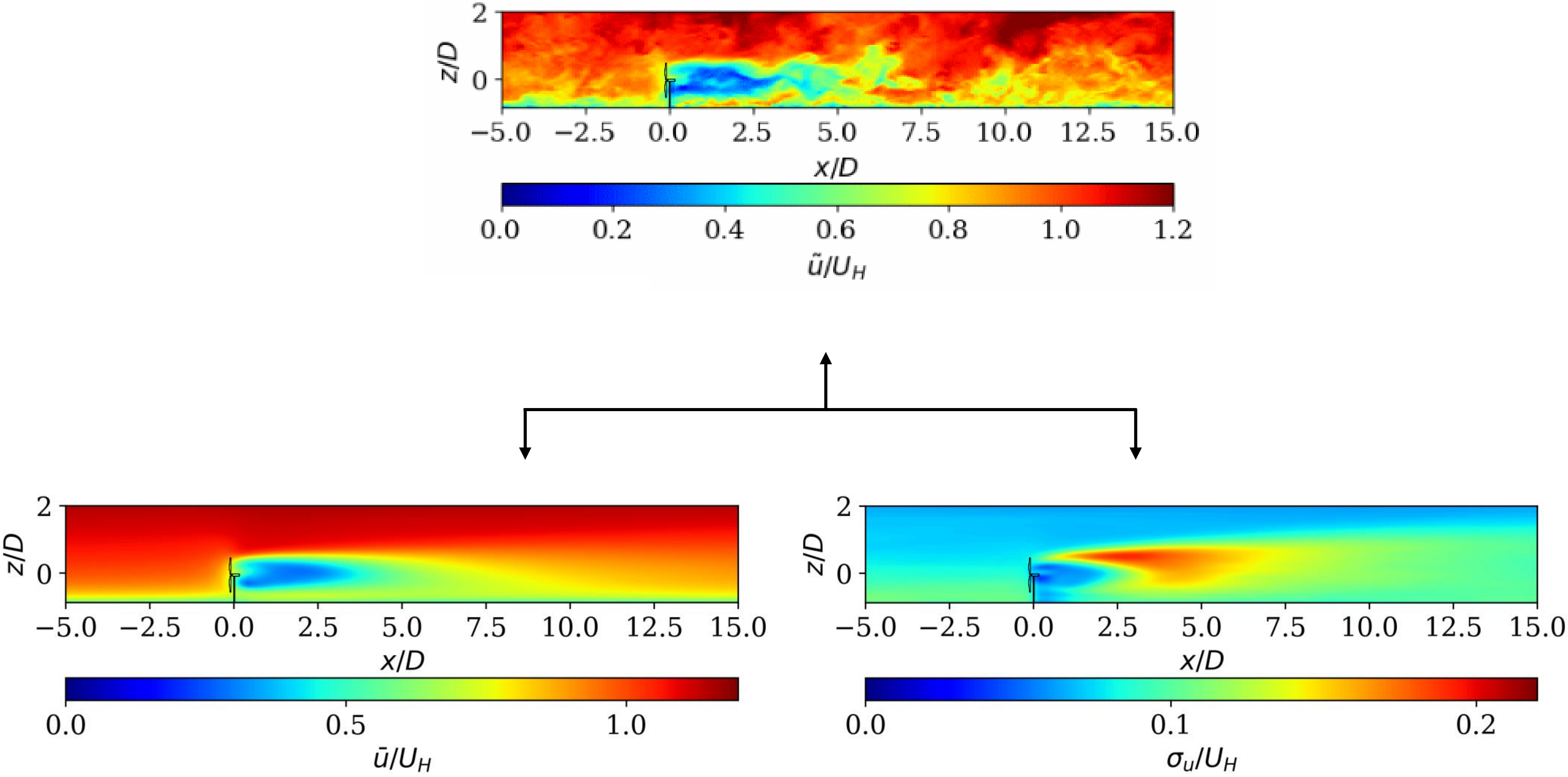
Introduction to the project

Computational Fluid Dynamics (CFD): A Simulation Tool for Environmental Transport Phenomena

Date: 13.09.2024
Presenter: Tristan Revaz
Contact: tristan.revaz@epfl.ch

General introduction and motivation


z/D
2
0
−5.0
−2.5
0.0
2.5
5.0
7.5
10.0
12.5
15.0
x/D
0.0
0.2
0.4
0.6
0.8
1.0
1.2
$\tilde{u}/U_H$
z/D
x/D
0.0
0.5
1.0
$\bar{u}/U_H$
z/D
x/D
0.0
0.1
0.2
σ_u/U_H

- The project will involve using the flow-modelling software ANSYS CFD (including Workbench, SpaceClaim, Meshing, Fluent and CFD-Post) in order to study **one of the 4 proposed topics in groups of 2-3 people**.

- The **Ansys 2021 R1** software is available via VDI (vdi.epfl.ch, VMware Horizon Client/HTML, ENAC-SSIE-WIN). The computer room N° GR B0 01 is the reference room for the project sessions.

- Please verify your log-in and your access to the abovementioned VDI and computer room. If you have any problem with access, please contact tristan.revaz@epfl.ch as soon as possible.

Topic: Choose from one of the 4 following topics (all must be modelled in **2D**):

- **Flows in rivers**: flows in river junctions, river beds, etc. From the base case: parametric studies, distribution of pollutant or temperature in the flow, application of theoretical concepts, etc.

- **Flows over topography**: flows over hills, in valleys, etc. From the base case: parametric studies, distribution of pollutant or temperature in the flow, application of theoretical concepts, etc.

