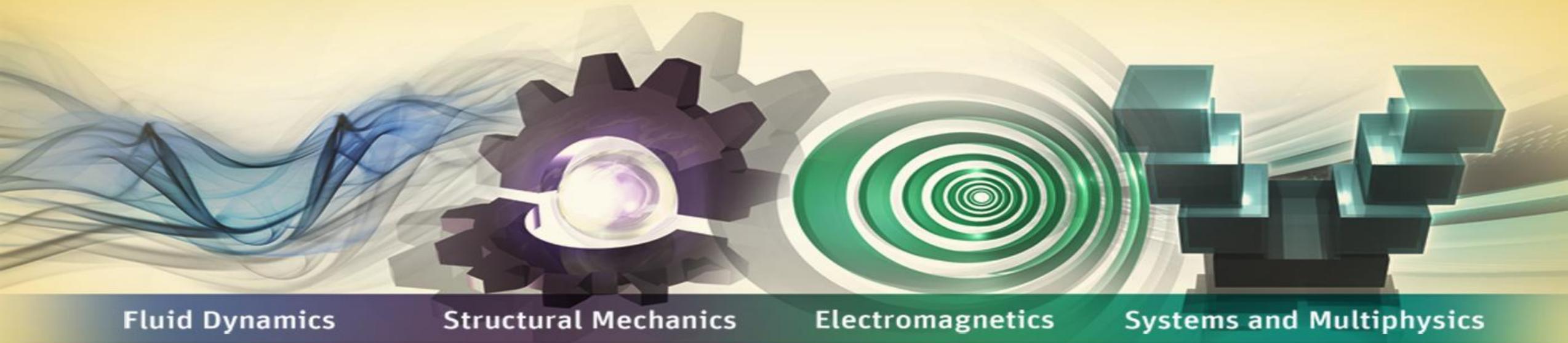


# Mesh Creation According to Predicted $y^+$



Fluid Dynamics

Structural Mechanics

Electromagnetics

Systems and Multiphysics

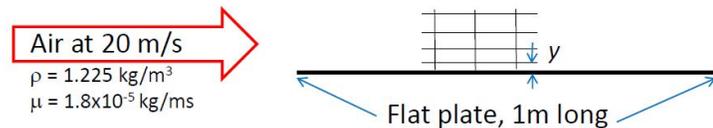
# Introduction

**This short tutorial demonstrates in practice the technique for Creating the desired wall spacing in ANSYS Mesh for matching a flat plate calculated  $y^+$  in ANSYS Fluent .**

# Part A: Predicting and calculating the desired wall spacing

## Example in Predicting Near-wall Cell Size

- During the pre-processing stage, you will need to know a suitable size for the first layer of grid cells (inflation layer) so that  $Y^+$  is in the desired range
- The actual flow-field will not be known until you have computed the solution (and indeed it is sometimes unavoidable to have to go back and remesh your model on account of the computed  $Y^+$  values)
- To reduce the risk of needing to remesh, you may want to try and predict the cell size by performing a hand calculation at the start, for example:



*The question is what height (y) should the first row of grid cells be. We will use SWF, and are aiming for  $Y^+ \approx 50$*

- For a flat plate, Reynolds number (  $Re_L = \frac{\rho V L}{\mu}$  ) gives  $Re_L = 1.4 \times 10^6$

Recall from earlier slide, flow over a surface is turbulent when  $Re_L > 5 \times 10^5$

