Impinging Jet Analysis – CFX

Author: Kacper Frelek

Supervisor: Ph.D. Michalina Kurkus - Gruszecka

Date: 20.03.2024

Abstract

0. Introduction

A jet impinging a surface is an important phenomenon in the fields of fluid mechanics and heat transfer. This technique can be used to increase heat transfer between the fluid and the surface. [2]

1. Motivation

The main goal of these calculations is to present results of impinging jet (Fig.1) analysis in Ansys CFX software, where different models were tested. Obtained results were then compared with the ones obtained using Star CCM+ presented in [1]. Results that are compared are obtained for following conditions:

- Distance to plate to Diameter ratio L/D ratio equal 2
- Diameter D = 25 mm
- Reynolds number Re=20 000
- Velocity profile v (x) = $v_{max}\ \left(1-\frac{r}{R}\right)^{\frac{1}{\alpha}}$
- v_{max} – velocity for which Re=20 000
- $\alpha=\frac{1}{\sqrt{f}}=6.1307$ (Coefficient for velocity profile)
- $f=4c_f=\ 0.0266\ \text{(x4 friction coefficient)}$
- $c_f=0.0791\ Re^{-\frac{1}{4}}=\ 0.00665$ (Blasius solution)
- Heat flux $q=100\frac{W}{m^2}$

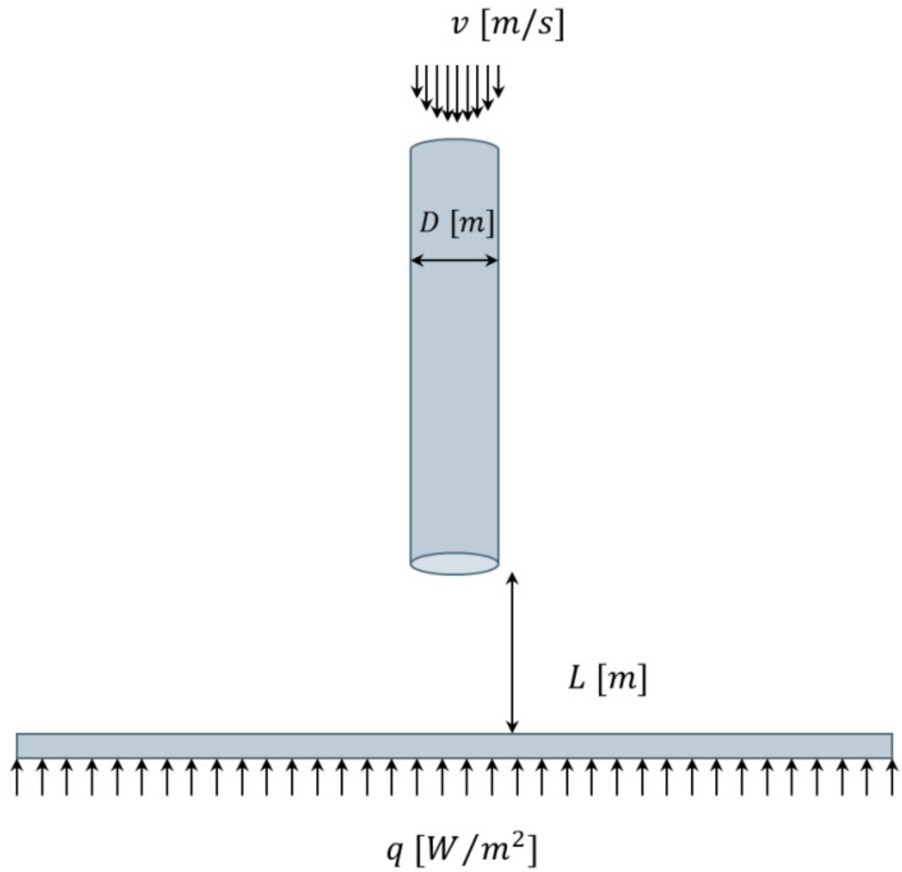

Fig.1 Scheme of a Impinging jet testing station [1]

2. Star CCM+ Results

Analysis conducted in Star CCM+ was performed by Siemens company, and the results were provided in the article [1]. Results for L/D=2 are presented in Fig.2.

