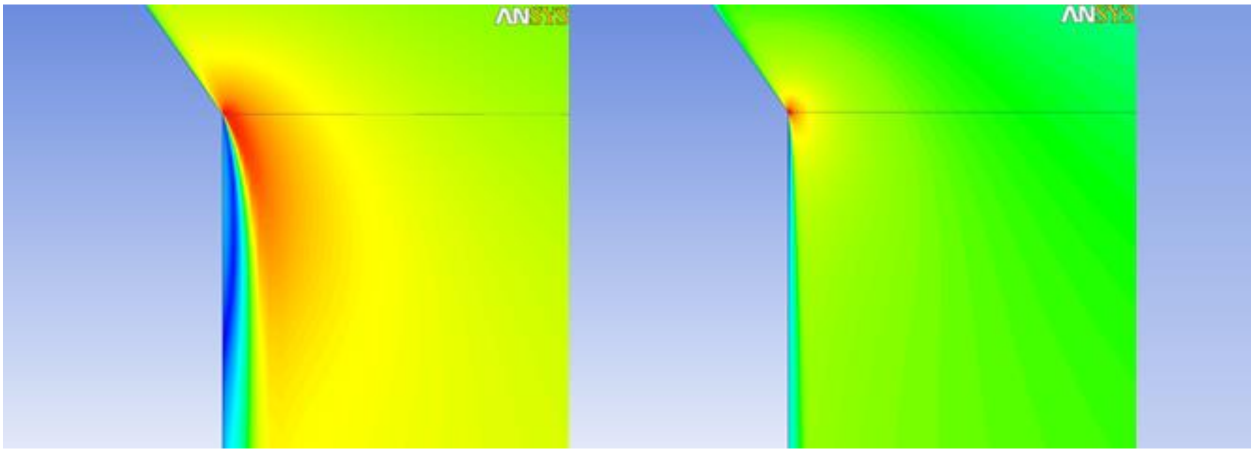

Fig. 2 Flow separation – Sharp edge (left) – Blended Edge (right)

# 3 GEOMETRY

## 3.1 SOLVER AND ARRANGEMENT OF THE FILES

Geometry dimensions were measured at the real IGV system and were implemented into the CAD model. The IGV CAD model has been generated at the ANSYS Design Modeler ver. 14.0. This software with a source of files is able to generate the IGV geometry with different angles of the guide vanes. Source files are located at IGV/GEOMETRY/GEOMETRY_GENERATOR.

## 3.2 FILE NAMING

IGV_0_NC_B

- IGV – indicates that the IGV part was investigated
- 0 – indicates the angle of the blades
- NC – indicates that the computation is without the compressor part (**N**o **C**ompressor)
- B/S – indicates that the inlet channel is **B**lended or **S**harp

## 3.3 GEOMETRY GENERATOR

The purpose of the source file "GEOMETRY GENERATOR.wbpj" is to generate the IGV geometry model for further computation. There are four particular projects generated into the project (Fig. 3).

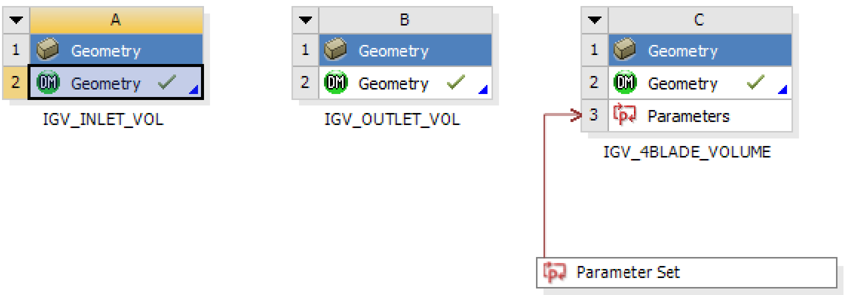


Fig. 3 First view to the "GEOMETRY GENERATOR" project

First two particular projects (IGV_INLET_VOL; IGV_OUTLET_VOL) generate the upper and the lower part of the IGV system (Fig. 4). The geometry is generated with sharp edges, and there is a possibility of further work or modification.

The third project generates the part with the guide vanes (Fig. 5). There is one parameter which is adjustable. This parameter is the angle of rotation of the guide vanes (Fig. 6). There is enough to change a value into the "Parameter Set" (Fig. 3) and Refresh () and Generate (). The geometry on Fig. 5 will be generated automatically.

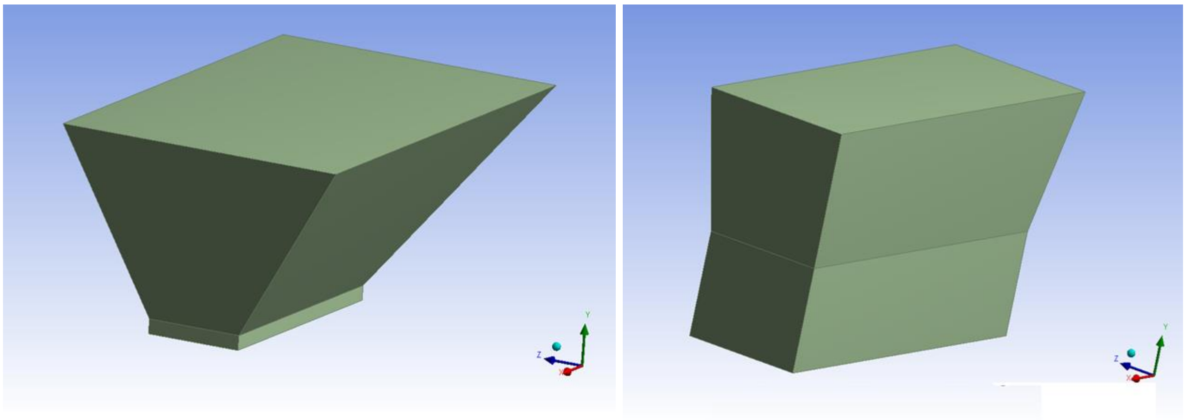

Fig. 4 Generated IGV_INLET_VOL (left) and IGV_OUTLET_VOL (right)