**Taken from lecture 7 in ANSYS Fluent introduction course**

## Calculating Wall Distance for a Given $y^+$

- Begin with the definition of  $y^+$  and rearrange:

$$y^+ = \frac{\rho U_\tau y}{\mu} \iff y = \frac{y^+ \mu}{U_\tau \rho}$$

- The target  $y^+$  value and fluid properties are known, so we need  $U_\tau$ , which is defined as:

$$U_\tau = \sqrt{\frac{\tau_w}{\rho}}$$

- The wall shear stress,  $\tau_w$ , can be found from the skin friction coefficient,  $C_f$ :

$$\tau_w = \frac{1}{2} C_f \rho U_\infty^2$$

- A literature search suggests a formula for the skin friction on a plate<sup>1</sup> thus:

$$C_f = 0.058 Re_L^{-0.2}$$

- $Re$  is known, so use the definitions to calculate the first cell height

$$C_f = 0.058 Re_L^{-0.2} = .0034$$

$$\tau_w = \frac{1}{2} C_f \rho U_\infty^2 = 0.83 \text{ kg/(m} \cdot \text{s}^2)$$

$$U_\tau = \sqrt{\frac{\tau_w}{\rho}} = 0.82 \text{ m/s}$$

- We know we are aiming for  $y^+$  of 50, hence:

$$y = \frac{y^+ \mu}{U_\tau \rho} = 9 \times 10^{-4} \text{ m}$$

**our first cell height y should be approximately 1 mm.**

<sup>1</sup>An equivalent formula for internal flows, with Reynolds number based on the pipe diameter is  $C_f = 0.079 Re_d^{-0.25}$

# Part B: Creating the desired wall spacing in ANSYS Mesh

- 1) Unarchive the supplied yPlus\_example.wbpz.

The geometry is already supplied, it consists of a simple rectangular domain, bottom wall of which will be meshed with the theoretic values from previous slide.

- 2) Open Mesh Component, define named selections as shown on the next slide.

Geometry Virtual Body

outline

Filter: Name

Project

- Model (B3)
  - Geometry
  - Coordinate Systems
  - Mesh
    - Edge Sizing
    - Face Meshing
    - Edge Sizing 2
  - Named Selections
    - cfid
    - inlet
    - outlet
    - wall\_top
    - wall\_bot

Details of "Geometry"

Definition	
Source	C:\Users\tensor\Desktop\Afeka\160420 - yPlus_exempl...
Type	SpaceClaim
Length Unit	Meters
Bounding Box	
Properties	
Statistics	
Basic Geometry Options	
Advanced Geometry Options	

ANSYS R17.0

Geometry 20-Apr-16 16:33

inlet

wall\_top

wall\_bot

outlet

0.000 0.125 0.250 0.375 0.500 (m)

# Part B: Creating the desired wall spacing in ANSYS Mesh

- 3) Select wall\_top and wall\_bot related edges, right click -> insert -> sizing.
- 4) Define sizing parameters as shown, these parameters will ensure that the stream wise aspect ratio of the near wall region cells will not exceed 5
- 5) Select inlet and outlet edges, right click -> insert -> sizing.
- 6) Define sizing parameters as shown.
- 7) When applying bias please note that one of the edges (inlet or outlet) will have to be reversed.

Details of "Edge Sizing 2" - Sizing	
<b>Scope</b>	
Scoping Method	Geometry Selection
Geometry	2 Edges
<b>Definition</b>	
Suppressed	No
Type	Number of Divisions
<input type="checkbox"/> Number of Divisions	150
Behavior	Hard
Bias Type	No Bias

Details of "Edge Sizing" - Sizing	
<b>Scope</b>	
Scoping Method	Geometry Selection
Geometry	2 Edges
<b>Definition</b>	
Suppressed	No
Type	Number of Divisions
<input type="checkbox"/> Number of Divisions	100
Behavior	Hard
Bias Type	_____
Bias Option	Bias Factor
<input type="checkbox"/> Bias Factor	7.
Reverse Bias	1 Edge

