Experimental/Computational dual Approach in Fluid Dynamics

Laminar vs turbulent boundary layer on a flat plate

Yannick BURY

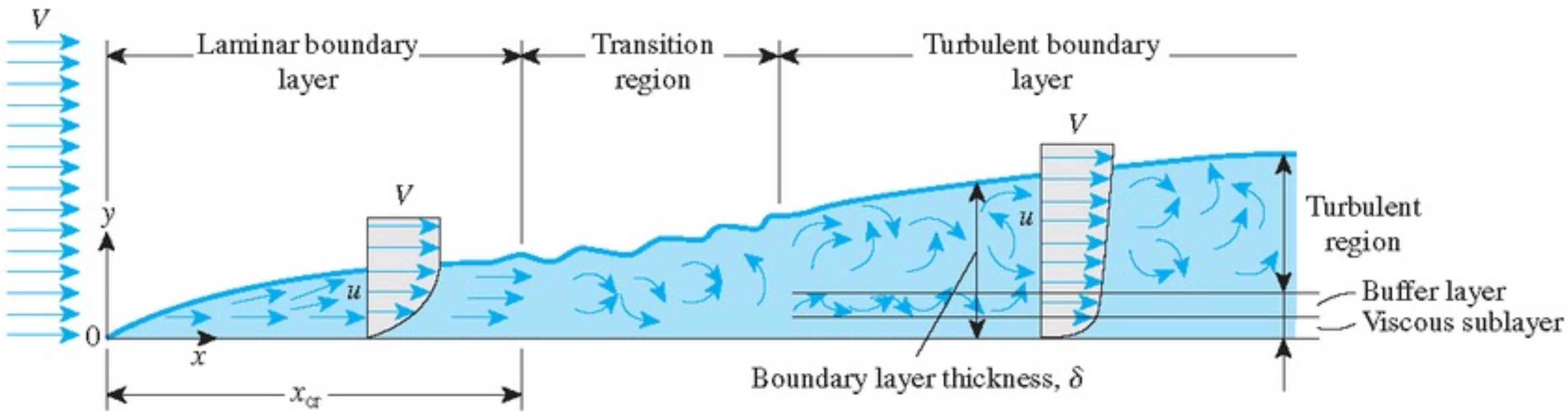
yannick.bury@isae.fr

- A first canonical flow: studying the boundary layer on a flat plate
  - Generate more or less refined 2D meshes
  - Simulate laminar or turbulent flow
  - Post process the results

- theory / tests / simulations correlation
  - Blasius laws in laminar flow
  - Boundary layer partition laws in turbulent flow

Description of the problem

- Boundary layer on a flat plate

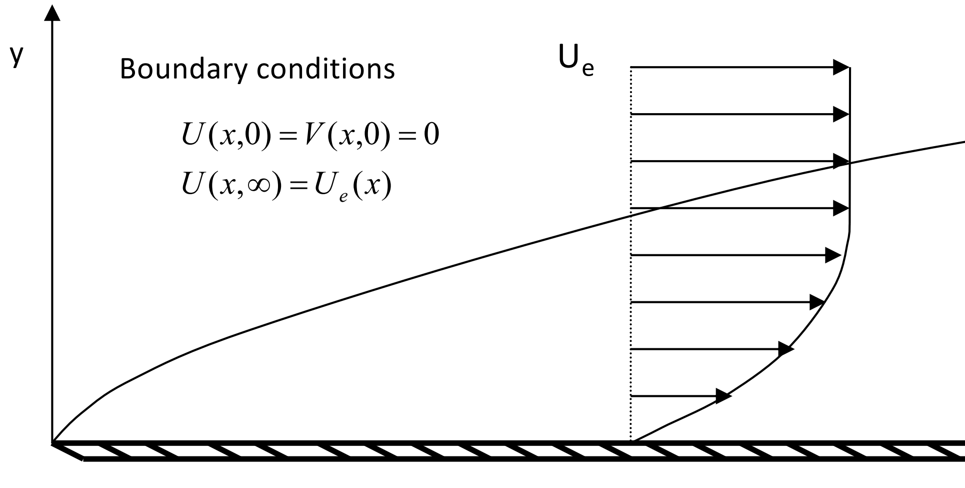


thermal-engineering.org

Laminar boundary layer on a flat plate with no pressure gradient

- General hypotheses
  - Two-dimensional, steady flow
  - Incompressible
  - Flat plate with zero incidence (no pressure gradient is expected)

- Equations of the laminar boundary layer on a flat plate



Boundary conditions

$$U(x,0) = V(x,0) = 0$$

$$U(x,\infty) = U_e(x)$$

$$P + \frac{1}{2}\rho U_e^2 = cste$$

$$\frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} = 0$$

$$U\frac{\partial U}{\partial x} + V\frac{\partial U}{\partial y} = -\frac{1}{\rho}\frac{\partial P}{\partial x} + \nu\frac{\partial^2 U}{\partial y^2}$$

$$\frac{\partial P}{\partial y} = 0$$

Laminar boundary layer on a flat plate with no pressure gradient

- Resolution of the "boundary layer" equations for a laminar flow

Initial problem

$$\begin{cases} \dfrac{\partial U}{\partial x} + \dfrac{\partial V}{\partial y} = 0 \\ U\dfrac{\partial U}{\partial x} + V\dfrac{\partial U}{\partial y} = -\dfrac{1}{\rho}\dfrac{\partial P}{\partial x} + \nu\dfrac{\partial^2 U}{\partial y^2} \\ \dfrac{\partial P}{\partial y} = 0 \end{cases}$$

$$U(x,0) = V(x,0) = 0$$
$$U(x,\infty) = U_e(x)$$

Introducing a stream function

$$U = \frac{\partial \Psi}{\partial y}, V = -\frac{\partial \Psi}{\partial x}$$

Search for self-similar solutions

$$\begin{cases} \Psi(x,y) = (U_e \nu . x)^{1/2} f(\eta) \\ \eta = y.(U_e/(\nu . x))^{1/2} \\ 2f''' + ff'' = 0 \end{cases}$$

$$f(0) = f'(0) = 0, f'(\infty) = 1$$

Determination of the velocity field and wall shear stress

$$\begin{cases} U(x,y) = U_e f'(\eta) \\ V(x,y) = \dfrac{1}{2}\left(\dfrac{U_e \nu}{x}\right)^{1/2}(\eta f' - f) \\ \dfrac{\tau_w(x)}{\rho U_e^2} = f''(0).\left(\dfrac{\nu}{U_e x}\right)^{1/2} \end{cases}$$

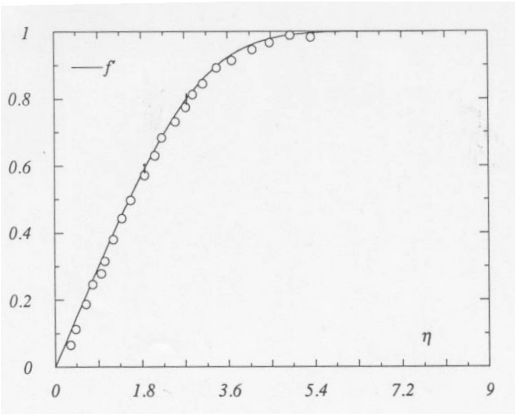
$$C_f = \frac{\tau_w}{\frac{1}{2}\rho U_e^2}$$