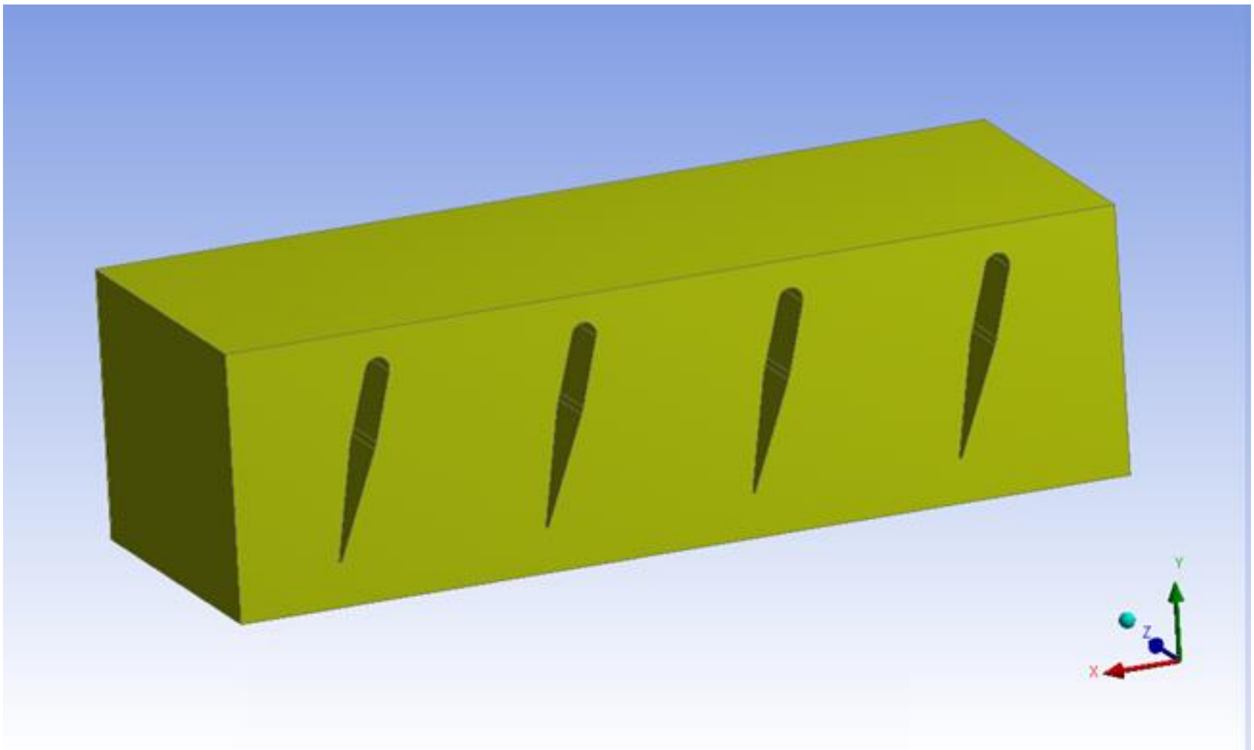

Fig. 5 Generated IGV_4BLADE_VOLUME (15 deg.)

Outline of All Parameters

| | A | B | C | D |
|---|---|---|---|---|
| 1 | ID | Parameter Name | Value | Unit |
| 2 | ⊟ Input Parameters | | | |
| 3 | ⊟ IGV_4BLADE_VOLUME (C1) | | | |
| 4 | P1 | AngleOfBlades | 15 | |
| * | New input parameter | New name | New expression | |
| 6 | ⊟ Output Parameters | | | |
| * | New output parameter | | New expression | |
| 8 | Charts | | | |

Fig. 6 Parameters tab. – Setting angle of blades

Each part was exported as an .igs file (IGV/GEOMETRY/IGS_FILES). These files were used to build an assembly, which is the 4th project in this source file. In this project, there are generated the last modifications on the geometry, and exporting the whole model (Fig. 7).

*Note: For further meshing into ICEM-CFD is recommended to export the whole assembly into .agdb files (IGV/GEOMETRY/AGDB_FILES).*

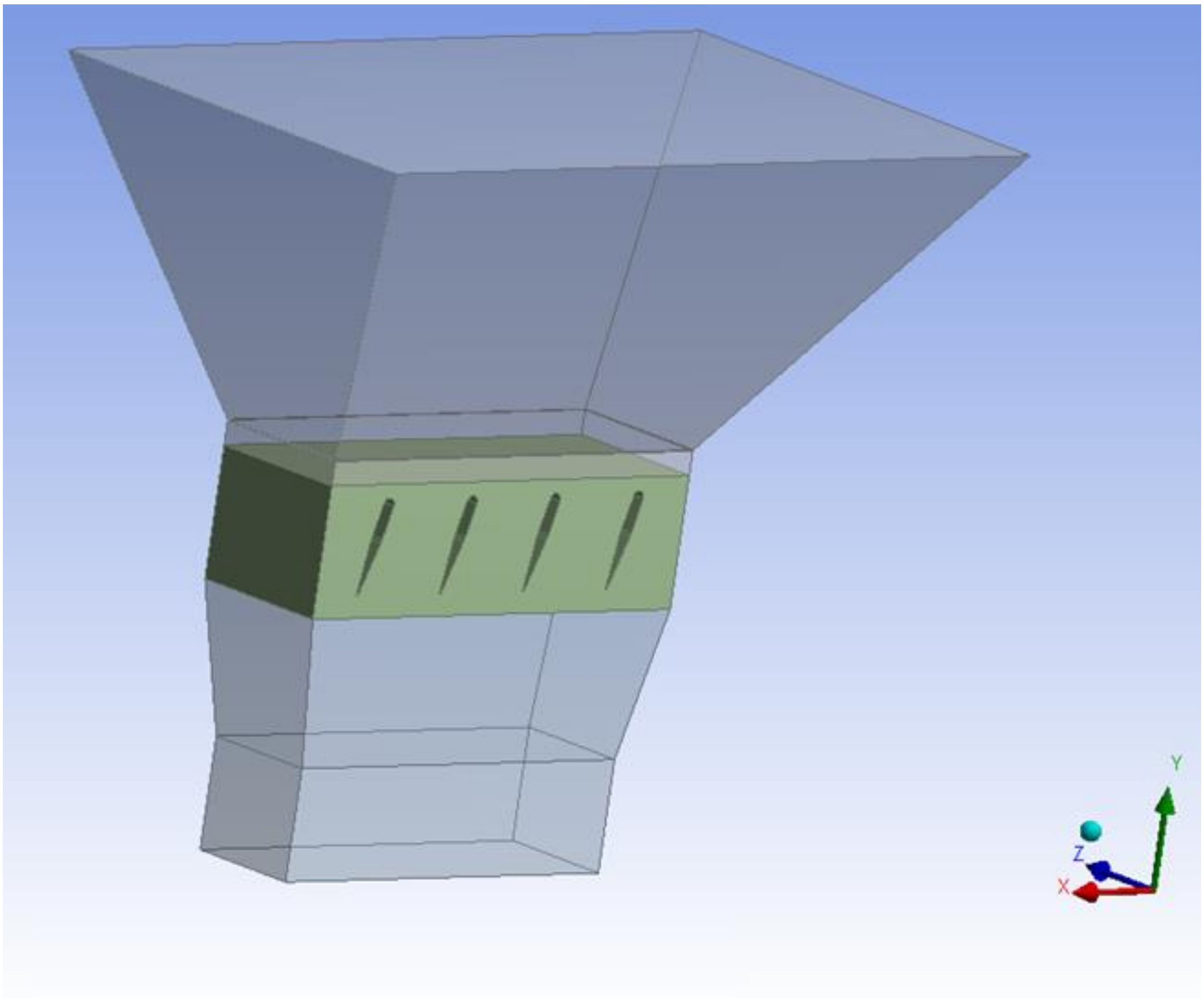


Fig. 7 IGV assembly generation

There were generated several geometries with different guide vane angle settings. There is a geometry for angle: **0°; 15°; 20°; 30°; 36°; 40°; 45°; 50°** and **60°.**

# 4 MESH

## 4.1 SOLVER AND ARRANGEMENT OF THE FILES

All mesh files for the above mentioned cases can be found in IGV/MESH/IGV_CHANNEL_FILES folder. Folders contain ICEM-CFD files, which allow reproducing the entire meshing process. CFX5_FILES folder contains .cfx5 mesh files, which are prepared for the import into CFX-Pre. Meshes were prepared into ICEM-CFD ver. 14.0.

## 4.2 MESH GENERATION

- Mesh for each case was generated separately (Tab. 1).
- All hexahedral with global quality criterion > 0.4
- Tab. 1 shows a number of elements for each case and the global maximum of the $Y^+$ value
- First layer distance 0.0001 [m]
- Growth rate: 1.2
- Additional volumes were created

Tab. 1 Mesh size and Y+ values for different cases

| Version | Mesh size [mil] | Y+ (max) for 5,0 [$kg.s^{-1}$] |
|---|---|---|
| **0_B** | 5.21 | 12.14 |
| **15_B** | 5.90 | 14.10 |
| **20_B** | 5.24 | 14.73 |
| **30_B** | 6.44 | 17.30 |
| **36_B** | 5.17 | 16.45 |
| **40_B** | 5.20 | 21.18 |
| **45_B** | 5.31 | 17.24 |
| **50_B** | 5.44 | 18.33 |
| **60_B** | 8.32 | 23.38 |

According to the Tab. 1, the mesh size was more or less normalized to around 5,5 mil. elements. In case of the versions 60_B and 30_B, there were expected imbalances caused by the flow behavior, and that’s the reason of a higher number of elements.