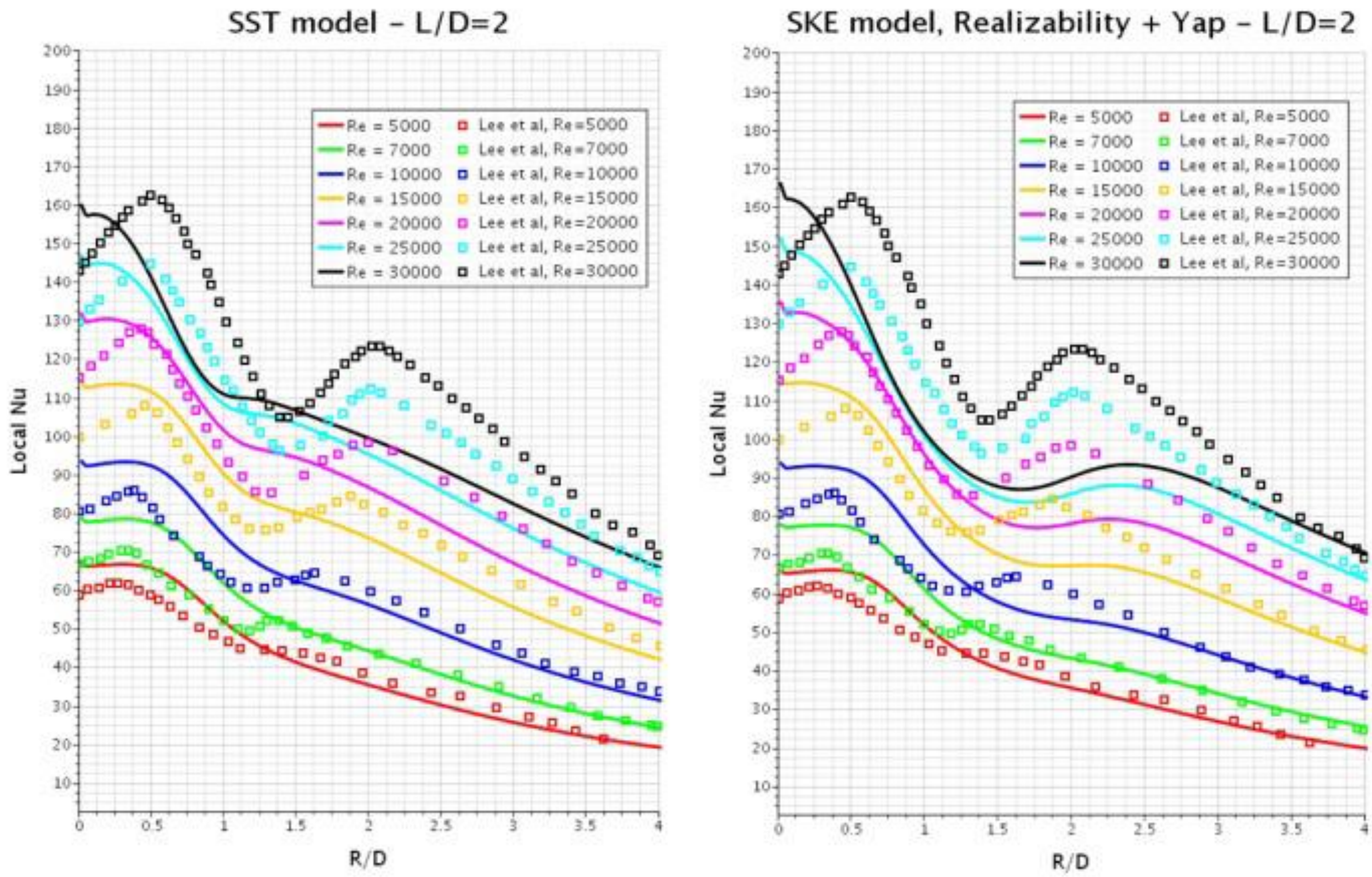

Fig.2 Results obtained by Siemens company in [1]

Main results of interest are presented as a pink line, where dots represent results that are expected to be obtained. One can denote that there are 2 peaks noticeable in the L/D<4 region, which are the main thing of interest to be observed during the analysis.

3. CFX calculations

a. Geometry

Geometry on which the analysis are made is imported and adjusted in the Ansys Design Modeler.