Local skin friction coefficient $C_f = \dfrac{0.664}{\sqrt{\mathrm{Re}_x}}$

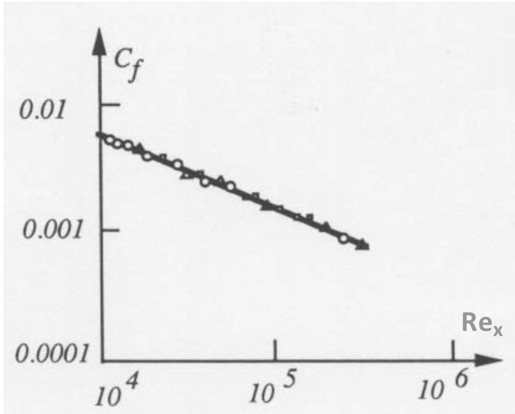
Skin friction coefficient of *L*-length plate. $C_D = \dfrac{1.328}{\sqrt{\mathrm{Re}_l}}$

Laminar boundary layer on a flat plate with no pressure gradient

- Main results for a laminar flow: the Blasius laws



Dimensionless velocity distribution at different Reynolds numbers (o: experimental measurements).



Evolution of the local skin friction coefficient along the flat plate (symbols: measurements).

Similarity of velocity profiles at different streamwise sections

Correctness of Blasius' law for the local skin friction coefficient

Test / theory correlation is validated

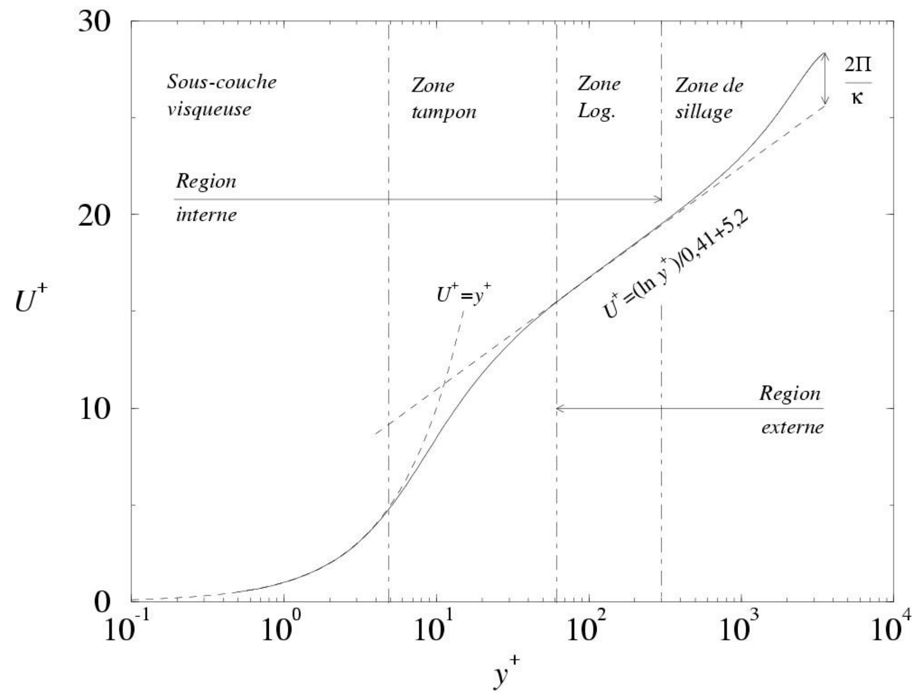
Validation criteria for simulations

Structure of the turbulent boundary layer

- Partitioning into 2 regions with distinct behavior
  - Internal region (near wall): partial similarity *via* viscous scales μ and τ_ω
  - External region (near boundary layer frontier): partial similarity via global scales δ and U_e
- Connection of the 2 regions by the so-called logarithmic zone

Reminder of notations

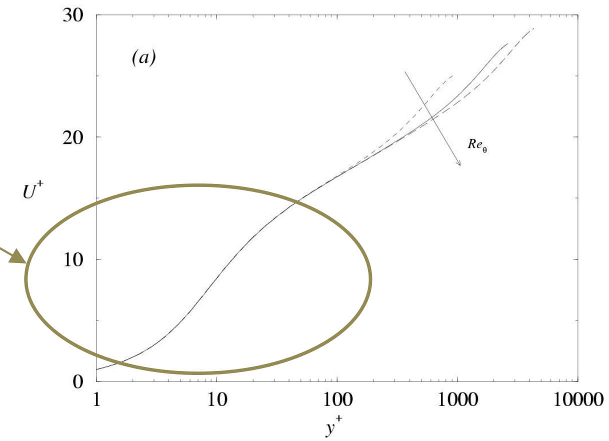
$$u_\tau = \sqrt{\tau_w / \rho}$$
$$U^+ = \overline{U}/u_\tau$$
$$y^+ = (u_\tau . y)/\nu$$



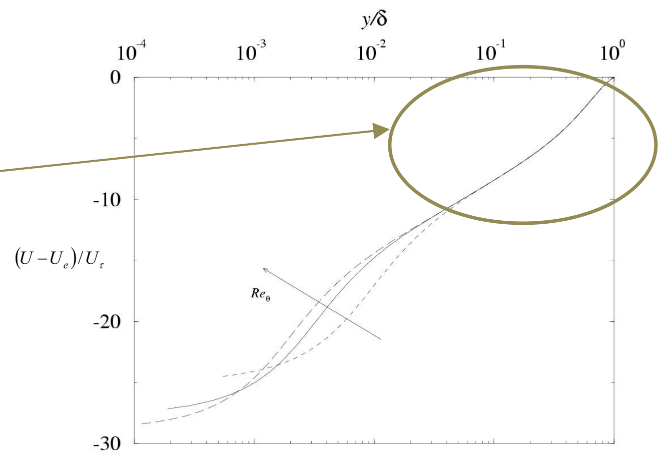
Structure of the turbulent boundary layer

❑ Revealing partial similarities

Varying the Reynolds number does not change the shape of the dimensionless velocity profile $U^+(y^+)$ in the inner region.



Varying the Reynolds number does not change the shape of the dimensionless velocity profile $[(U-U_e)/U_\tau](y/\delta)$ in the outer region.



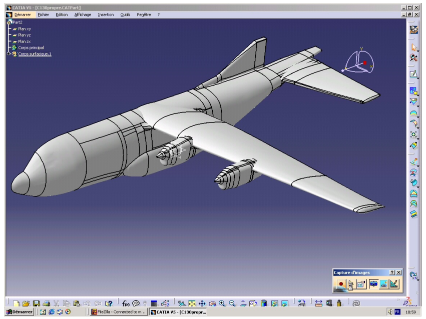
Want to learn more about turbulent boundary layers ?
Click here!