There were created additional extra volumes. Cases **DO NOT** include a compressor part, so the lower part was replaced by extra volumes during the meshing process (Fig. 8).

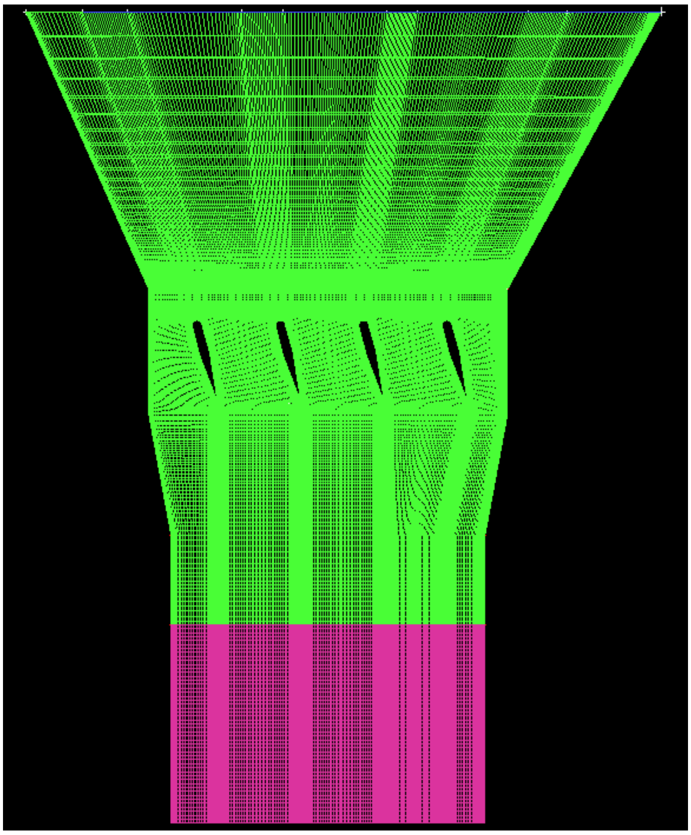

Fig. 8 IGV's mesh (green) with extra volume (purple)

# 5 BOUNDARY CONDITIONS AND CALCULATION

## 5.1 SOLVER AND ARRANGEMENT OF THE FILES

Calculation was provided by ANSYS CFX ver. 14.0. Files for modifying calculations are in IGV/CALCULATIONS/CFX_FILES folder. Prepared .def files to run calculation are situated into IGV/CALCULATIONS/DEF_FILES folder. There are another subfolders inside the folders. Numbers indicate the current mass flow rate of e.g. 5_0, it means 5.0 [$kg.s^{-1}$].

## 5.2 BOUNDARY CONDITIONS AND NUMERICAL SOLVER SETTINGS

- Steady state calculation with a stationary domain
  - Applied a false time step (solver formulation is robust and fully implicit)
- Material: Air Ideal Gas
  - Properties of this fluid is calculated by the equation of state
    - $\frac{p}{\rho} = R.T$
  - *cp =cp(T)*
  - *dh=cp.dT*

### 5.2.1 Fluid models

Basic fluid can be described by three basic equations:

The continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho.\boldsymbol{U}) = 0$$

The momentum equation:

$$\frac{\partial(\rho.\boldsymbol{U})}{\partial t} + \nabla.(\rho.\boldsymbol{U} \times \boldsymbol{U}) = -\nabla p + \nabla\tau + \boldsymbol{S}_M$$

The Total Energy Equation

$$\frac{\partial(\rho\, h_{tot})}{\partial t} - \frac{\partial p}{\partial t} + \nabla.(\rho\, \boldsymbol{U}\, h_{tot}) = \nabla.(\lambda\, \nabla T) + \nabla.(\boldsymbol{U}.\tau) + \boldsymbol{U}.\boldsymbol{S}_M + \boldsymbol{S}_E$$

### 5.2.2 Turbulence model

Shear Stress Transport turbulence model with an automatic wall function is very well fitted to modeling a boundary layer, which was very important in this case, especially because the “WALL” boundary condition was set to “SMOOTH”. This model gives a highly accurate flow separation prediction and free shear flow prediction far from the wall as well. This model is based on a k-$\omega$ model, and the proper transport behavior can be obtained by a limiter to the formulation of the eddy-viscosity.

### 5.2.3 Boundary conditions

After the mesh import, the CFX generates the topology of the calculation domain. The imported mesh is divided into primitive regions (volumes, surfaces). At the beginning, these regions are defined as a part of “Default Domain”. In the next steps it is necessary to define what kind of boundaries are these regions. These boundaries are can be divided to four main types:

- Fluid boundary – Represents boundaries, where the flow passes through (inlet; outlet; open, symmetry)
- Solid boundary – represents the usual wall
- Interfaces
- Fluid - Solid
- Solid – Solid

### 5.2.4 Boundary conditions applied in this case

- Inlet:
    - Subsonic flow regime
    - 4.4 – 5.4 [kg.s$^{-1}$] mass flow rate was tested
    - Medium intensity turbulence (5%)
    - 288 [K] Static temperature
- Wall:
    - No slip and smooth wall
    - Adiabatic heat transfer
- Outlet:
    - Subsonic flow regime
    - Static pressure: – 0.13 [mbar] **RELATIVE TO REFFERENCE PRESSURE**
- Solver:
    - High resolution advection scheme and turbulence numeric
    - 1000 iteration per each case with the auto timescale
    - RSM 1e-3 and conservation target 0.01 was reached
- Additional settings:
    - Pressure drop inside the entire domain and into IGV part were monitored