Due to the fact that CFX is mainly used for 3D cases, the creation of pure 2D geometry is impossible. To create a body that would act as a 2D body there is a need to create 3D body, which is 1 cell deep so it would act as a two-dimensional model. A 5 degree section of the cylinder structure was used in further studies.

Main geometry (Fig.3) was provided by the Framatome company, where it is same as the one used in [1].

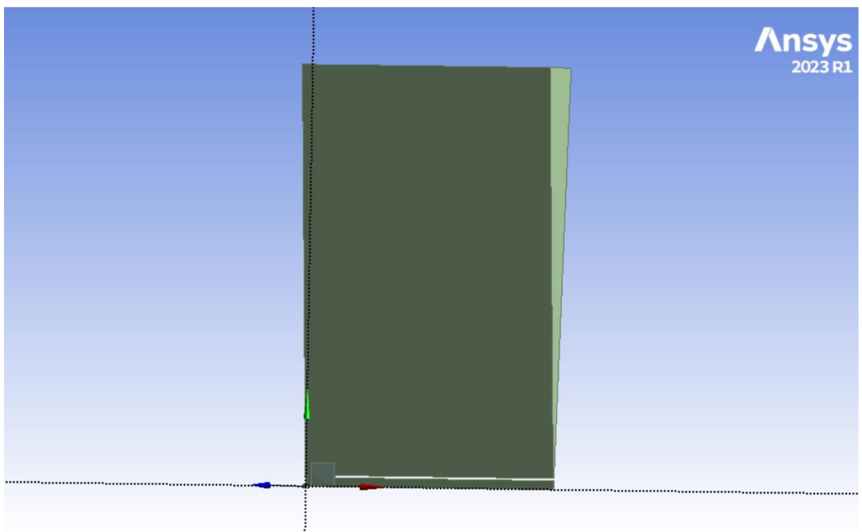

Fig. 3 Whole geometry analised in the studies

Moreover, to ensure good quality of results close to the outlet of the pipe, the body of influence where mesh will be denser, is also created (Fig. 4).

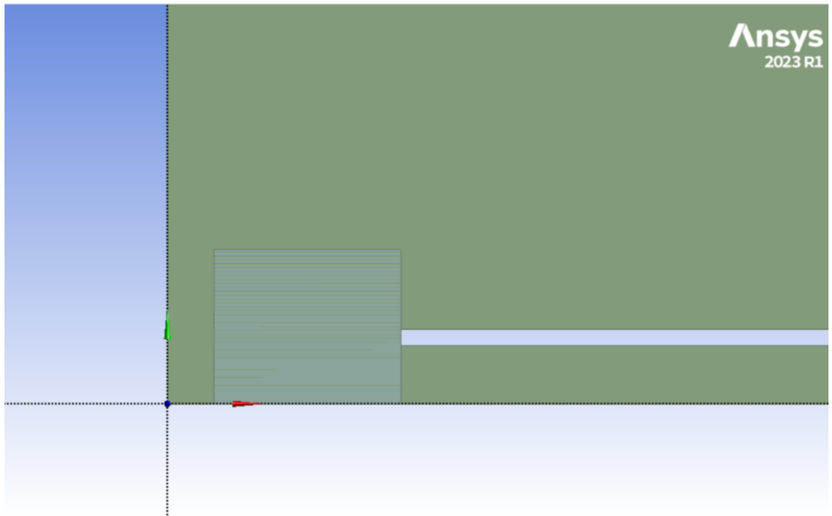

Fig.4 Body of Influence where mesh is set to be denser

b. k-ε model

The biggest expectations are put on the k-epsilon model, which is used as a main model to be checked.. For every turbulence model, there is a need to adjust the mesh to needs of a model. In the k-ε model the one of the values that is important to be observed is y+ value, which is expected to be y+~30. To obtain that, the boundary layers at the heated wall were introduced (Fig.5). Many different boundary layers sizes were tested, as to check Nusselt number dependency on y+.