(or open additional_slides-CFD-BL.pdf)

General approach

Bury Yannick, Morton Scott and Charles Richard. "Experimental Investigation of the Flow Field in the Close Wake of a Simplified C130 Shape : A Model Approach of Airflow Influence on Airdrop." *26th AIAA Applied Aerodynamics Conference, Honolulu, Hawaii*. 2008.

Generate the CAD model

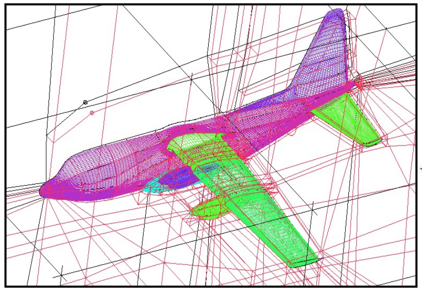


- Some geometrical simplifications may be required to conduct the study
- 2D vs 3D ? Full vs half geometry? *etc*.
- Some limitations may be necessary for the size of the computational domain, to be related with the choice of the boundary conditions

Reductionism principle…
but up to a reasonable limit
Click here !

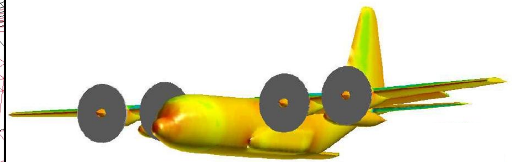
or go to https://arxiv.org/ftp/arxiv/papers/1109/1109.0214.pdf

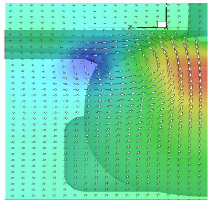
Meshing



- Which meshing strategy (poly/tetra/quad)
- A mesh convergence study is mandatory…
- To be associated with a time step convergence study in case the problem is unsteady

Computations





- Which initial/boundary conditions to impose, so to keep consistent with the problem
- Steady vs unsteady ?
- Which data to save

CFD/tests correlation

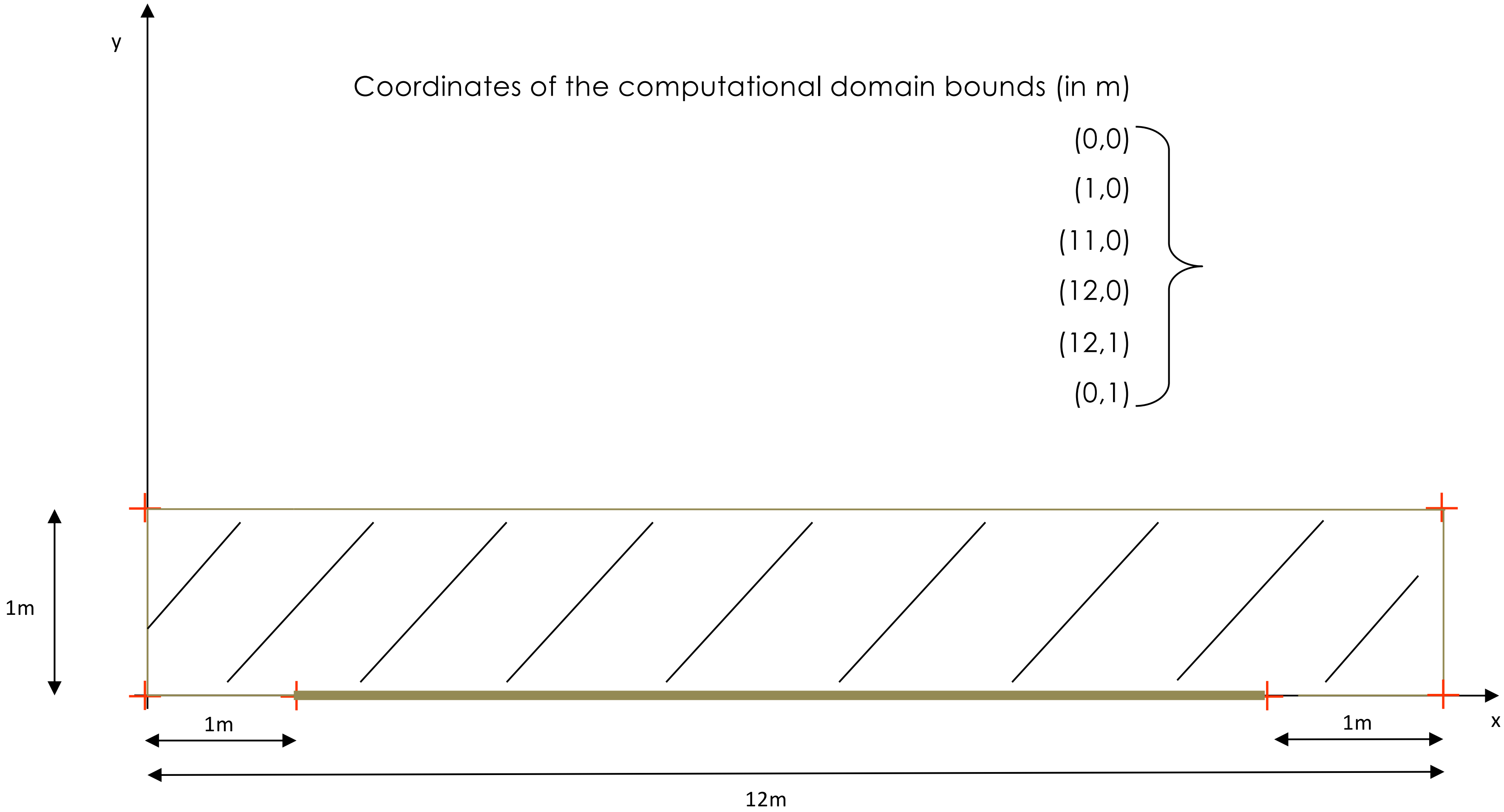
- Which metrics to use for the CFD validation
- Are these data (CFD/tests/theory) really comparable?

1st step: the CAD

- The 2D flat plate model / computational domain

Coordinates of the computational domain bounds (in m)

(0,0)
(1,0)
(11,0)
(12,0)
(12,1)
(0,1)



2nd step: the mesh

- Generate 2 meshes of increasing resolution

Coarse mesh

- first cell in the log zone: $50<y^+<500$.
- 10 cells, at least, in the boundary layer

Refined mesh

- first cell in the viscous sublayer: $y^+ \approx 1$.
- 20 cells, at least, distributed in the viscous sublayer and buffer zone

But…

How to evaluate y^+?

How to determine, *a priori*, boundary layer thicknesses and to distribute the cells in the mesh?