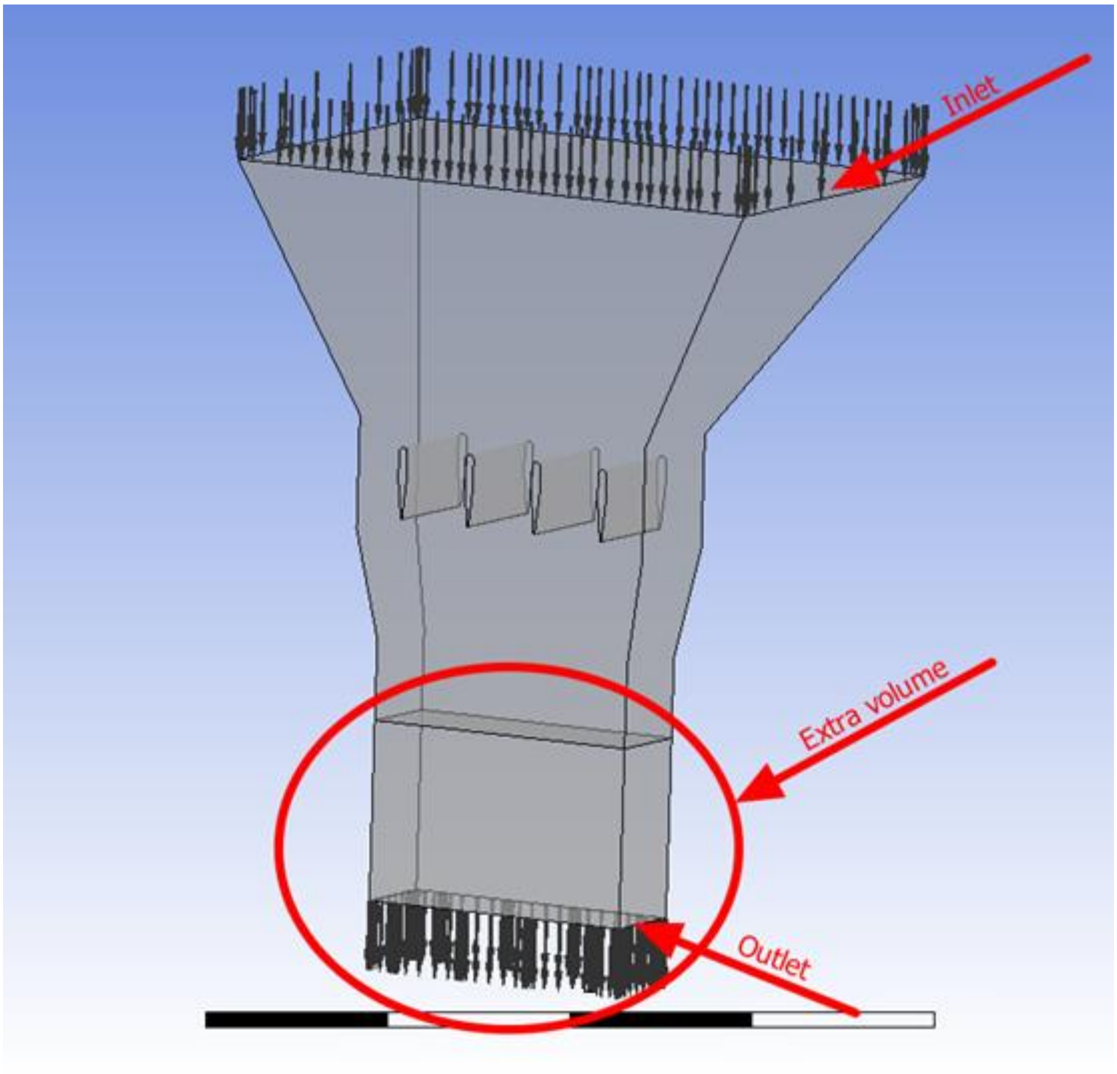


Fig. 9 Boundary conditions for the IGV channel

# 6 RESULTS

## 6.1 SOLVER AND ARRANGEMENT OF THE FILES

The calculation was evaluated by ANSYS CFD-POST ver. 14.0. Result files are situated in IGV/CALCULATIONS/RESULTS folder. There are another subfolders inside the "RESULTS" folder. Numbers indicate the current mass flow rate of e.g. 5_0 it means 5.0 [$kg.s^{-1}$].

## 6.2 INVESTIGATED PARAMETERS

The main goal of this project was to investigate:

- Pressure drop generated by the IGV with a different blade position and with a different mass flow rate.
    - Investigation at the measurement point
    - Investigation at the plane which is situated at the same level as the measurement point is situated
    - Investigation along the wall, where the measuring device is situated
- Investigation of the flow behavior inside the channel

Evaluated areas are displayed in the Fig. 10.

## 6.3 TABLES DESCRIPTION

- The first column (from left) displays which position of IGV blades is investigated 0_B = 0 angle; blended inlet
- The second "Pdiff(manuf.)" part indicates the pressure drop given by the manufacturer
- Pdiff (AreaAve) – Indicates the pressure drop:
    - Area averaged static pressure at "Inlet" - Area averaged static pressure at "Measuring Plane"
- Pdiff (Point) – Indicates the pressure drop:
    - Area averaged static pressure at "Inlet" - Static pressure at "Measuring Point"
- Pdiff (versions) – Indicates the difference of values of the Pdiff (AreaAve) and the Pdiff (Point) between different blade positions.

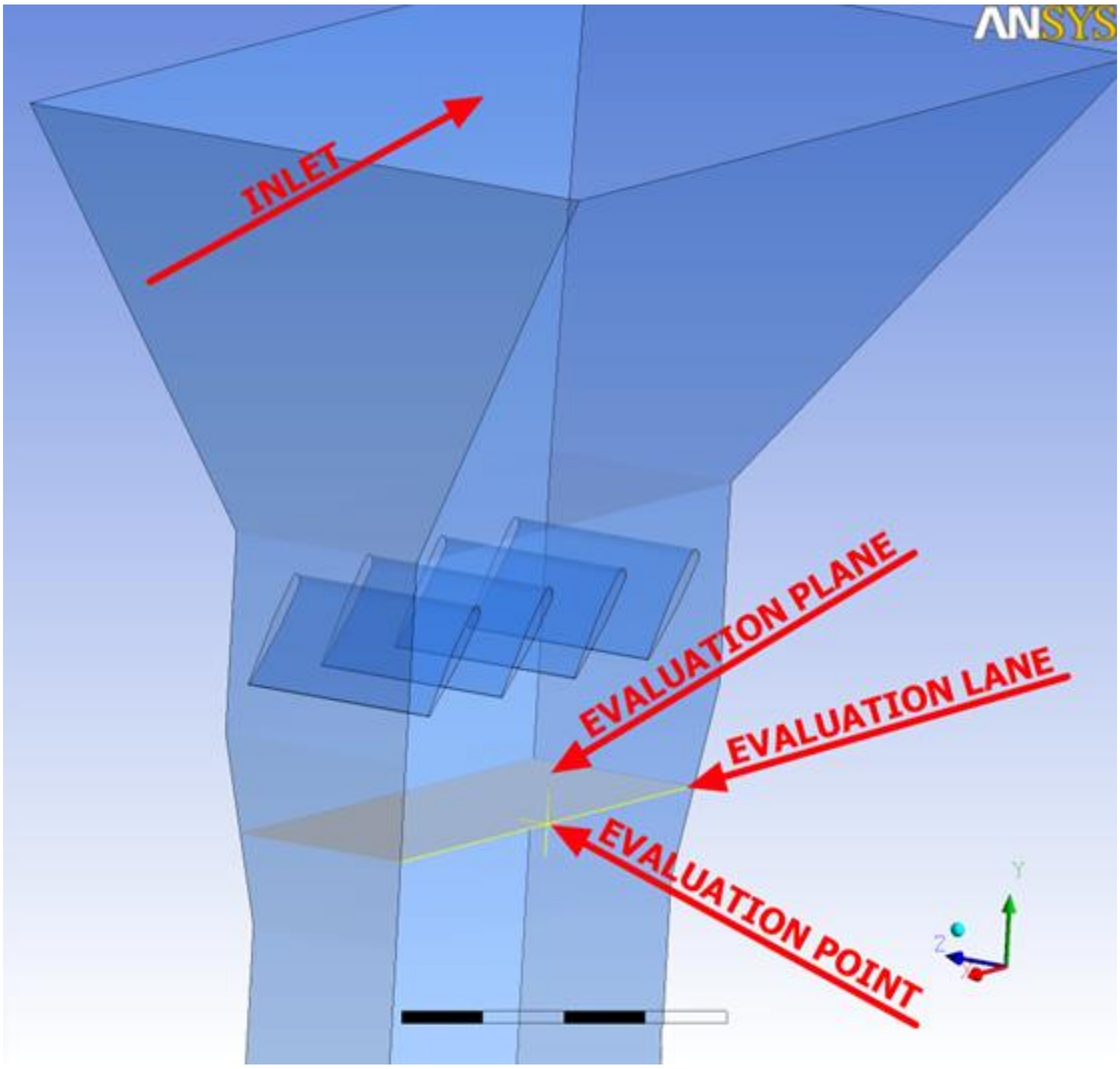


Fig. 10 Evaluated areas

## 6.4 TABLES

Tab. 2 Pressure drop for different blade positions and the mass flow rate (4.4 [kg.s$^{-1}$] – 4.8 [kg.s$^{-1}$])