- **Street canyon**: flows in cities, etc. From the base case: parametric studies, distribution of pollutant or temperature in the flow, application of theoretical concepts, etc.

- **Wind turbine/farm Flows**: From the base case: parametric studies, effect of layout, effect of base flow, application of theoretical concepts, etc.

Project Tasks

1. **Step-by-step tutorial (individual) → screenshots and sim files**

   → available W2 (September 20), 2 Q&A sessions W3-W4 (27 September and 4 October), deadline W5 (October 11)

2. **Project exercise (group) → screenshots and sim files**

   → available W6 (October 18), 2 Q&A sessions W7-W8 (1 November and 8 November), deadline W9 (November 15)

3. **Main project (group) → one report per group (7-10 pages)**

   → The 4 last weeks W11-W14, class time will be dedicated to the project
   → Deadline for the report January 10

Grading (RECALL: 30% of the overall course grade)

1. **Step-by-step tutorial 10% (deadline October 11)**

2. **Project exercise 20% (deadline November 15)**

3. **Main project 70% (deadline January 10)**

    - **10%**: A clearly stated research question for the main project
    - **30%**: Comprehensive description of your model, featuring: logical geometry to represent the problem, appropriate mesh to discretize the geometry, numerical methods well-suited to the problem, key results appropriately post-processed
    - **20%**: Discussion of results, link to initial question, limitations of the approach
    - **10%**: Quality of presentation: clearly and concisely expressed ideas, well presented results, organization/layout

Detailed Timeline:

- **W1 - September 13 :** Introduction to the project
- **W2 - September 20 :** Tutorial available
- **W3 - September 27 :** Q&A (17-18H+) [Tutorial + Group/Topic]
- **W4 - October 4 :** Q&A (17-18H+) [Tutorial + Group/Topic]
- **W5 - October 11 :** Tutorial submission deadline in Moodle by 23:59. The groups and topics must be submitted to tristan.revaz@epfl.ch.
- **W6 - October 18 :** Project exercise available
- **W7 - November 1 :** Q&A (17-18H+) [Project Exercice + Main Project Definition]
- **W8 - November 8 :** Q&A (16-18H+) [Project Exercice + Main Project Definition]
- **W9 - November 15 :** Project exercise submission deadline to your TA by email by 23:59.
- **W11-14 - November 29, December 6,13,20 :** Class time will be dedicated to the project
- **January 10**: Final report submission deadline to your TA by email by 23:59.

Introduction to the project: Organization

Topic: Choose from one of the 4 following topics (all must be modelled in **2D**):

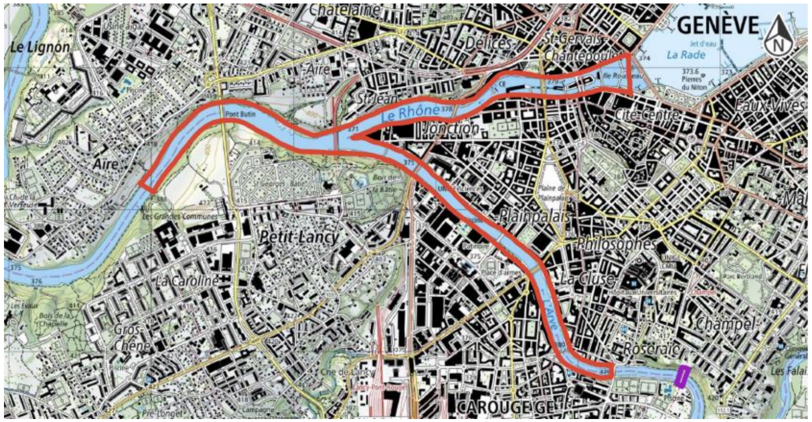
- **Flows in rivers**: flows in river junctions, river beds, etc. From the base case: parametric studies, distribution of pollutant or temperature in the flow, application of theoretical concepts, etc.

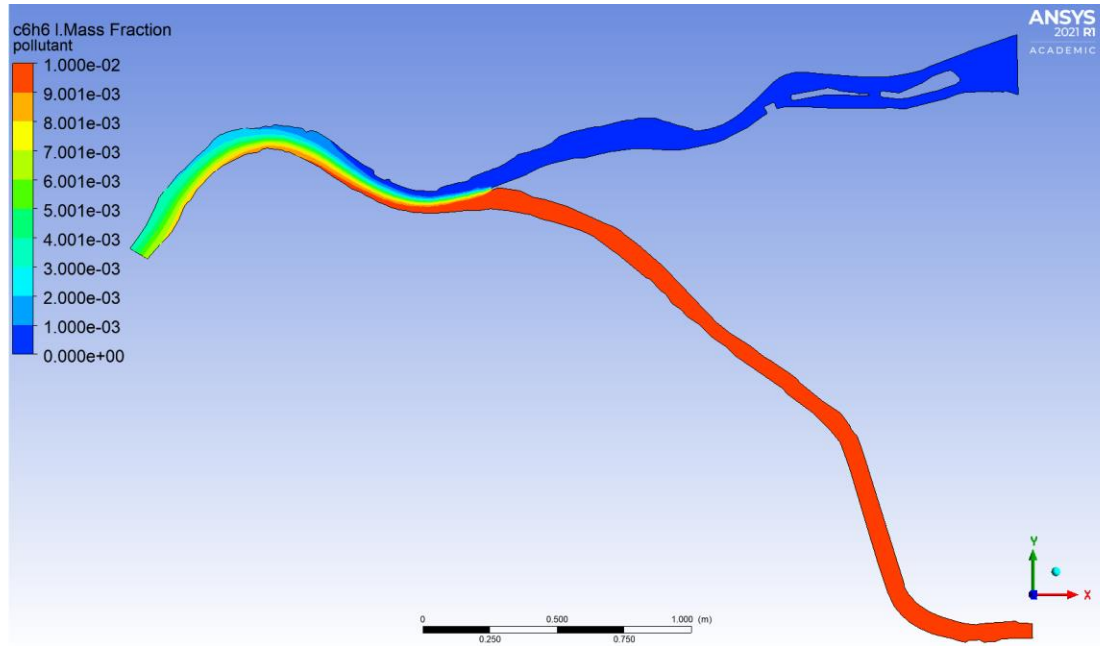
- **Flows over topography**: flows over hills, in valleys, etc. From the base case: parametric studies, distribution of pollutant or temperature in the flow, application of theoretical concepts, etc.