Meshing the boundary layer

- Choosing the y1 height of the first cell, so to fix y+ value
  - Estimate the local skin friction coefficient, *via* empirical or approximated relations
    - External aerodynamics → flat plate $\overline{C}_f / 2 \approx 0.0359\,\mathrm{Re}_L^{-0.2}$
    - Internal aerodynamics → flow in a tube $\overline{C}_f / 2 \approx 0.039\,\mathrm{Re}_D^{-0.2}$
  - Compute the friction velocity $u_\tau \equiv \sqrt{\tau_w / \rho} = U_e \sqrt{\overline{C}_f / 2}$
  - Compute the required height from the wall y1, depending on the targeted y^+-based wall treatment (external region *vs* sublayer)
    - first cell in the log zone $y^+ > 50$ $y_1 = 50\nu / u_\tau$
    - first cell in the viscous sublayer $y^+ \approx 1$ $y_1 = \nu / u_\tau$
  - Run the computation, then post process the results to confirm (or not) y^+ values
  - Close-to-wall remeshing may be requested, so to satisfy the targeted y^+ criterion

Meshing the boundary layer

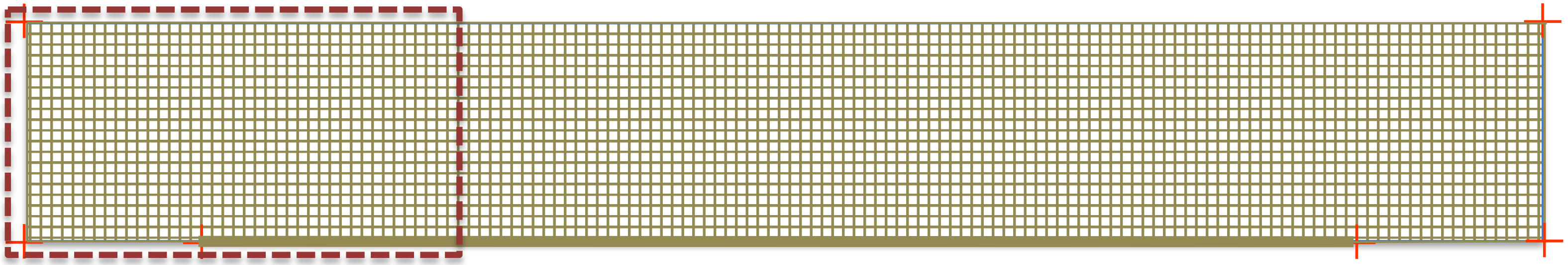
❑ Application to the flat plate

- ❑ General parameters
  - ❑ $U_0 = 1m/s$
  - ❑ $L_{plate} = 10m$ ➔ $Re_L \approx 6.66.10^5$ (a bit too large for a strictly laminar flow)
  - ❑ $\nu_{air} = 1.5.10\text{-}5\ m^2/s$
- ❑ y1 value for $y^+=50$: $1.5.10^{-2}m$
- ❑ y1 value for $y^+=1$: $3.10^{-4}m$
- ❑ Which cell distribution to impose in the boundary layer
  - ❑ In practice, on complex systems, boundary layer thickness is difficult (or impossible) to estimate *a priori*
  - ❑ Thinking in terms of orders of magnitude is a reasonable approach
    - ❑ Boundary layer thickness: a few to a few tens of a cm…
    - ❑ Viscous sublayer: a few tenth's of a mm to a few mms…

Being pragmatic by adopting a « realistic » approach, based on these very first orders of magnitude, for want of anything better !

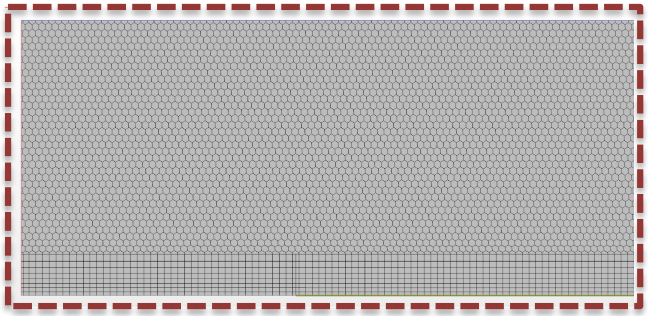
Generate the mesh

❑ Global parameters



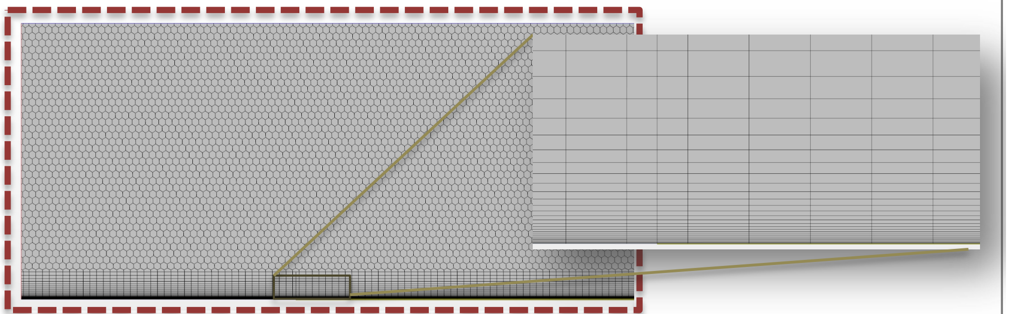
Coarse mesh (~ 20000 cells)

- ❑ first cell in the log zone: height ~15mm.
- ❑ ~10 cells in the boundary layer
- ❑ cell-to-cell growth factor in the BL: 1.1
- ❑ out-of-BL uniform distribution of the cells
- ❑ out-of-BL cell size: ~ 25mm



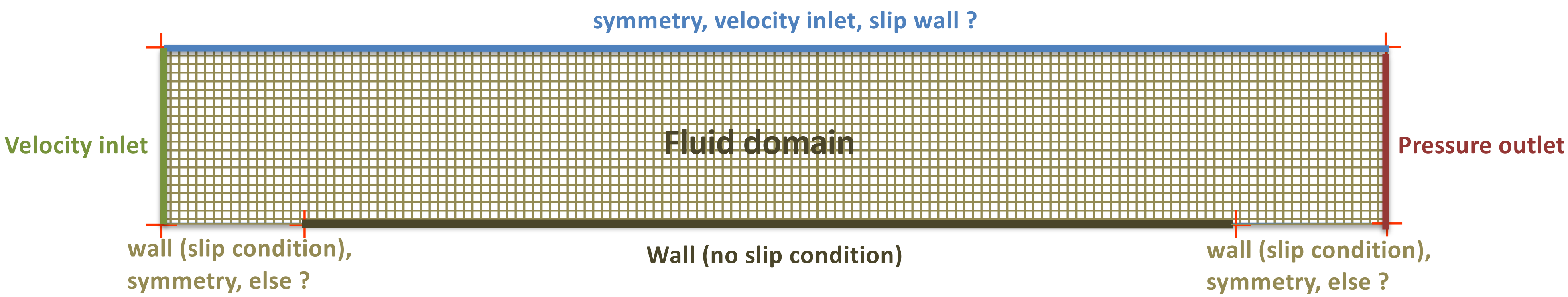
Refined mesh (~35000cells)