| | | Mass flow 4.4 kg.s-1 | | Mass flow 4.6 kg.s-1 | | Mass flow 4.8 kg.s-1 | |
|---|---|---|---|---|---|---|---|
| Version | Pdiff _(manuf.) | Pdiff (AreaAve) | Pdiff(Point) | Pdiff (AreaAve) | Pdiff(Point) | Pdiff (AreaAve) | Pdiff(Point) |
| **0_B** | 1.34 | 1.4021 | 1.42218 | 1.53014 | 1.55309 | 1.66396 | 1.68961 |
| **15_B** | 1.55 | 1.64575 | 1.64879 | 1.79804 | 1.81237 | 1.95007 | 1.94791 |
| **20_B** | N/A | 1.83885 | 1.84102 | 2.00998 | 2.0148 | 2.18749 | 2.19093 |
| **30_B** | 2.05 | 2.42762 | 2.31881 | 2.64045 | 2.52514 | 2.86339 | 2.74238 |
| **36_B** | N/A | 3.51415 | 3.2161 | 3.80988 | 3.50871 | 4.12725 | 3.78901 |
| **40_B** | N/A | 3.30638 | 2.99385 | 3.56819 | 3.22817 | 3.86064 | 3.49751 |
| **45_B** | 3.14 | 5.41641 | 5.08174 | 5.90843 | 5.54207 | 6.41838 | 6.0196 |
| **50_B** | N/A | 6.98147 | 6.9592 | 7.62102 | 7.59253 | 8.29828 | 8.26434 |
| **60_B** | 5.9 | 10.2379 | 12.1098 | 11.1884 | 13.1505 | 11.9296 | 13.8525 |

Pdiff (versions)

| | | | | | | | |
|---|---|---|---|---|---|---|---|
| **0_B - 15_B** | 0.21 | 0.24365 | 0.22661 | 0.2679 | 0.25928 | 0.28611 | 0.2583 |
| **0_B - 20_B** | N/A | 0.43675 | 0.41884 | 0.47984 | 0.46171 | 0.52353 | 0.50132 |
| **0_B - 30_B** | 0.71 | 1.02552 | 0.89663 | 1.11031 | 0.97205 | 1.19943 | 1.05277 |
| **0_B - 36_B** | N/A | 2.11205 | 1.79392 | 2.27974 | 1.95562 | 2.46329 | 2.0994 |
| **0_B - 40_B** | N/A | 1.90428 | 1.57167 | 2.03805 | 1.67508 | 2.19668 | 1.8079 |
| **0_B - 45_B** | 1.8 | 4.01431 | 3.65956 | 4.37829 | 3.98898 | 4.75442 | 4.32999 |
| **0_B - 60_B** | 4.56 | 8.8358 | 10.68762 | 9.65826 | 11.59741 | 10.26564 | 12.16289 |

Tab. 3 Pressure drop for different blade positions and the mass flow rate (5.0 [kg.s$^{-1}$] – 5.4 [kg.s$^{-1}$])

| | | Mass flow 5 kg.s-1 | | Mass flow 5.2 kg.s-1 | | Mass flow 5.4 kg.s-1 | |
|---|---|---|---|---|---|---|---|
| Version | Pdiff (manuf.) | Pdiff (AreaAve) | Pdiff(Point) | Pdiff (AreaAve) | Pdiff(Point) | Pdiff (AreaAve) | Pdiff(Point) |
| **0_B** | 1.34 | 1.80503 | 1.83338 | 1.94923 | 1.98018 | 2.10046 | 2.13411 |
| **15_B** | 1.55 | 2.10605 | 2.12867 | 2.27182 | 2.27293 | 2.44151 | 2.46651 |
| **20_B** | N/A | 2.36367 | 2.37749 | 2.53926 | 2.56203 | 2.74258 | 2.75252 |
| **30_B** | 2.05 | 3.09386 | 2.96314 | 3.33095 | 3.20299 | 3.57475 | 3.42825 |
| **36_B** | N/A | 4.46303 | 4.10849 | 4.82391 | 4.42249 | 5.18405 | 4.77926 |
| **40_B** | N/A | 4.17138 | 3.75669 | 4.04043 | 4.04043 | 4.82076 | 4.3258 |
| **45_B** | 3.14 | 6.95675 | 6.52303 | 7.49049 | 7.02345 | 8.07022 | 7.57287 |
| **50_B** | N/A | 8.98576 | 8.94575 | 9.68132 | 9.6357 | 10.4193 | 10.3685 |
| **60_B** | 5.9 | 13.1706 | 15.782 | 13.9831 | 16.656 | 14.9785 | 17.5996 |

Pdiff (versions)

| | | | | | | | |
|---|---|---|---|---|---|---|---|
| **0_B - 15_B** | 0.21 | 0.30102 | 0.29529 | 0.32259 | 0.29275 | 0.34105 | 0.3324 |
| **0_B - 20_B** | N/A | 0.55864 | 0.54411 | 0.59003 | 0.58185 | 0.64212 | 0.61841 |
| **0_B - 30_B** | 0.71 | 1.28883 | 1.12976 | 1.38172 | 1.22281 | 1.47429 | 1.29414 |
| **0_B - 36_B** | N/A | 2.658 | 2.27511 | 2.87468 | 2.44231 | 3.08359 | 2.64515 |
| **0_B - 40_B** | N/A | 2.36635 | 1.92331 | 2.0912 | 2.06025 | 2.7203 | 2.19169 |
| **0_B - 45_B** | 1.8 | 5.15172 | 4.68965 | 5.54126 | 5.04327 | 5.96976 | 5.43876 |
| **0_B - 60_B** | 4.56 | 11.36557 | 13.94862 | 12.03387 | 14.67582 | 12.87804 | 15.46549 |

## 6.5 PRESSURE DISTRIBUTION ALONG THE WALL

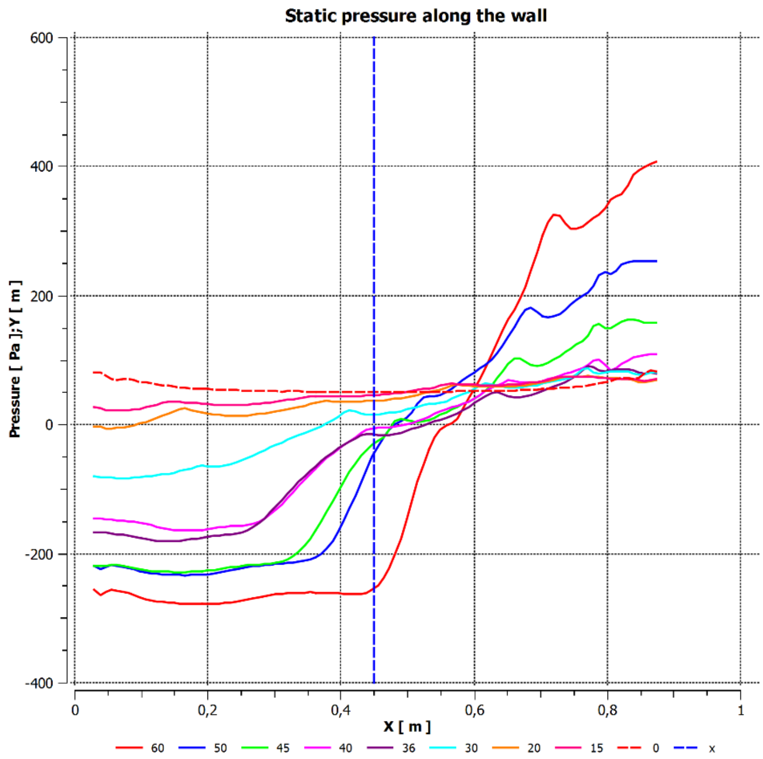


Fig. 11 Static pressure distribution along the wall for 4.4 [kg.s$^{-1}$]