- **Street canyon**: flows in cities, etc. From the base case: parametric studies, distribution of pollutant or temperature in the flow, application of theoretical concepts, etc.

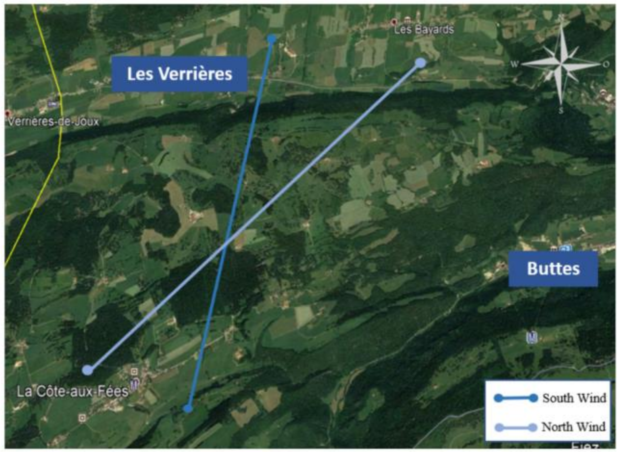
- **Wind turbine/farm Flows**: From the base case: parametric studies, effect of layout, effect of base flow, application of theoretical concepts, etc.

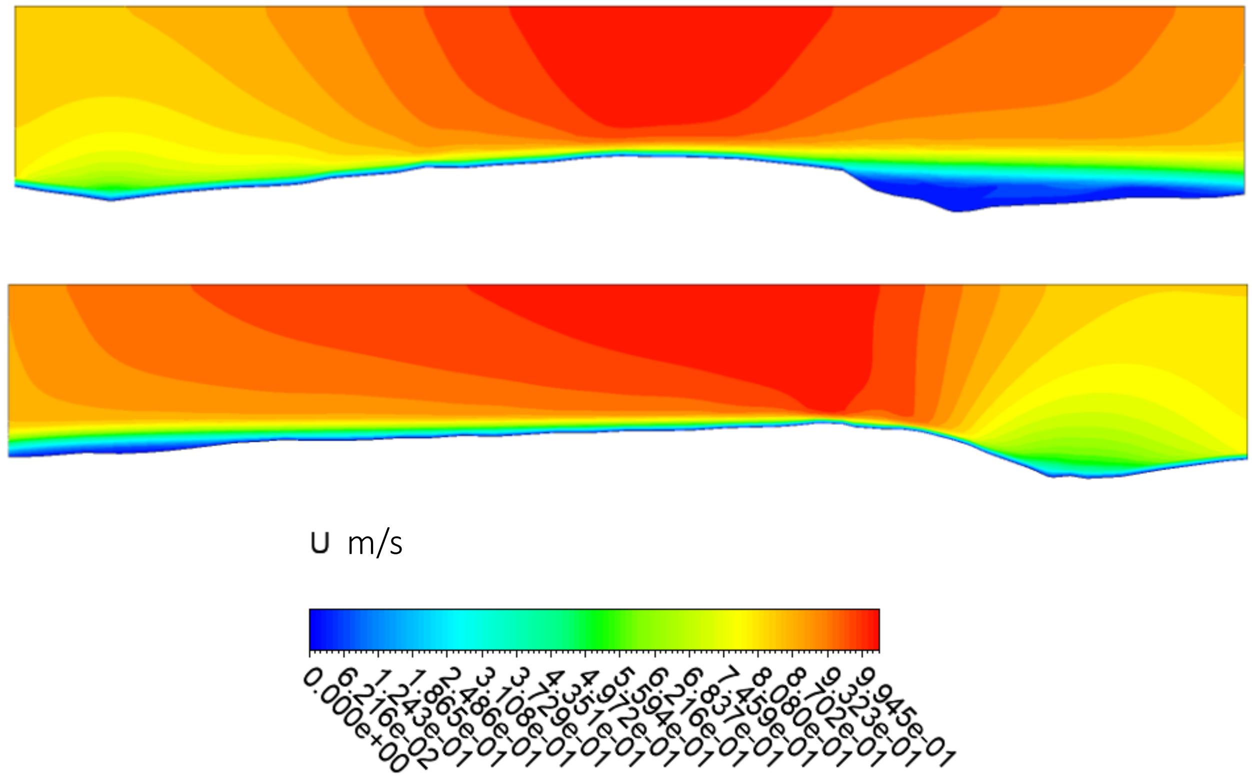
Flows in rivers



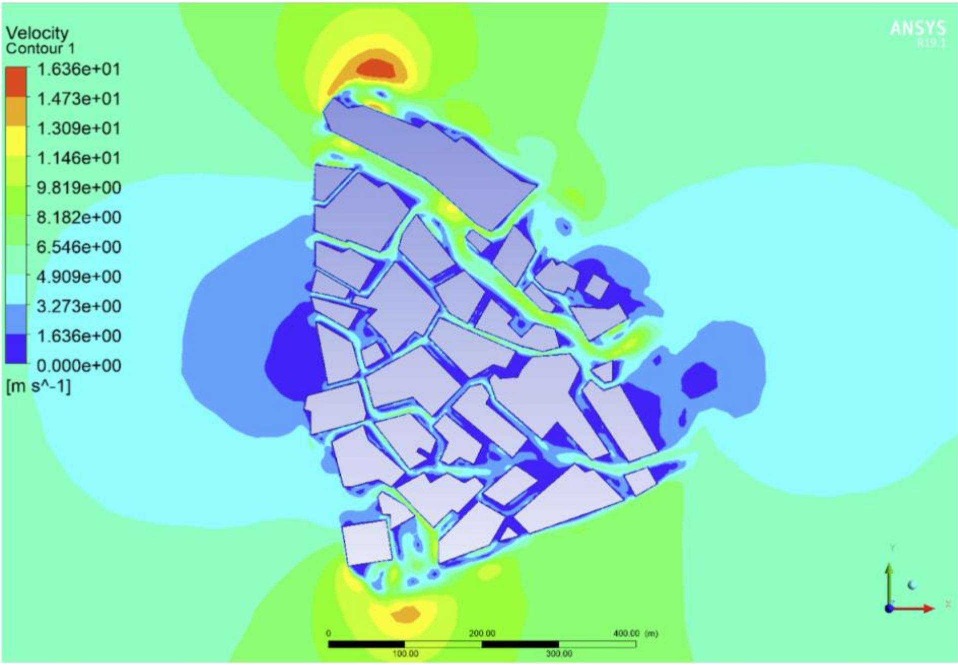


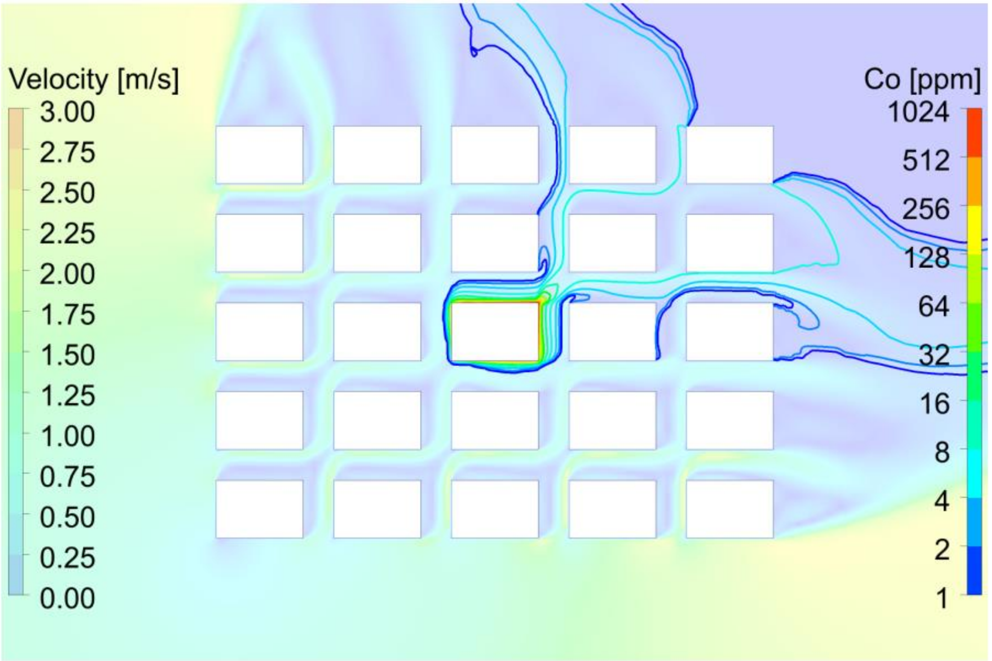
Flows over topography





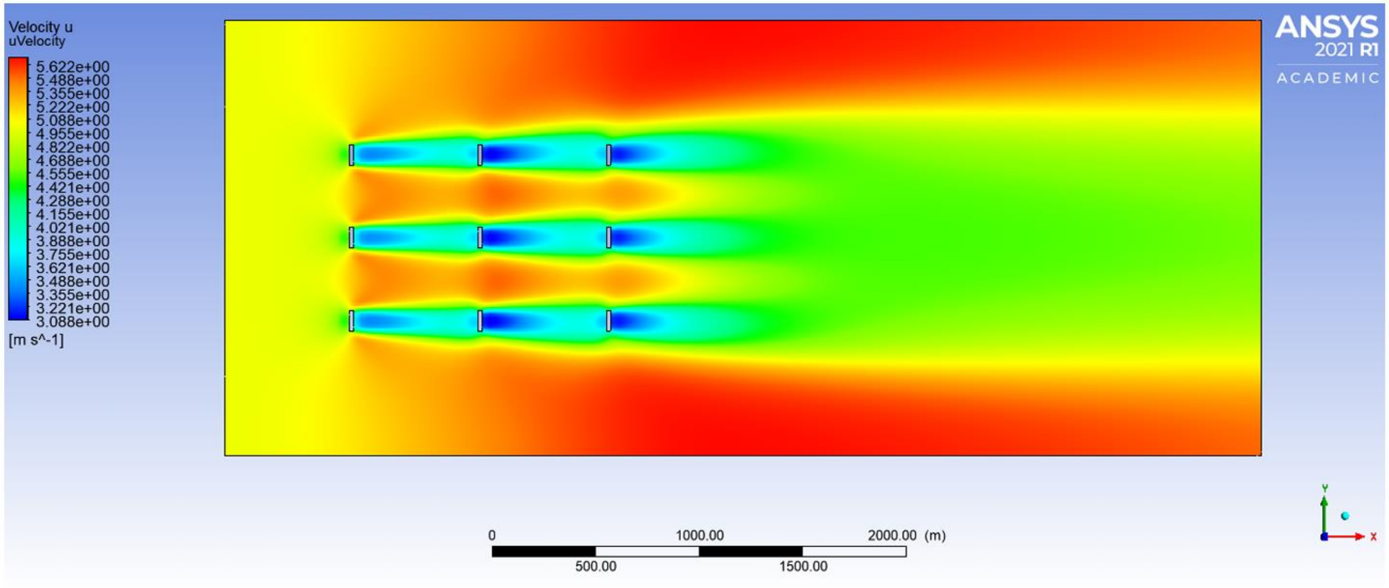
Street canyon





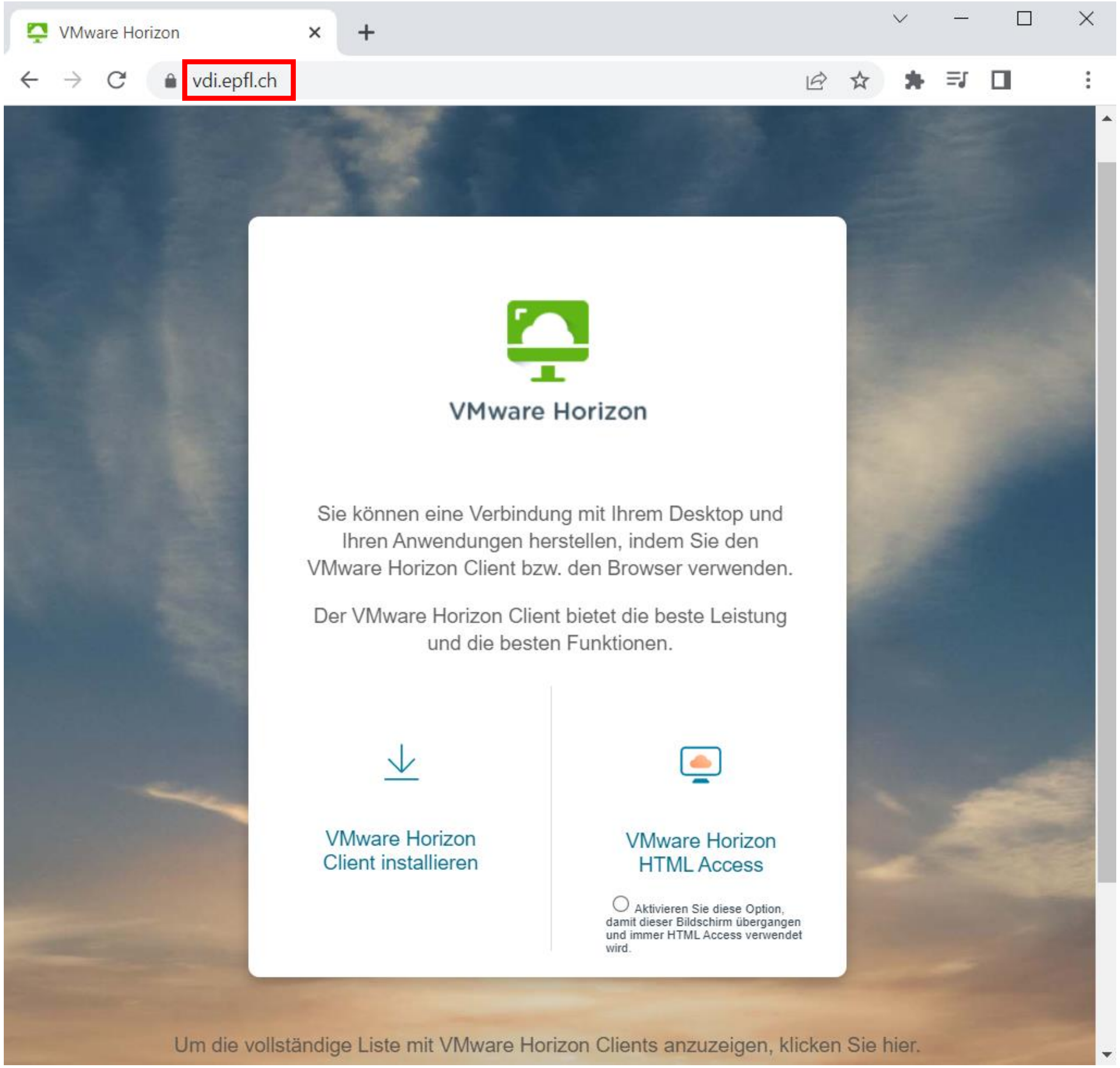
Introduction to the project: Topics