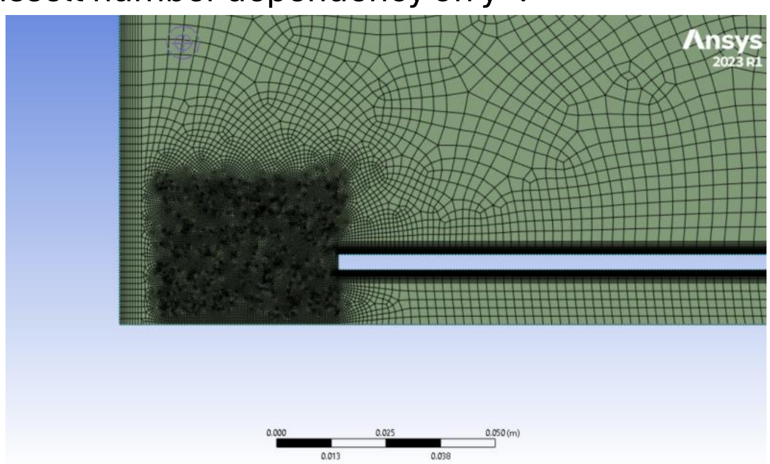

Fig. 5 Main mesh with later modified boundary layer

To present these results (Fig. 7), they were put next to the results obtained by Siemens in [1] on one graph, moreover with their y+ values. The graphs are presented as a Local Nusselt number on the plate, as a function of radial distance from the impingement point.

Case	1	2	3	4	5
#nodes	341941	340138	348811	345542	262277
#Layers	10	8	14	12	4
Max layer thickness [mm]	5	5	5	5	5
Y+ max at heated surface	38.85	54.26	19.83	28.59	126.7
Nu max at heated surface	235	226.9	230.8	234.7	183.8
Element size [mm]	3.75	3.75	3.75	3.75	5