- ❑ first cell in the viscous sublayer: height ~0.2mm.
- ❑ 30 cells distributed in the viscous sublayer and buffer zone
- ❑ cell-to-cell growth factor in the BL: 1.1
- ❑ out-of-BL uniform distribution of the cells
- ❑ out-of-BL cell size: ~ 25mm



Defining the boundary conditions

❑ About the bounds



- ❑ Discuss the choice of the boundary conditions
- ❑ Test their influence on the computation behaviour and results

Implementing the computation

- Create the various meshes
  - Right click on 3D-CADs model, New, then right click on the created 3D-CAD Model 1, edit, then create the sketch by right click in plane XY, then extrude it (a few mm's is enough), Close 3D-CAD. Right click on Body1 (in Bodies directory) and choose New geometry part
  - Once the geometry is transferred in Parts, don't forget to split the unique surface into as many surfaces as required for the definition of the boundary conditions
  - Assign part to region and define the boundary conditions
  - Then, 2 options: either "generate a 3D mesh (create a mesh continuum to do so), then convert to 2D" (see previous classes), or "right click on Body1, Create mesh operation/mesh/Badge for 2D mesh and Automated Mesh (2D)" from right click on the previously created geometry part (demonstration in this class)

- Create the Physics continuum associated with the so-created region
  - Models should be: 2D, steady, Segregated flow, constant density, gas, laminar or turbulent (if so, k-omega SST, then choose all y+. The solver will determine which is the best approach for the wall treatment, depending on the actual y+ values)
  - Impose the initial conditions and velocity/pressure/turbulence levels (if considered) in the previously defined boundary conditions
  - Run until convergence

Flow simulated as fully turbulent when « turbulent model » activated

laminar – turbulent transition still harsh to predict on a CFD code

Analyzing the CFD results

- First of all…

Once the computation is converged, it is necessary to estimate which confidence to be given to the results. To do so, several "basic" checks can be carried out:

  - *e.g.*, check the flow balances between the different inlets and outlets of the computational domain (report mass flows, fluxes)
  - Ensure that the orders of magnitude of the different computed variables (pressure, velocity, temperature, density, *etc.*) are consistent with the physics: are there any outliers?
  - Is the flow structure consistent with the physics: are there any outliers?

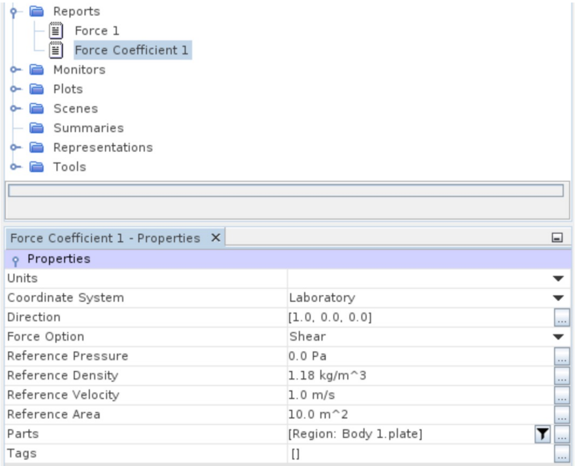
Analyzing the CFD results

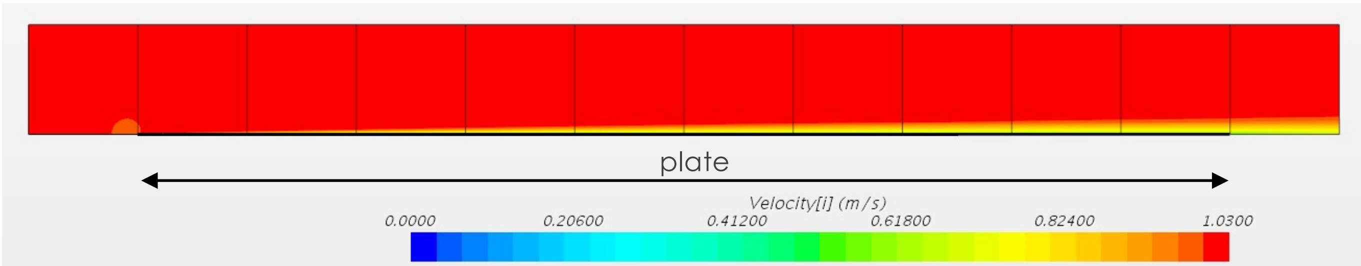
- Analysis based on velocity profiles and wall shear stress

  - Fundamentals of the theory/tests/computations correlation for this canonical configuration (see slides 6-8)

  - Create drag force and drag force coefficient reports for the flat plate. Plot the skin friction coefficient C_f and wall shear stress τ_ω along the flat plate.

  - Create derived parts (normal-to-X planar sections) at various X-locations, from $x=x_0=1m$ (flat plate leading edge) to x=11m (flat plate trailing edge), equally distributed every 1m, to plot velocity profiles. Then create a plot and select all your derived parts as an input.



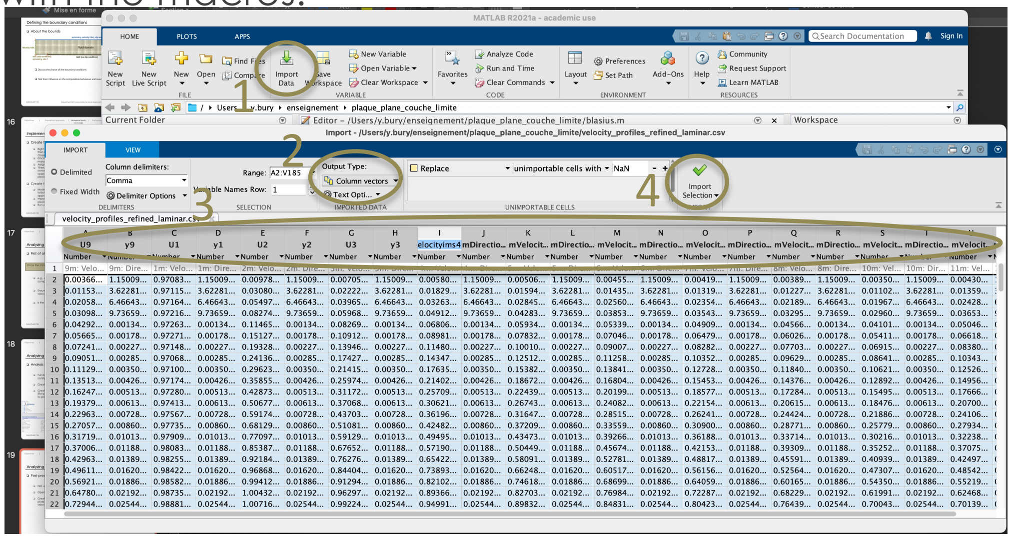


Analyzing the CFD results

❑ Post process on Matlab (or Python)

- ❑ First, export your StarCCM results. Run reports and note the values. Plot velocity profiles (all on the same plot), skin friction coefficient C_f and wall shear stress τ_ω along the plate: right click on the different plots (in the tree menu), export as csv.