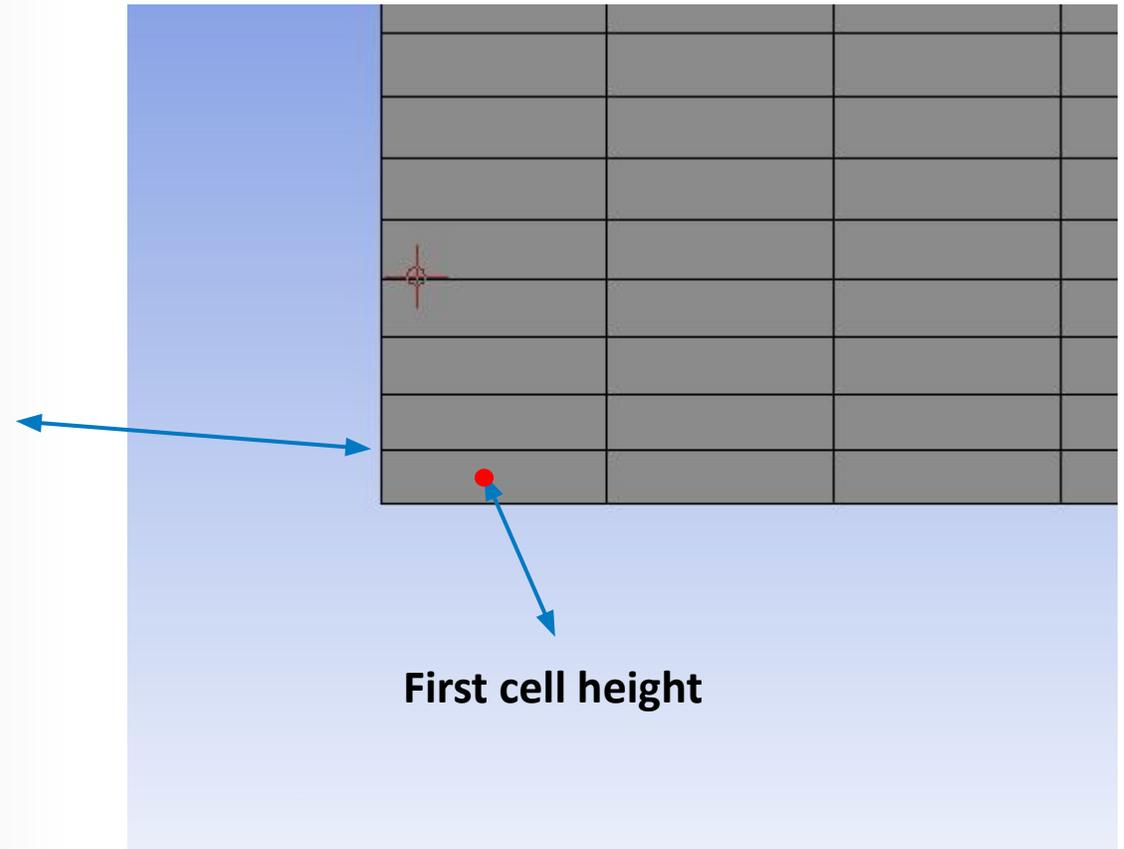


# Part B: Creating the desired wall spacing in ANSYS Mesh

Bias factor and number of divisions on inlet and outlet edges were calculated in a way that the first cell centroid will be situated at the theoretical value of  $\sim 0.9$  [mm] calculated earlier, please note that in terms of mesh it dictates first “layer” height at  $\sim 1.8$  [mm].

Remember that you also can obtain desired first cell height wall spacing with the help of local or global inflation controls.