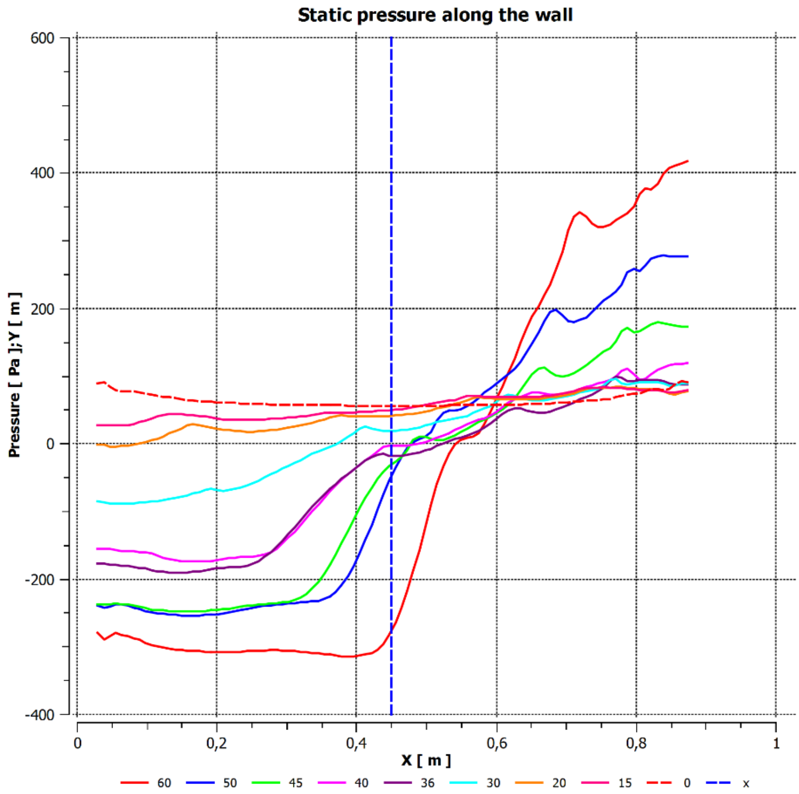


Fig. 12 Static pressure distribution along the wall for 4.6 [kg.s$^{-1}$]

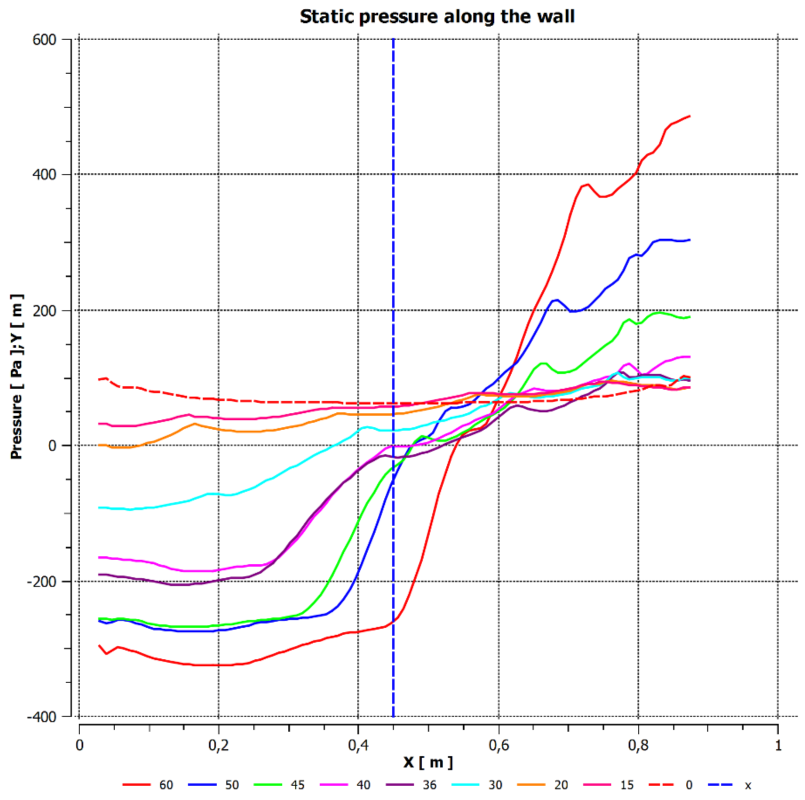


Fig. 13 Static pressure distribution along the wall for 4.8 [kg.s-1]

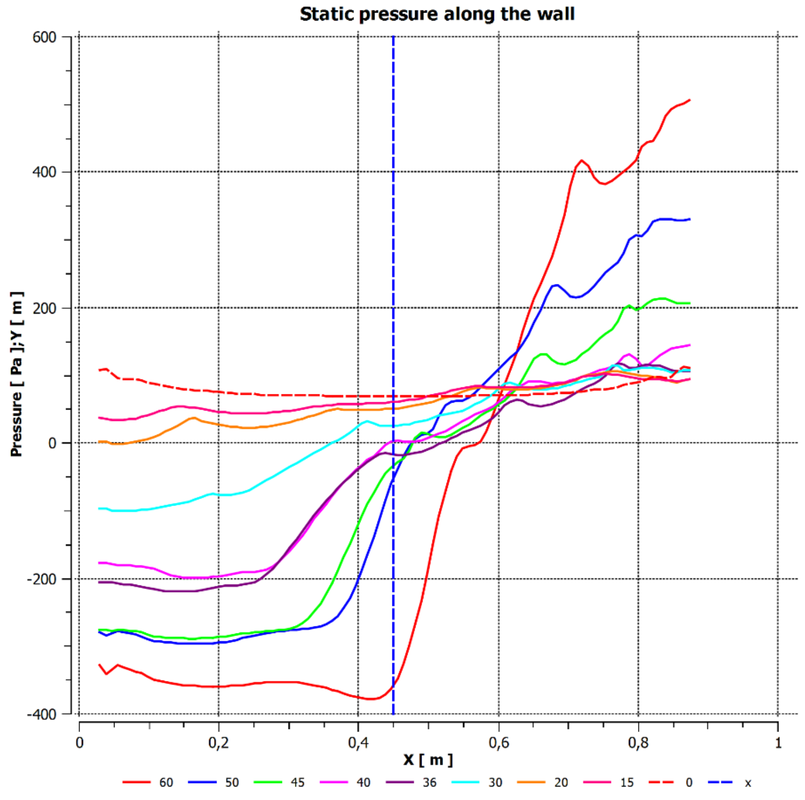


Fig. 14 Static pressure distribution along the wall for 5.0 [kg.s-1]