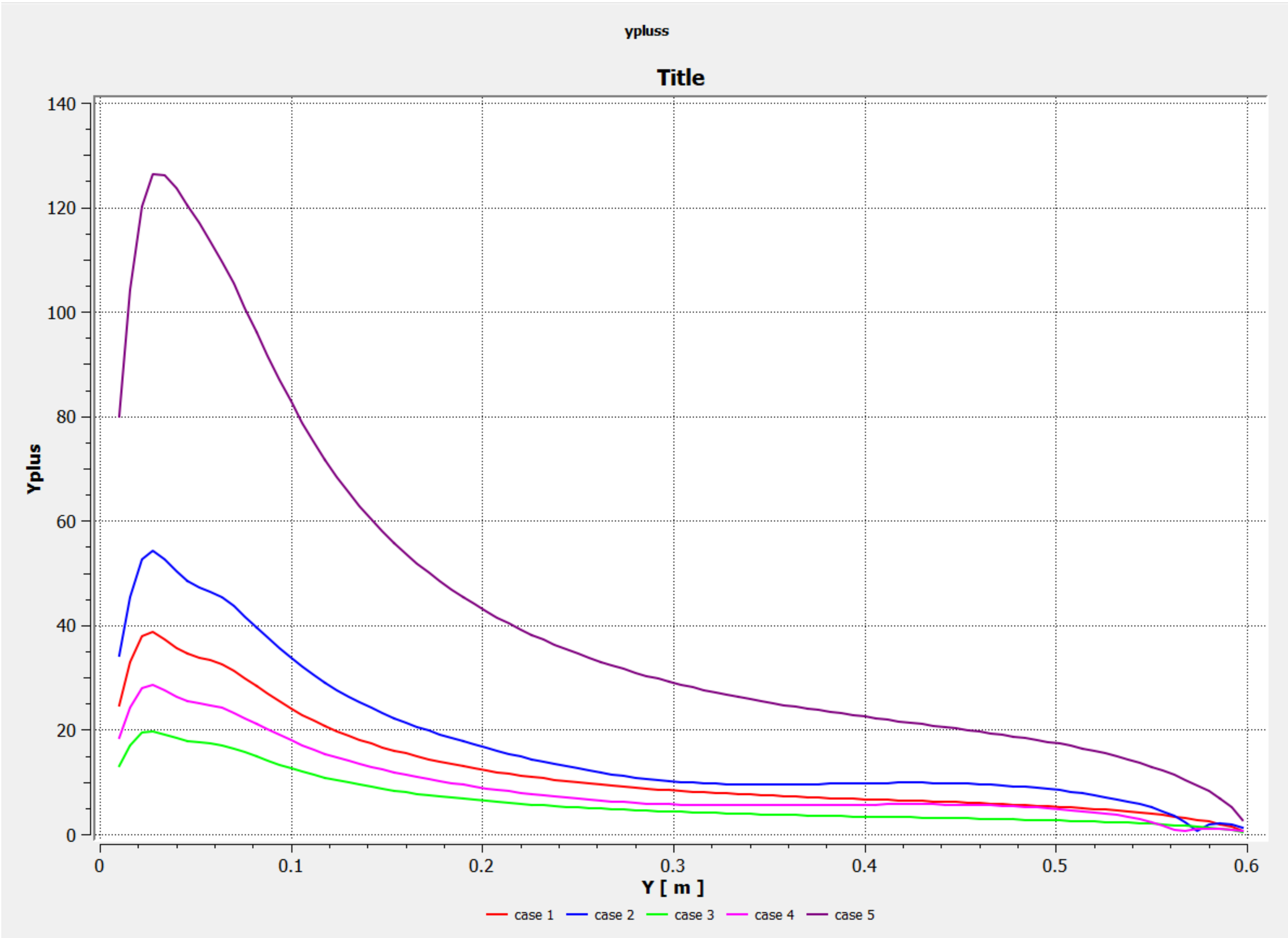

Fig. 7a Y+ value results obtained for different boundary layer settings on the heated wall