First layer height



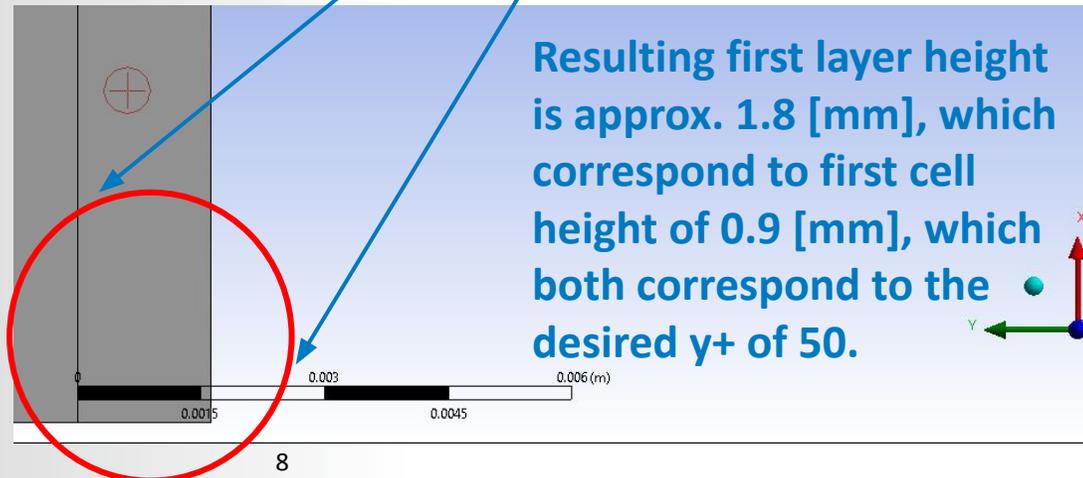
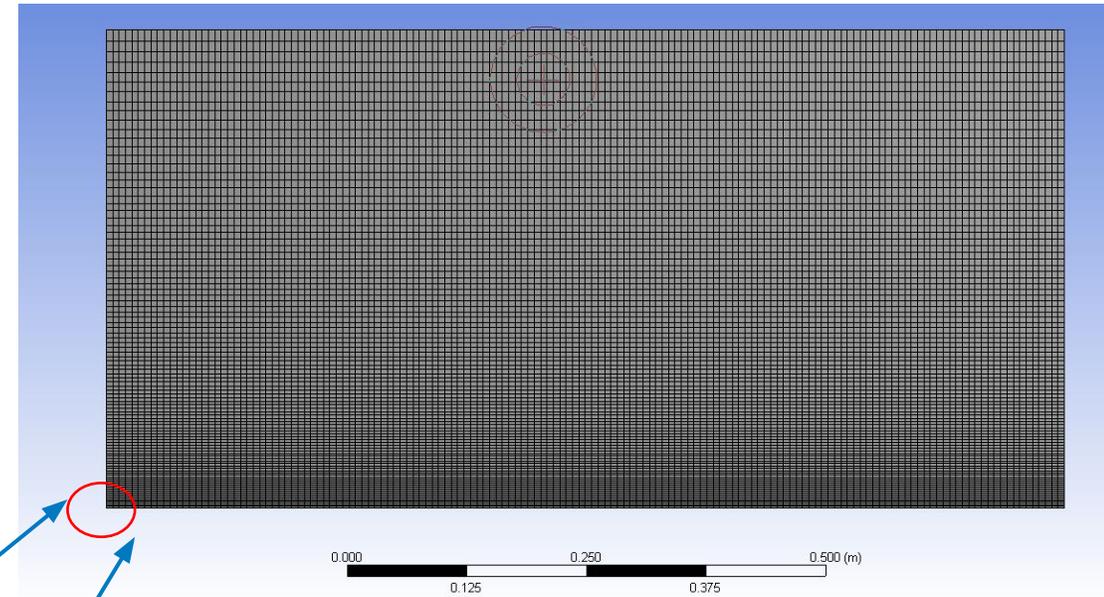
First cell height

# Part B: Creating the desired wall spacing in ANSYS Mesh

- 8) Select the rectangular face, right click -> insert -> face meshing.

This is done so that the resulting mesh will be smooth and mapped.