Wind turbine/farm

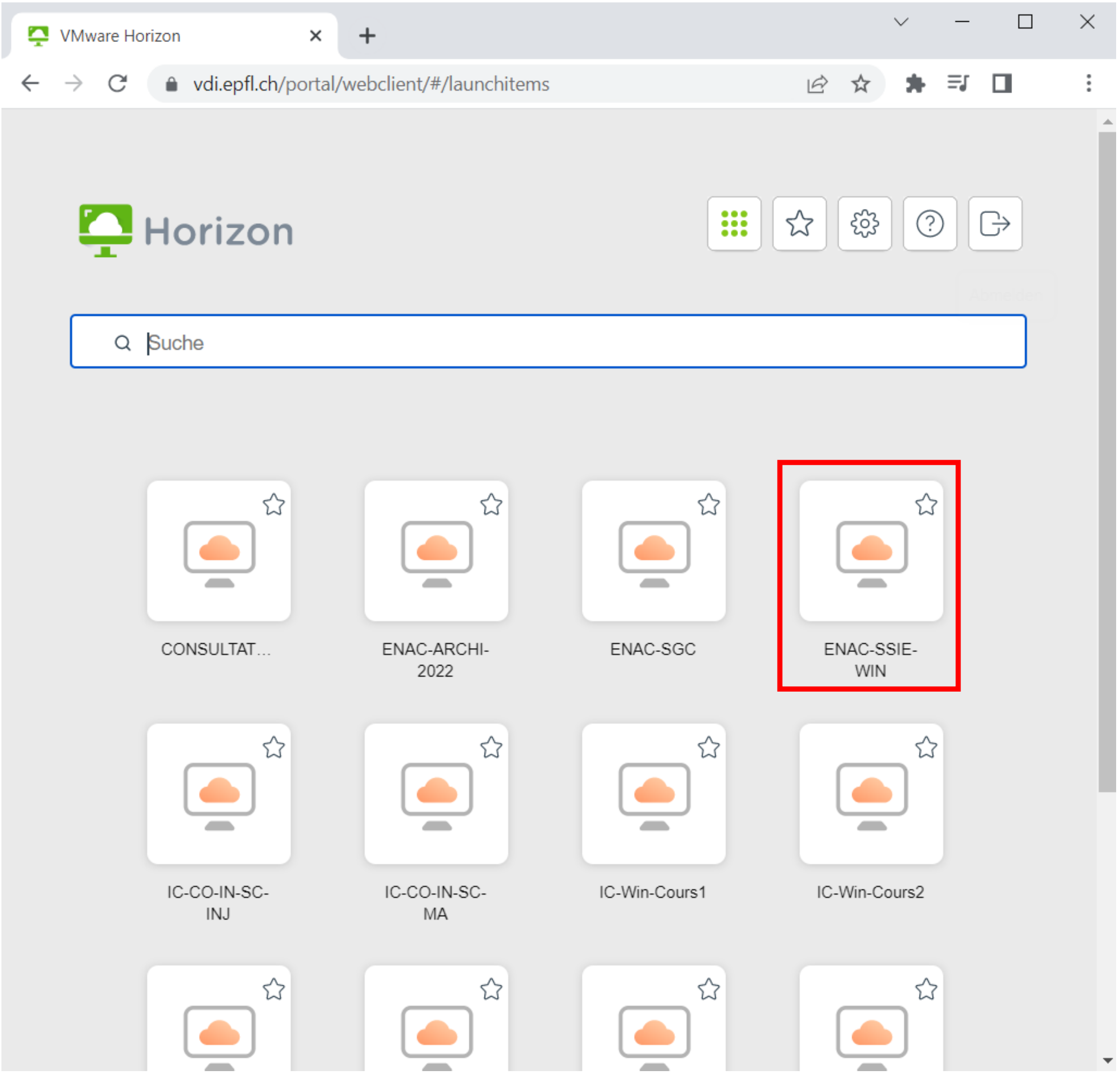


Turbine	Power
Central Turbine 1	1
Central Turbine 2	0.6669
Central Turbine 3	0.6866
Lateral Turbine 1	0.9841
Lateral Turbine 2	0.6597
Lateral Turbine 3	0.6781

Introduction to the project: VDI



Introduction to the project: VDI



Introduction to the project: VDI