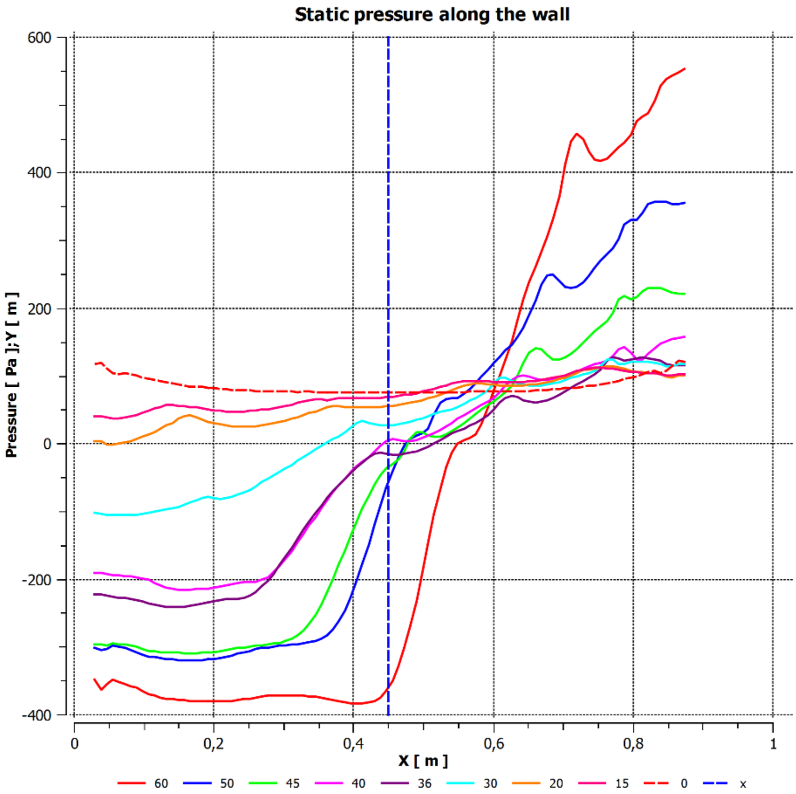


Fig. 15 Static pressure distribution along the wall for 5.2 [kg.s$^{-1}$]

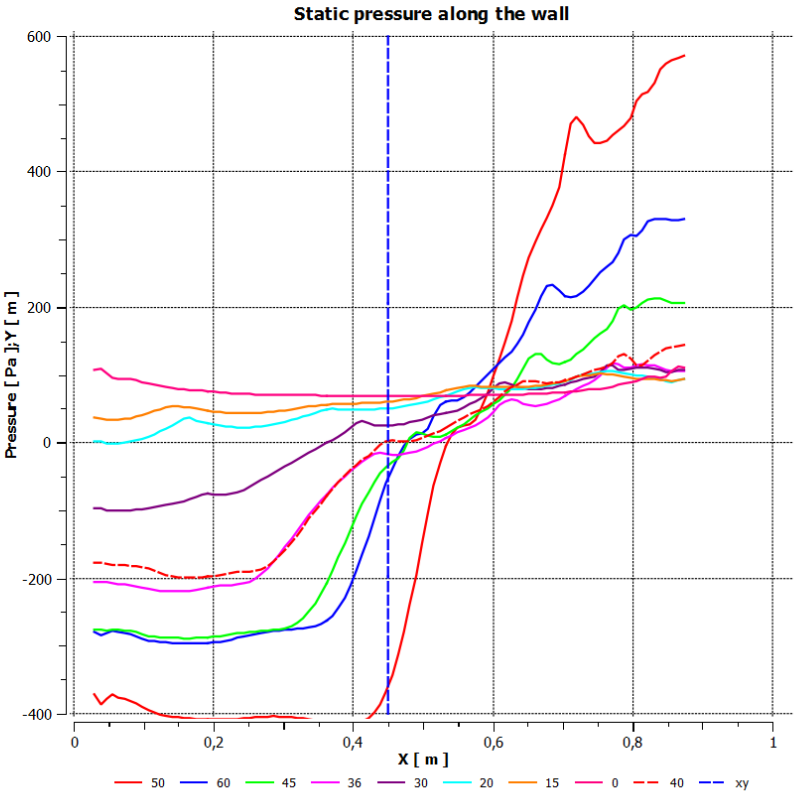


Fig. 16 Static pressure distribution along the wall for 5.4 [kg.s$^{-1}$]

## 6.6 VELOCITY FIELDS

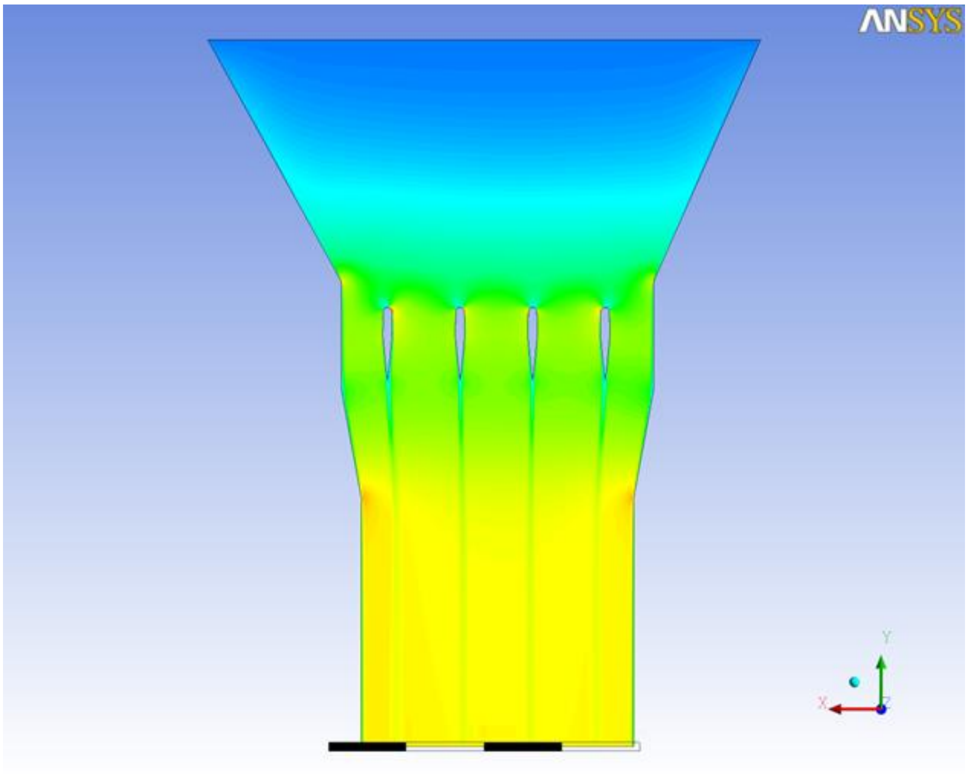

Fig. 17 Velocity field – blade position 0°; mass flow 5,0 [kg.s$^{-1}$]