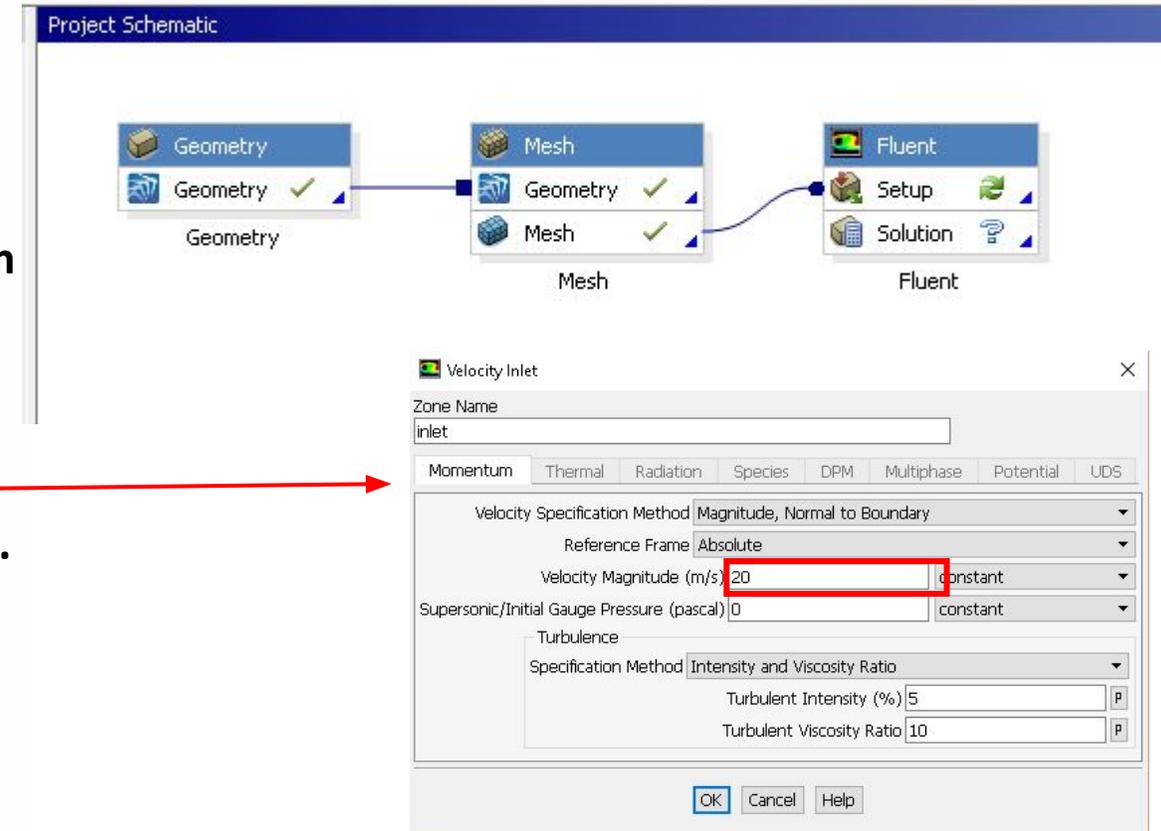
- 9) Click “Generate Mesh” and observe the resulting mesh. You can zoom in, rotate and use the ruler on the screen in order to verify the height of the first cell.



# Part C: Fluent Setup

- 10) Close ANSYS Mesh, connect it to a new Fluent component.
- 11) Update Mesh component if necessary.
- 12) Open Fluent setup in with double precision option.
- 13) Models -> Viscous -> K-Epsilon.
- 14) Boundary conditions -> Inlet
- 15) Solution initialization -> hybrid -> Initialize.
- 16) Run calculation -> 100 Iterations -> Calculate.