- ❑ Open Matlab. For each of the computed flow case, import as column vectors the csv files of the velocity profiles, skin friction coefficient and wall shear stress distribution along the flat plate. Name the columns intelligibly (yi, Ui for the velocity profiles, where i stands for a given xi profile (derived part)) to be used with the macros.



Analyzing the CFD results

❑ Post process on Matlab (or Python)

laminar

Dimensionless velocity profiles

```
%when importing the csv file of the velocity profiles at various cross
sections along the flat
%plate, name yi the column depicting the distance from the wall for
%velocity profile i, and Ui the associated velocity

% import Blasius profiles
%uiopen('/Users/y.bury/Desktop/Blasius.csv',1); rename the first
column
%eta_blasius and the second column fprime_blasius

nu=0.000015;
nombre_profils=11;
X0=1;
figure;hold on

for i=2:nombre_profils
    eval(['[a,b]=sort(y',int2str(i),',''ascend'');'])
    eval(['eta',int2str(i),'=(y',int2str(i),'(b))*sqrt(max(U',int2str(
i),')/(nu*(',int2str(i),'-x0)));'])
    eval(['plot(eta',int2str(i),',U',int2str(i),'(b)/max(U',int2str(i)
,'),''+-'');']);
end

plot(eta_blasius,fprime_blasius,'o')
```

Skin friction coefficient vs blasius law

Import your coarse and refined mesh-based Cf distribution,
and compare with the Blasius law

turbulent

Dimensionless velocity profiles

```
%when importing the csv file of the velocity profiles at various cross sections
along the flat
%plate, name yi the column depicting the distance from the wall for
%velocity profile i, and Ui the associated velocity
%when importing the csv file of the wall shear stress tau_p along the flat plate,
name x the
%column depicting the coordinates along the plate, and tau_p the column
%depicting the wall shear stress

dx=mean(diff(x));
nu=0.000015;
rho=1.18415;
nombre_profils=11;

%inner region partial similarity
figure;hold on
i=3;
    clear a b c
    eval(['c=find(abs(x-',int2str(i),')<dx);']);
    eval(['u_tau',int2str(i),'=sqrt(tau_p(min(c))/rho);'])
    eval(['U_plus',int2str(i),'=U',int2str(i),'/u_tau',int2str(i),';'])
    eval(['yplus',int2str(i),'=y',int2str(i),'*u_tau',int2str(i),'/nu;'])
    eval(['semilogx(yplus',int2str(i),',U_plus',int2str(i),')'])
```

Vary manually i from 2 to 11 and copy paste the following lines (looping on i malfunctions) !!

```
%outer region partial similarity
figure;hold on
i=5;

    eval(['[a,b]=sort(y',int2str(i),',''ascend'');'])
    eval(['c=find(abs(x-',int2str(i),')<dx);']);
    eval(['u_tau',int2str(i),'=sqrt(tau_p(min(c))/rho);'])
    eval(['y_umax',int2str(i),'=find(abs((U',int2str(i),'(b)-
max(U',int2str(i),'))/max(U',int2str(i),'))<0.01);'])
    eval(['indice_y_umax',int2str(i),'=min(y_umax',int2str(i),');'])
    eval(['semilogx(y',int2str(i),'(b)/y',int2str(i),'(b(indice_y_umax',int2str(i
),')),(U',int2str(i),'-max(U',int2str(i),'))/u_tau',int2str(i),');'])
```

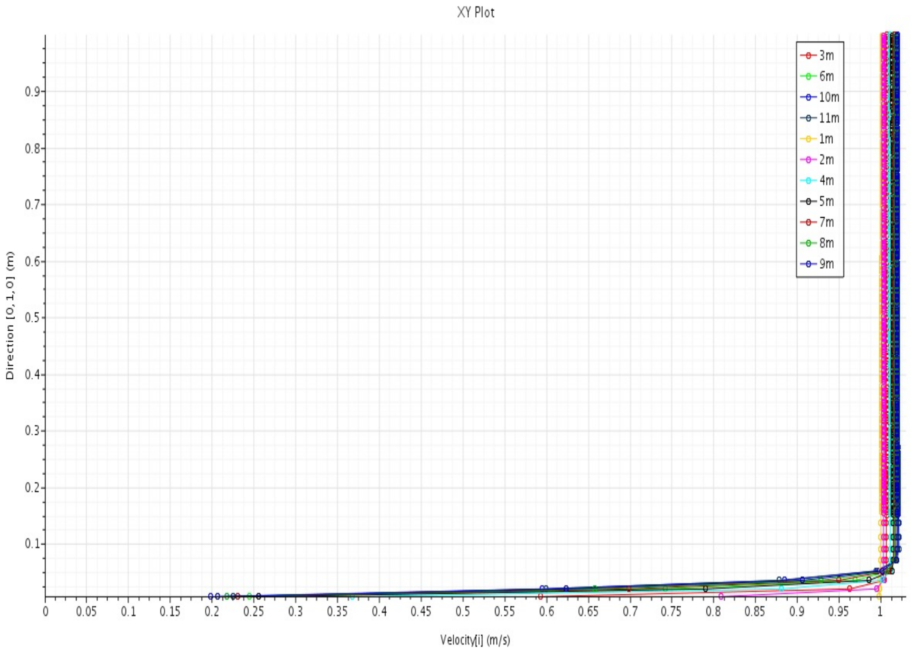
Vary manually i from 2 to 11 and copy paste the following lines (looping on i malfunctions) !!

Qualitative analysis of the results

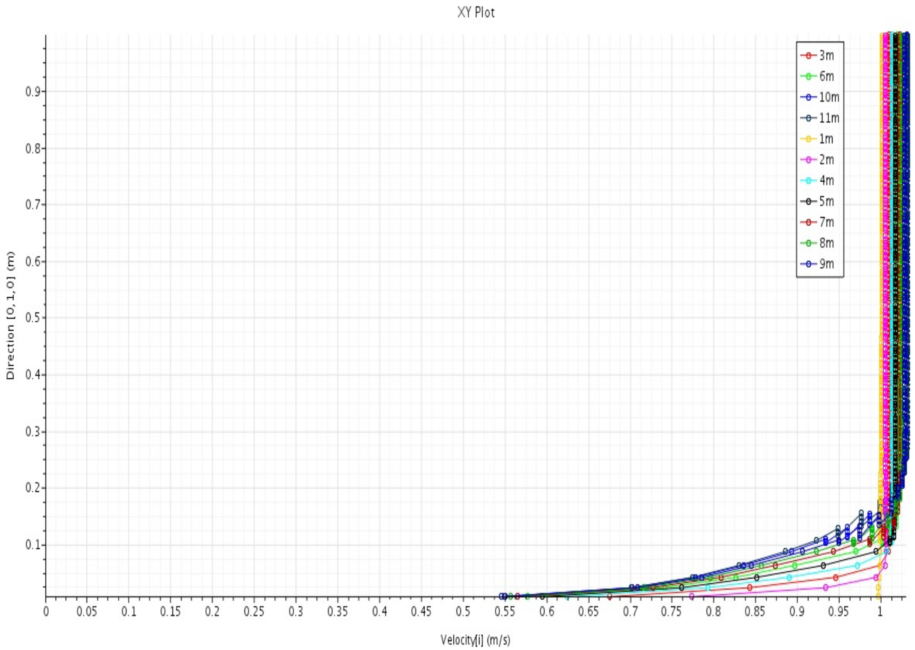
❑ Evolution of the velocity profiles along the flat plate