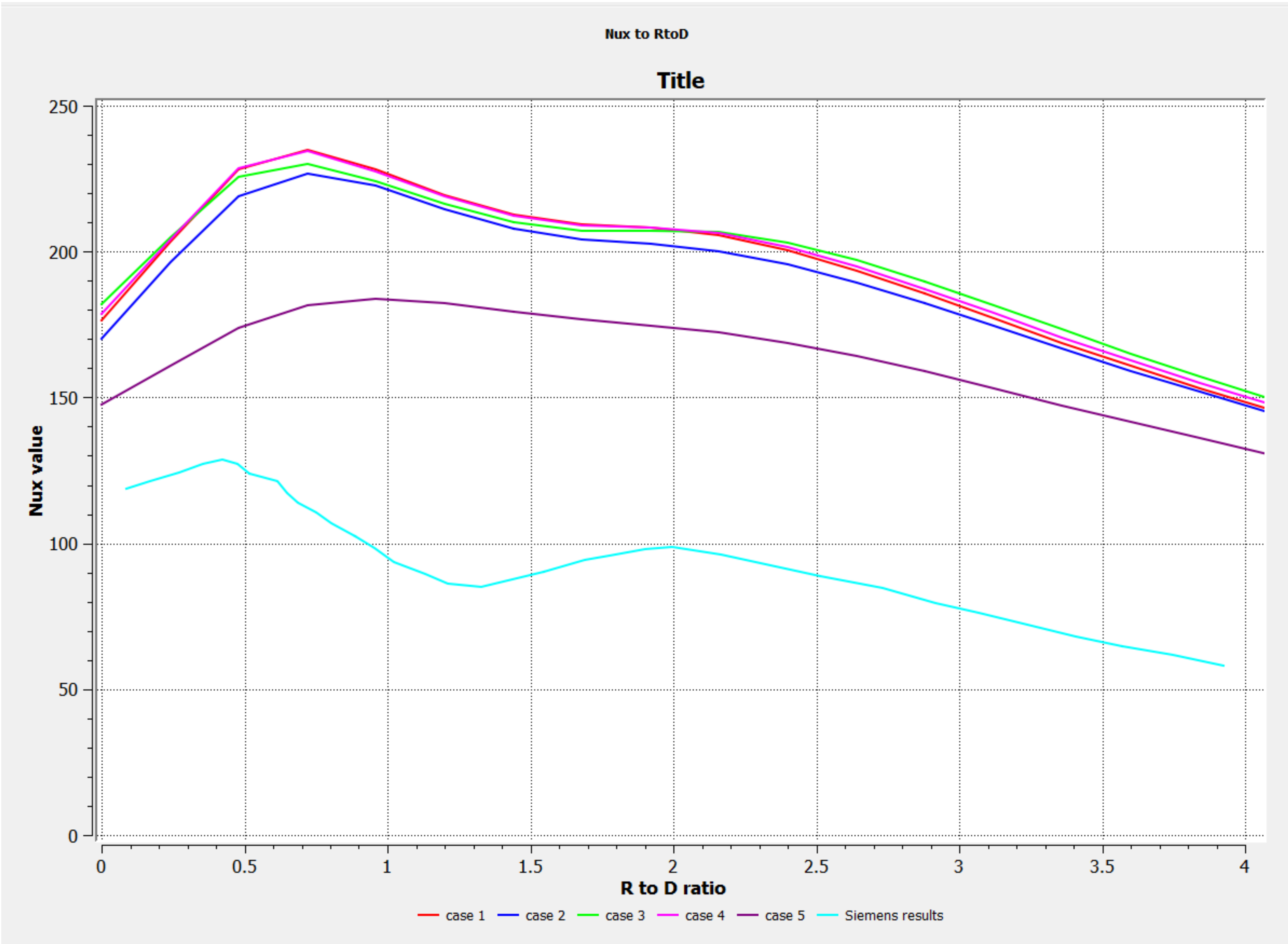

Fig. 7b Nu value results obtained for different boundary layer settings on the heated wall

One can denote that expected two peaks are not as clearly visible as in [1] and the values differ.

c. CFX manual

Directly from CFX manual guide

Due to the calculation errors obtained in the case of Y+ of 1, the theory of scalable wall functions was introduced.

2.8. Modeling Flow Near the Wall

This section presents the mathematical details of how flow near to a no-slip wall is modeled in Ansys CFX. An introduction to near-wall flow, modeling details and guidelines on using wall functions are presented. For details, see Modeling Flow Near the Wall in the *CFX-Solver Modeling Guide*.

2.8.1. Mathematical Formulation

The wall-function approach in Ansys CFX is an extension of the method of Launder and Spalding [13]. In the log-law region, the near wall tangential velocity is related to the wall-shear-stress, τ_ω, by means of a logarithmic relation.

In the wall-function approach, the viscosity affected sublayer region is bridged by employing empirical formulas to provide near-wall boundary conditions for the mean flow and turbulence transport equations. These formulas connect the wall conditions (for example, the wall-shear-stress) to the dependent variables at the near-wall mesh node, which is presumed to lie in the fully-turbulent region of the boundary layer.

The logarithmic relation for the near wall velocity is given by:

$$u^{+}=\frac{U_t}{u_\tau}=\frac{1}{\kappa}\ln\left(y^{+}\right)+C \quad (2.219)$$

where:

$$y^{+}=\frac{\rho\,\Delta y\,u_\tau}{\mu} \quad (2.220)$$

$$u_\tau=\left(\frac{\tau_\omega}{\rho}\right)^{1/2} \quad (2.221)$$

u^{+} is the near wall velocity, u_τ is the friction velocity, U_t is the known velocity tangent to the wall at a distance of Δy from the wall, y^{+} is the dimensionless distance from the wall, τ_ω is the wall shear stress, κ is the von Karman constant and C is a log-layer constant depending on wall roughness (natural logarithms are used).

A definition of Δy in the different wall formulations is available in Solver Yplus and Yplus (p. 155).

2.8.1.1. Scalable Wall Functions

Equation 2.219 (p. 153) has the problem that it becomes singular at separation points where the near wall velocity, U_t, approaches zero. In the logarithmic region, an alternative velocity scale, u^* can be used instead of u_τ:

$$u^* = C_\mu^{1/4} k^{1/2} \tag{2.222}$$

This scale has the useful property that it does not go to zero if U_t goes to zero. Based on this definition, the following explicit equation for u_τ can be obtained:

$$u_\tau = \frac{U_t}{\frac{1}{\kappa}\ln\left(y^*\right)+C} \tag{2.223}$$

The absolute value of the wall shear stress τ_ω, is then obtained from:

$$\tau_\omega = \rho\, u^*\, u_\tau \tag{2.224}$$

where:

$$y^* = \left(\rho\, u^*\, \Delta y\right) / \mu \tag{2.225}$$

and u^* is as defined earlier.

One of the major drawbacks of the wall-function approach is that the predictions depend on the location of the point nearest to the wall and are sensitive to the near-wall meshing; refining the mesh does not necessarily give a unique solution of increasing accuracy (Grotjans and Menter [10]). The problem of inconsistencies in the wall-function, in the case of fine meshes, can be overcome with the use of the **Scalable Wall Function** formulation developed by Ansys CFX. It can be applied on arbitrarily fine meshes and allows you to perform a consistent mesh refinement independent of the Reynolds number of the application.