Information

Informations importantes !

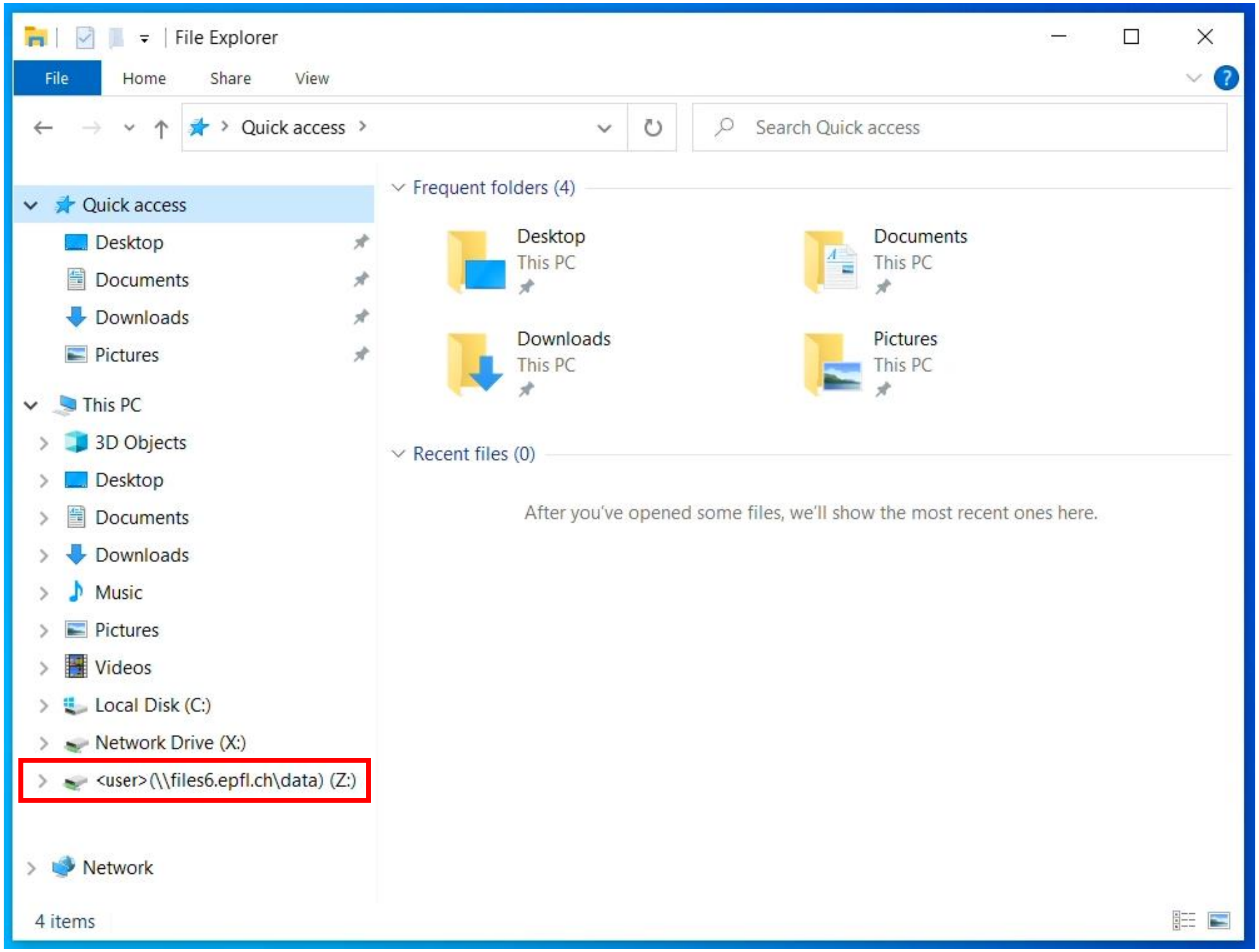
Les machines VDI étant réinitialisées à chaque démarrage, il est impératif de sauvegarder vos fichiers uniquement sur votre espace personnel (le lecteur Z:), et de ne rien stocker sur le bureau!
Tout fichier stocké en local sera définitivement perdu.