Coarse mesh

laminar



turbulent

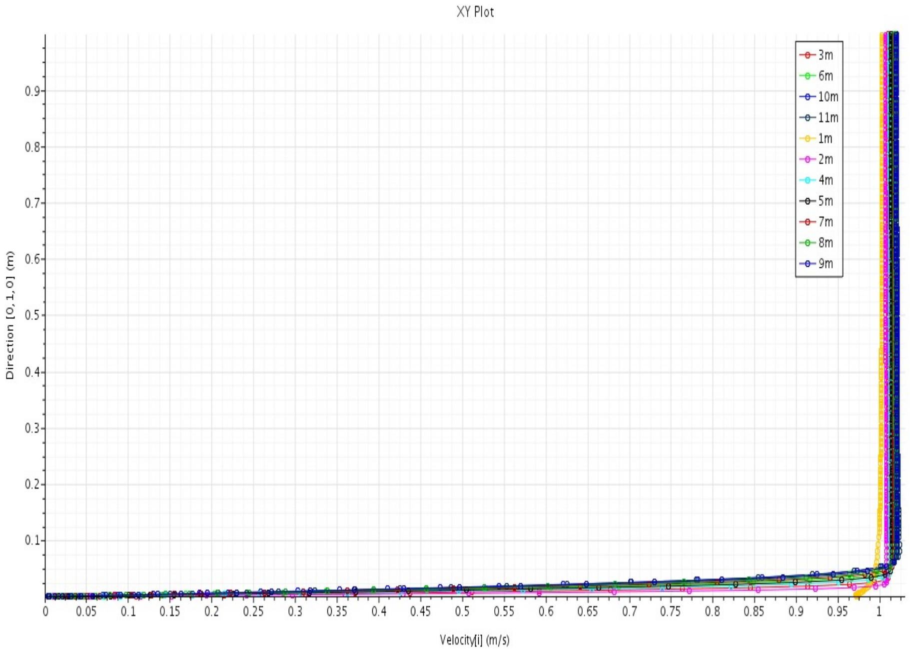


Qualitative analysis of the results

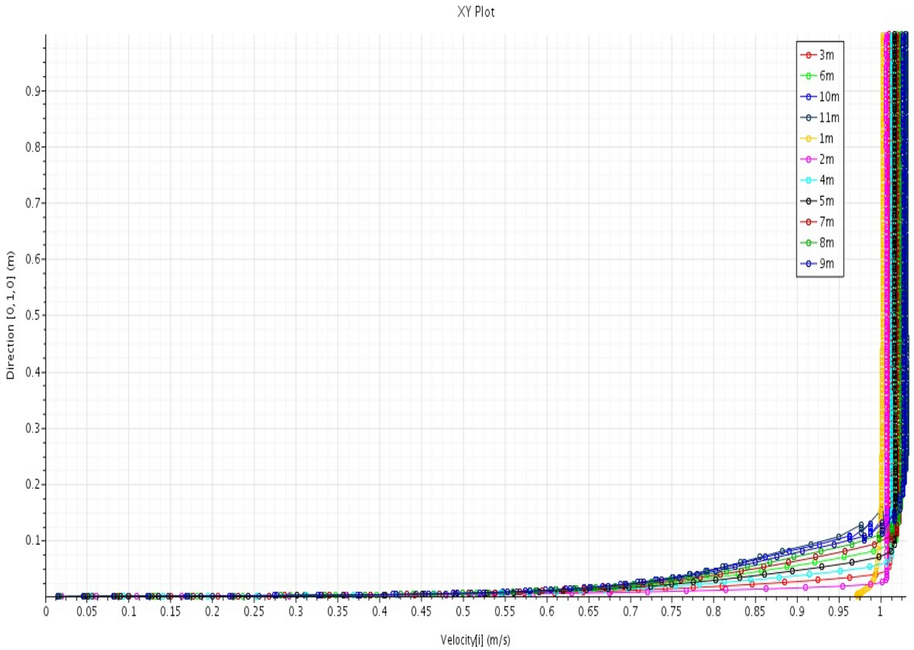
❑ Evolution of the velocity profiles along the flat plate

Refined mesh

laminar



turbulent



Qualitative analysis of the results

- Resulting wall shear stress coefficient on the flat plate

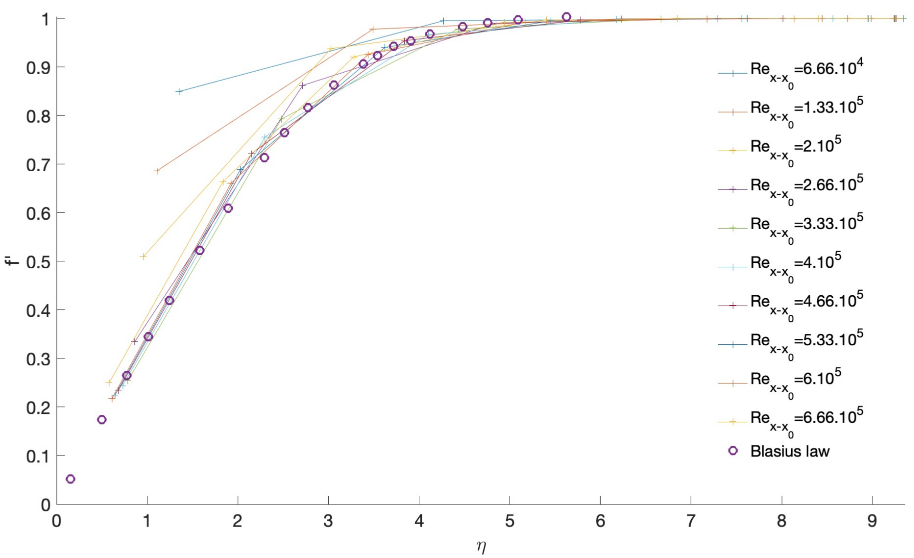
	LAMINAR			TURBULENT	
mesh resolution	coarse	refined	Blasius	coarse	refined
1000 x C_f	1.77	1.74	1.62	5.2	4.5
deviation from ref.	9.2%	7.4%		15.5%	