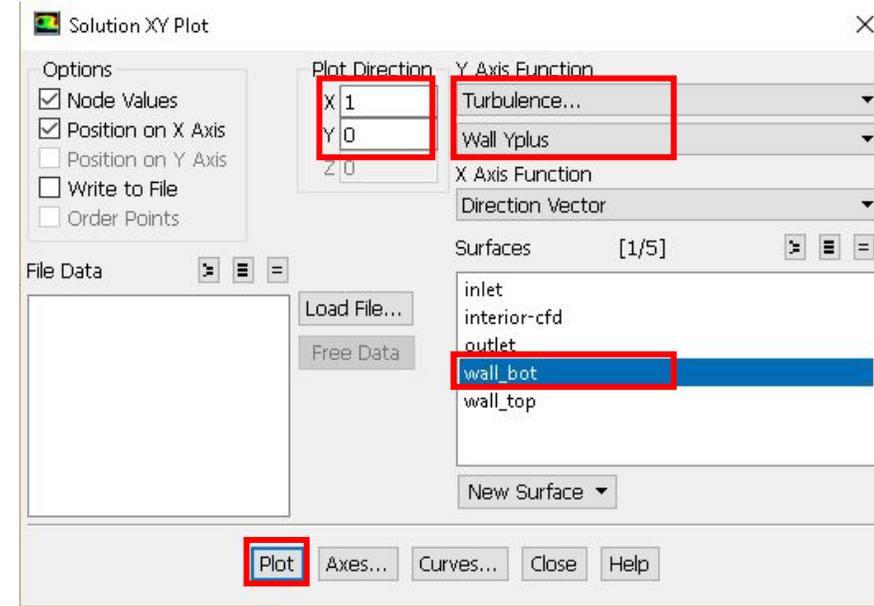
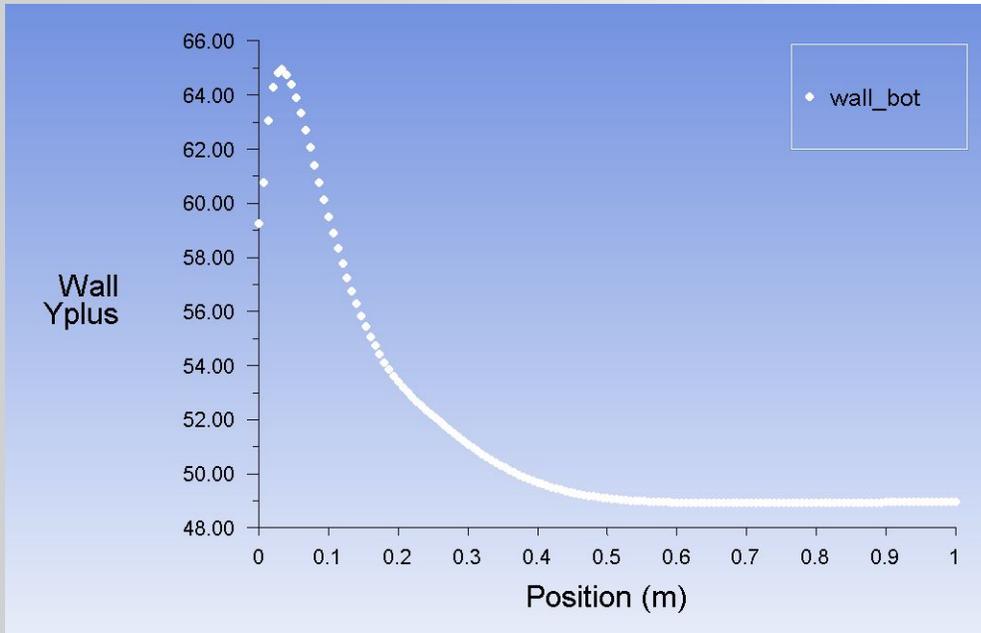
The run should converge after ~40 iterations.



# Part C: Fluent Setup

The final step of this tutorial will be checking what  $y^+$  values we actually have for our first cell adjacent to wall\_bot bc.

- 17) Results -> Plots -> XY Plot.
- 18) Set the parameters as shown and click plot.
- 19) Observe the  $y^+$  distribution next to wall\_bot.



Please note that it takes about 0.4 [m] for the flow to develop and then we can see that we get the values that we expected from theoretical calculation.

# Conclusions

**This tutorial demonstrated how to implement theoretical  $y^+$  calculation in to the computational grid and how to verify the  $y^+$  value of the grid after solution convergence.**

**As it was mentioned, it takes the flow some distance to develop, that's why we can see values of up to 64 in terms of  $y^+$  in the first 0.4 [m] of the plate/wall.**

**You can try and see what values of  $y^+$  you will receive for developed flow at the inlet. You can achieve this by sampling the velocity magnitude at the outlet to a profile and then assigning this profile to the inlet.**