The basic idea behind the scalable wall-function approach is to limit the y^* value used in the logarithmic formulation by a lower value of $\tilde{y}^* = \max\left(y^*, 11.06\right)$ where 11.06 is the value of y^* at the intersection between the logarithmic and the linear near wall profile. The computed $\tilde{y}^*$ is therefore not allowed to fall below this limit. Therefore, all mesh points are outside the viscous sublayer and all fine mesh inconsistencies are avoided.

The boundary condition for the dissipation rate, ε, is then given by the following relation, which is valid in the logarithmic region:

$$\varepsilon=\frac{\rho u^{*}}{\tilde{y}^{*}\mu}\frac{C_{\mu}^{3/4}}{\kappa}k^{3/2} \quad (2.226)$$

It is important to note the following points:

- To fully resolve the boundary layer, you should put at least 10 nodes into the boundary layer.
- Do not use Standard Wall Functions unless required for backwards compatibility.
- The upper limit for y^+ is a function of the device Reynolds number. For example, a large ship may have a Reynolds number of 10^9 and y^+ can safely go to values much greater than 1000. For lower Reynolds numbers (for example, a small pump), the entire boundary layer might only extend to around $y^+ = 300$. In this case, a fine near wall spacing is required to ensure a sufficient number of nodes in the boundary layer.

If the results deviate greatly from these ranges, the mesh at the designated Wall boundaries will require modification, unless wall shear stress and heat transfer are not important in the simulation.

In most turbulent flows the turbulence kinetic energy is not completely zero and the definition for u^* given in Equation 2.222 (p. 154) will give proper results for most cases. In flows with low free stream turbulence intensity, however, the turbulence kinetic energy can be very small and lead to vanishing u^* and therefore also to vanishing wall shear stress. If this situation happens, a lower limiter can be applied to u^* by setting the expert parameter `'ustar limiter = t'`. u^* will then be calculated using the following relation:

$$u^{*}=\max\left(C_{\mu}^{1/4}\sqrt{k}\,,Coef\cdot U_{t}/\tilde{y}^{*}\right) \quad (2.227)$$

The coefficient used in this relation can be changed by setting the expert parameter `'ustar limiter coef '`. The default value of this parameter is 0.01.

2.8.1.2. Solver Yplus and Yplus

In the solver output, there are two arrays for the near wall y^+ spacing. The definition for the Yplus variable that appears in the post processor is given by the standard definition of y^+ generally used in CFD:

$$y^{+}=\frac{\sqrt{\tau_{\omega}/\rho}\cdot\Delta n}{v} \quad (2.228)$$

where Δn is the distance between the first and second mesh points off the wall.

In addition, a second variable, Solver Yplus, is available and contains the y^+ used in the logarithmic profile by the solver. It depends on the type of wall treatment used, which can be one of three different treatments in Ansys CFX. They are based on different distance definitions and velocity scales. This has partly historic reasons, but is mainly motivated by the desire to achieve an optimum performance in terms of accuracy and robustness:

- Standard wall function (based on $\Delta y=\Delta n/4$)
- Scalable wall function (based on $\Delta y=\Delta n/4$)
- Automatic wall treatment (based on $\Delta y=\Delta n$)

The scalable wall function y^+ is defined as:

$$y^{+}=\max\left(y^{*},11.06\right) \quad y^{*}=\frac{u^{*}\Delta n/4}{v} \quad (2.229)$$

and is therefore based on $^1/_4$ of the near wall mesh spacing.

Note that both the scalable wall function and the automatic wall treatment can be run on arbitrarily fine meshes.

Moreover CFX uses a cell-centered FVM (Finite Volume Method) and '*In CFX, Scalable Wall Functions are used for all turbulence models based on the ε-equation. For k-ω based models (including the SST model), an Automatic near-wall treatment method is applied'* as stated in Ansys CFX manual.

4. Summation

After simulation impinging jet conditions in the Ansys CFX and comparing them with experimental results, one can state that this software is not the most accurate approach, as the characteristic of Nusselt number is not as desired. Moreover choice of k-epsilon model was challenging, as the only wall function available for this model is a scalable wall function.

Appendix

[1] *Verification: Impinging Circular Jet with Heat Transfer* Simcenter STAR-CCM+

[2] *Aerodynamic and heat transfer analysis of a impinging jet on a concave surface* G.J. Poitras, A. Babineau, G. Roy, L.-E. Brizzi