Quantitative analysis of the results

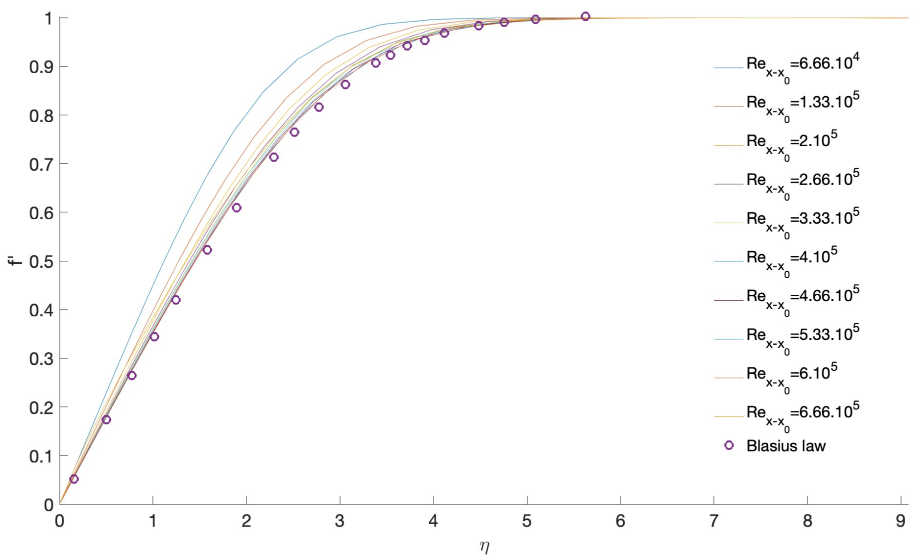
❑ Laminar flow

- ❑ Evolution of the dimensionless velocity profiles along the flat plate

Coarse mesh



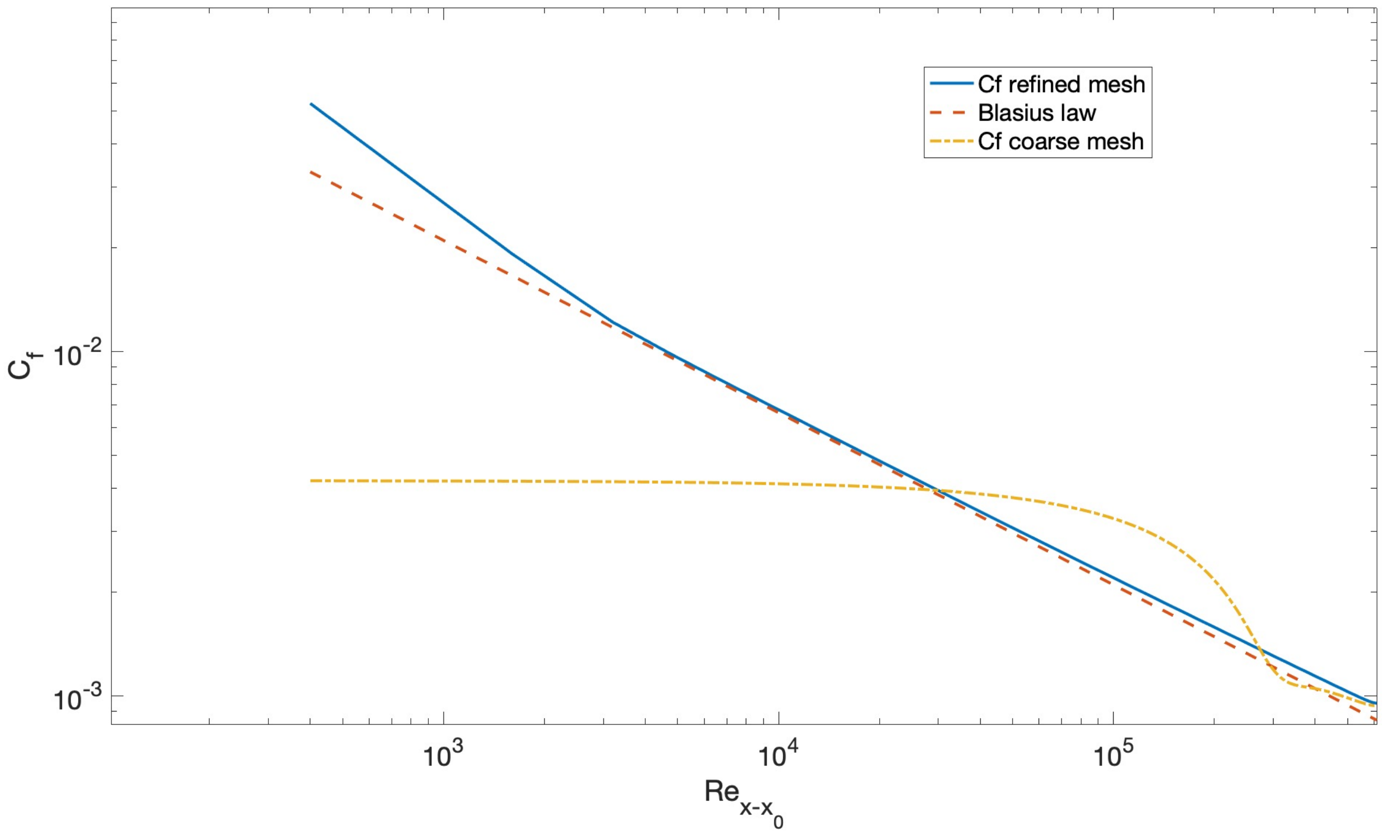
Refined mesh



Quantitative analysis of the results

❑ Laminar flow

- ❑ Evolution of the skin friction coefficient along the flat plate

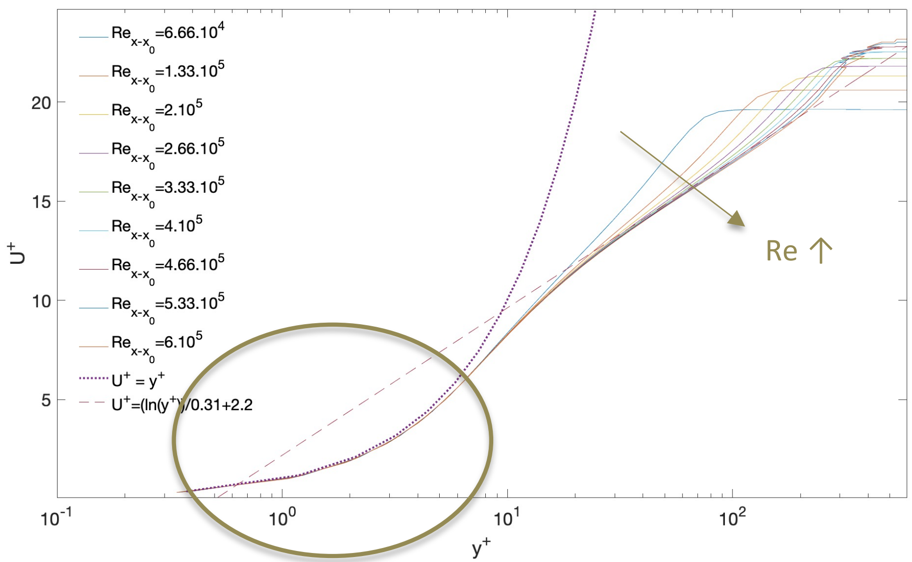


Quantitative analysis of the results

❑ Turbulent flow

- ❑ Evolution of the dimensionless velocity profiles along the flat plate (refined mesh)

Inner region partial similarity



Outer region partial similarity