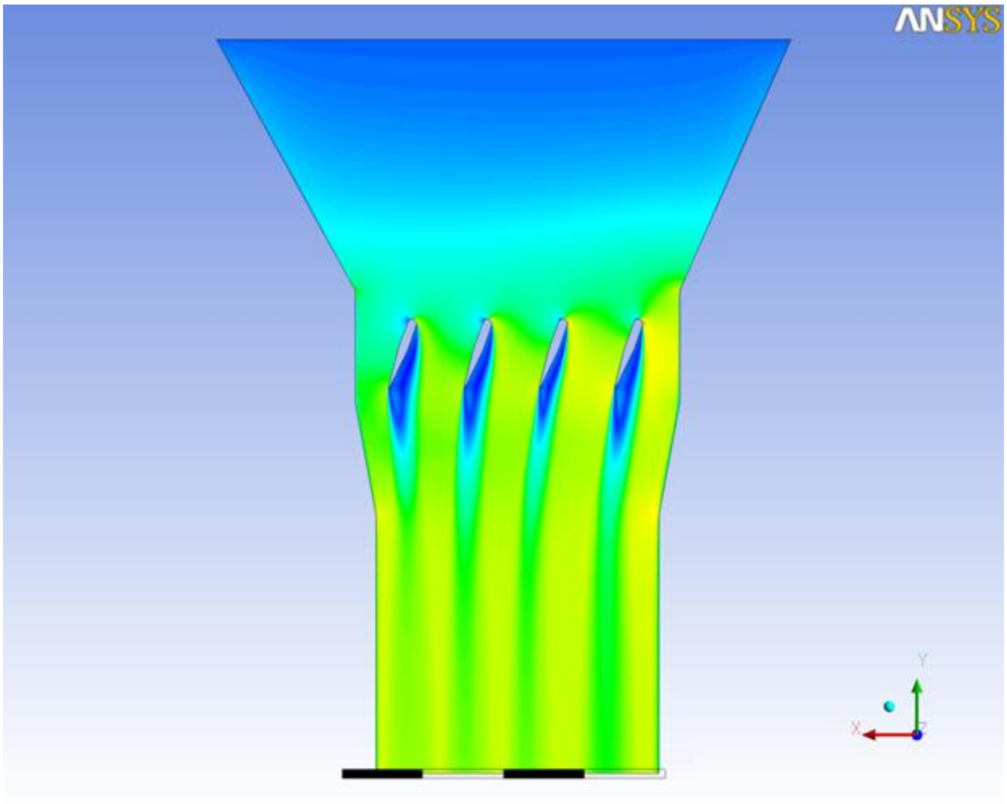

Fig. 18 Velocity field – blade position 20°; mass flow 5.0 [kg.s$^{-1}$]

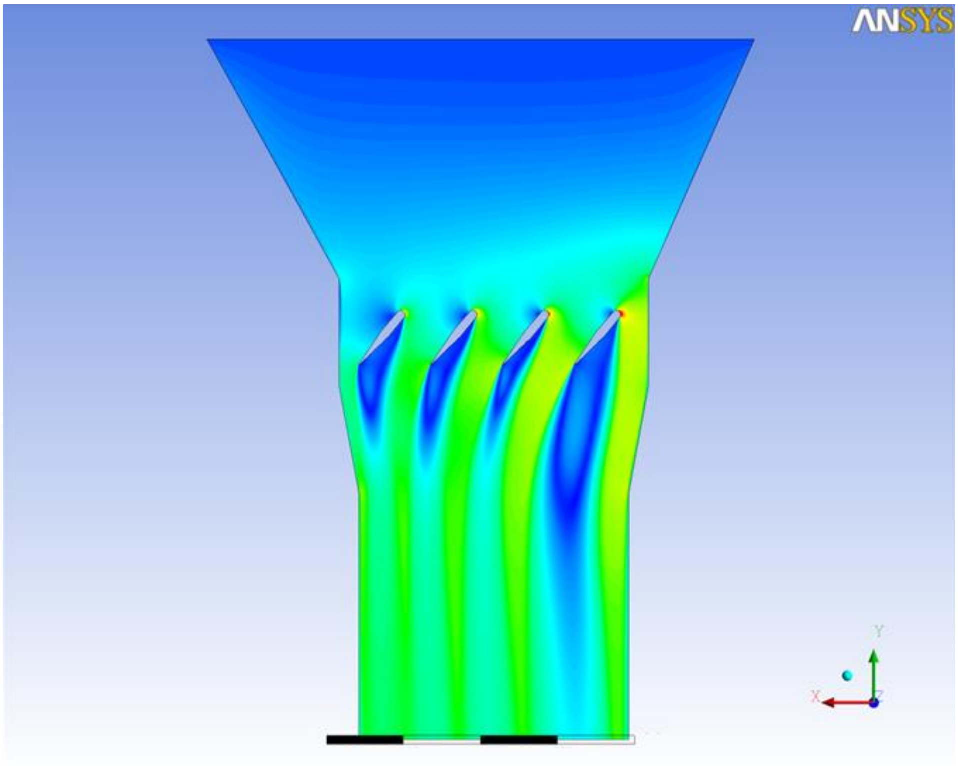


Fig. 19 Velocity field – blade position 40°; mass flow 5,0 [kg.s$^{-1}$]

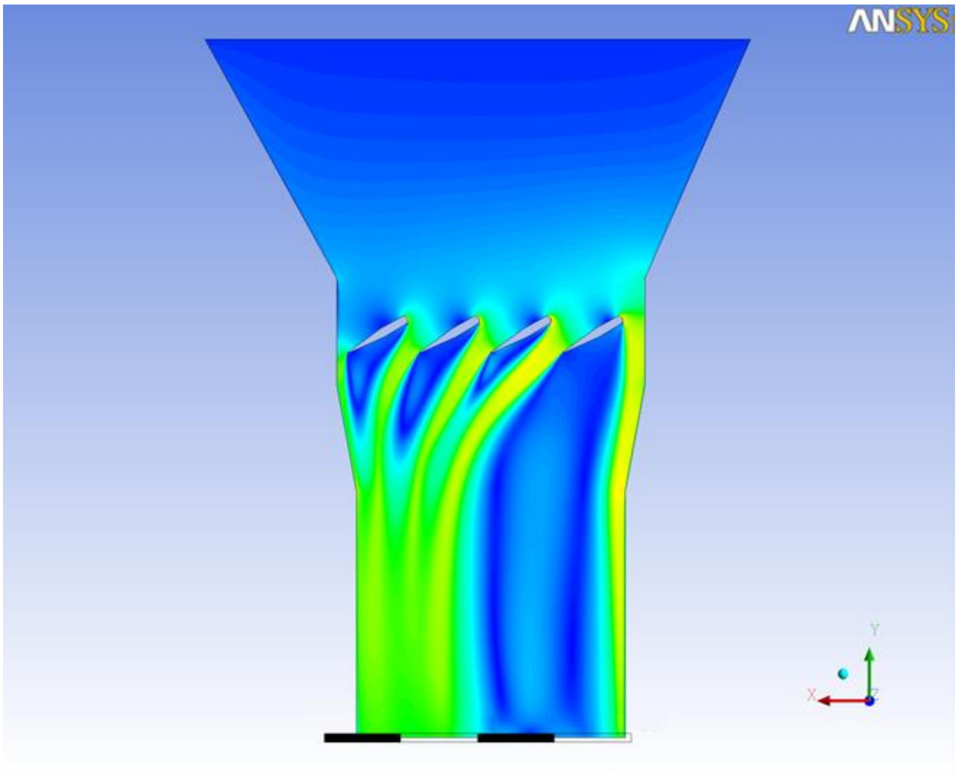


Fig. 20 Velocity field – blade position 60°; mass flow 5.0 [kg.s$^{-1}$]

# 7 CONCLUSIONS

This calculation was focused to investigate the behavior of the IGV system. The calculation showed the evolution of the pressure drop and its distribution along the IGV system. There are several conclusions from the results:

- The pressure drop behavior is according to the expectation. The pressure drop is changing according to the angle of the rotation of the blades and the mass flow rate (Tab. 2; Tab. 3)
- The difference between the values of measurement at the laboratory is very promising.
- The calculated values of a pressure drop are more pessimistic than the measured data. Which can be caused by an absence of the lower part, where the compressor is not included, or bad boundary conditions, Fig. 8
- The calculation revealed that the pressure distribution under the IGV blades is not homogenous (Fig. 11 – Fig. 16). When the blade rotates, one side creates a tapering channel and the other a diffuser channel (Fig. 17 – Fig. 20). These channels have a high effect to the pressure distribution and its effect increases with higher angles of the blade position.

## 7.1 SUGGESTIONS FOR FURTHER WORK

- To implement the lower part of the IGV into the numerical investigation, with a compressor, and to improve boundary conditions
- To redesign the measurements of the pressure drop