ENAC-IT / Sept. 2022

Pour toute question / remarque / problème sur les VDI du pool ENAC-SSIE-WIN, contactez enacit@epfl.ch

Ok

Introduction to the project: VDI



The 5 steps of a CFD project:

0. **Analyze the flow problem**
1. **Design the geometry and the flow domain**
2. **Generate the mesh**
3. **Setup the solver and run the simulation**
4. **Extract the important results**

0. Analyze the flow problem

1. Design the geometry and the flow domain
2. Generate the mesh
3. Setup the solver and run the simulation
4. Extract the important results

ANSYS Workbench

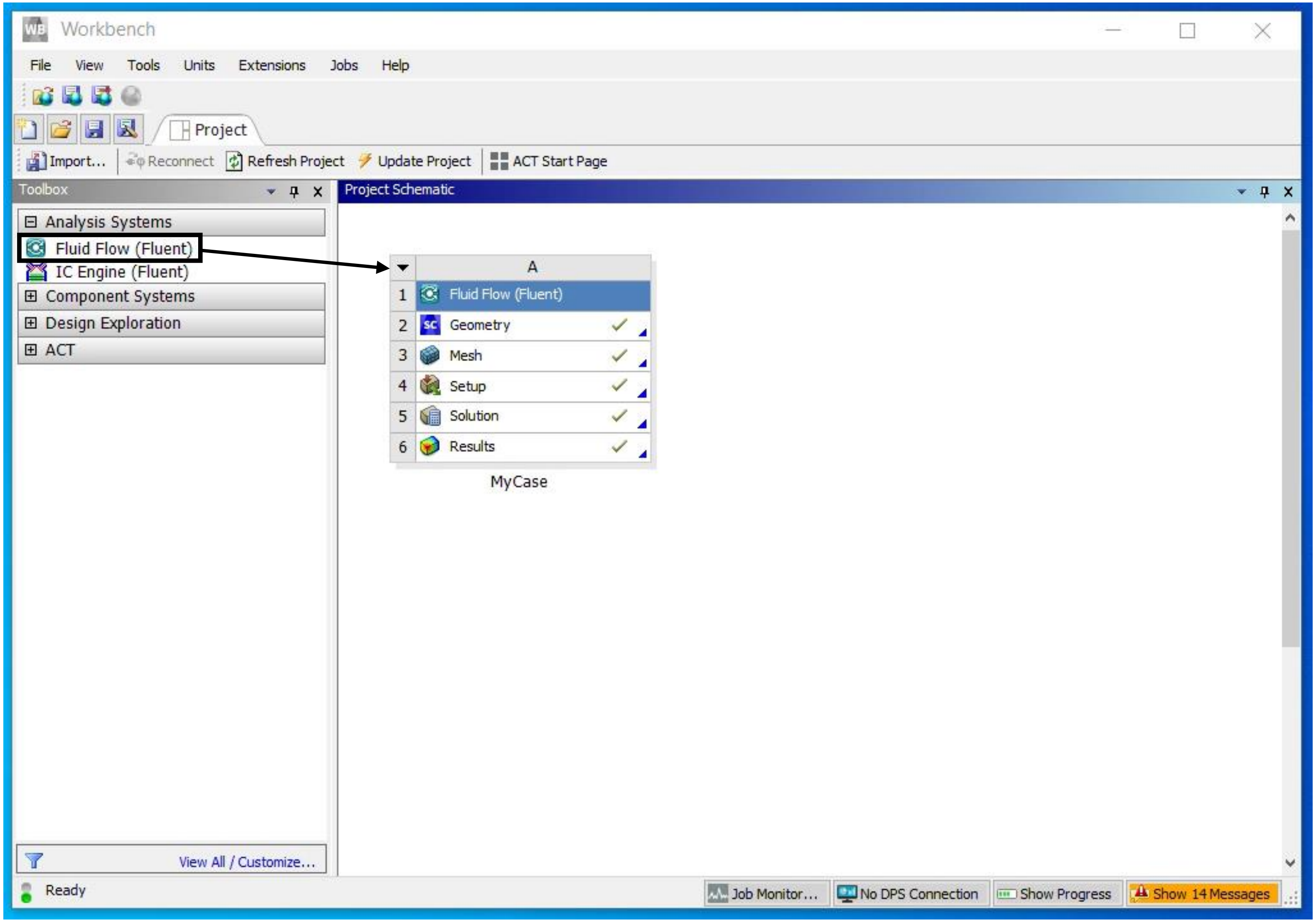
1 SPACECLAIM CORPORATION

2 Ansys Meshing

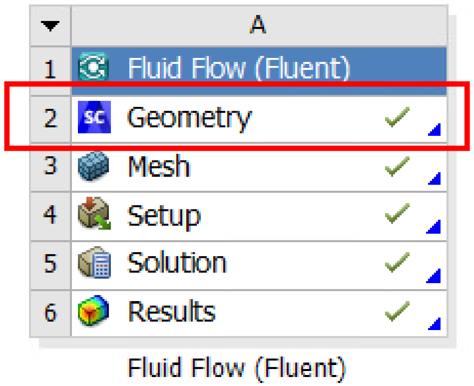
3 ANSYS FLUENT

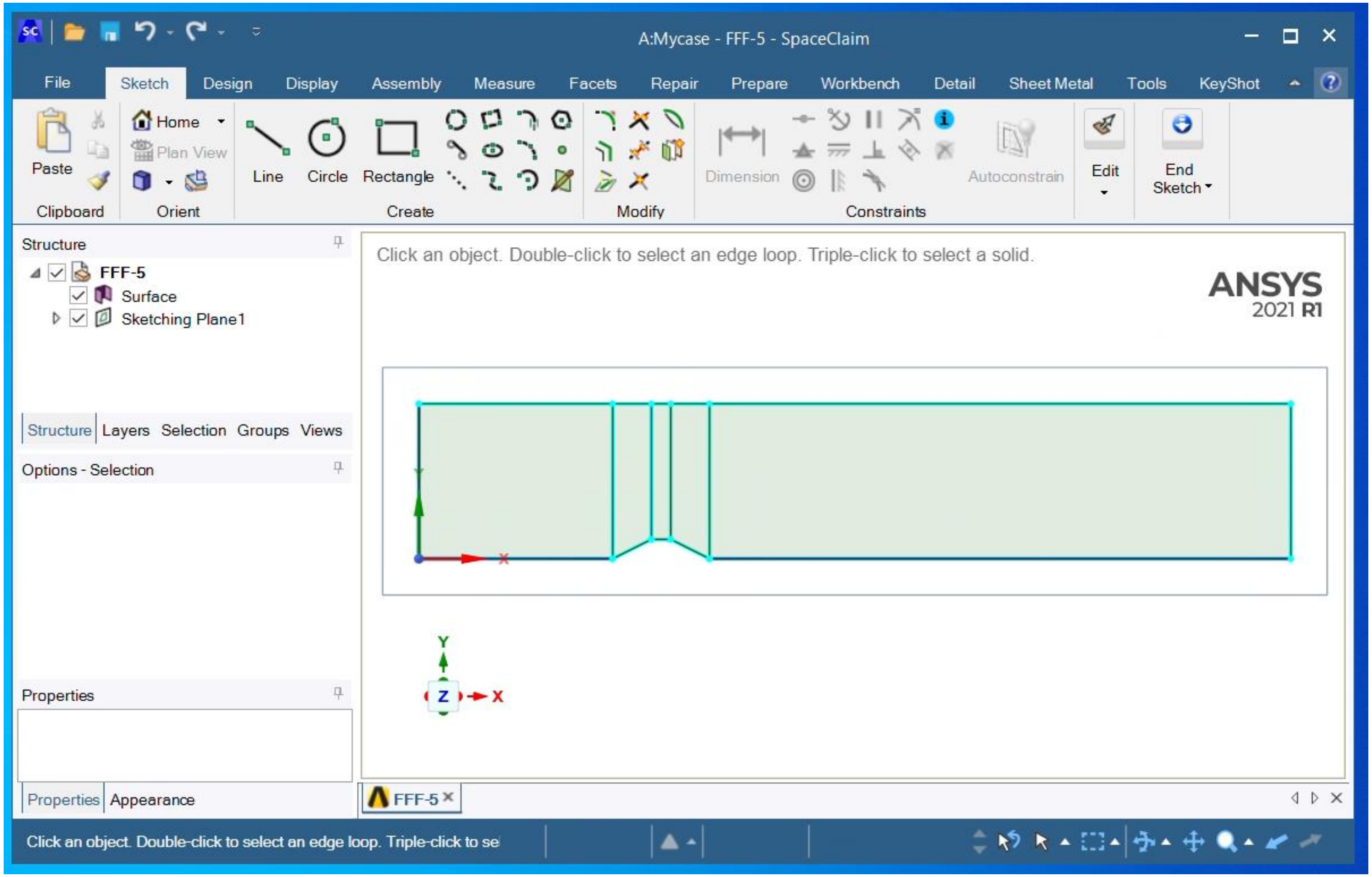
4 Ansys CFD Post

Introduction to the project: The 5 steps of a CFD project



Introduction to the project: The 5 steps of a CFD project

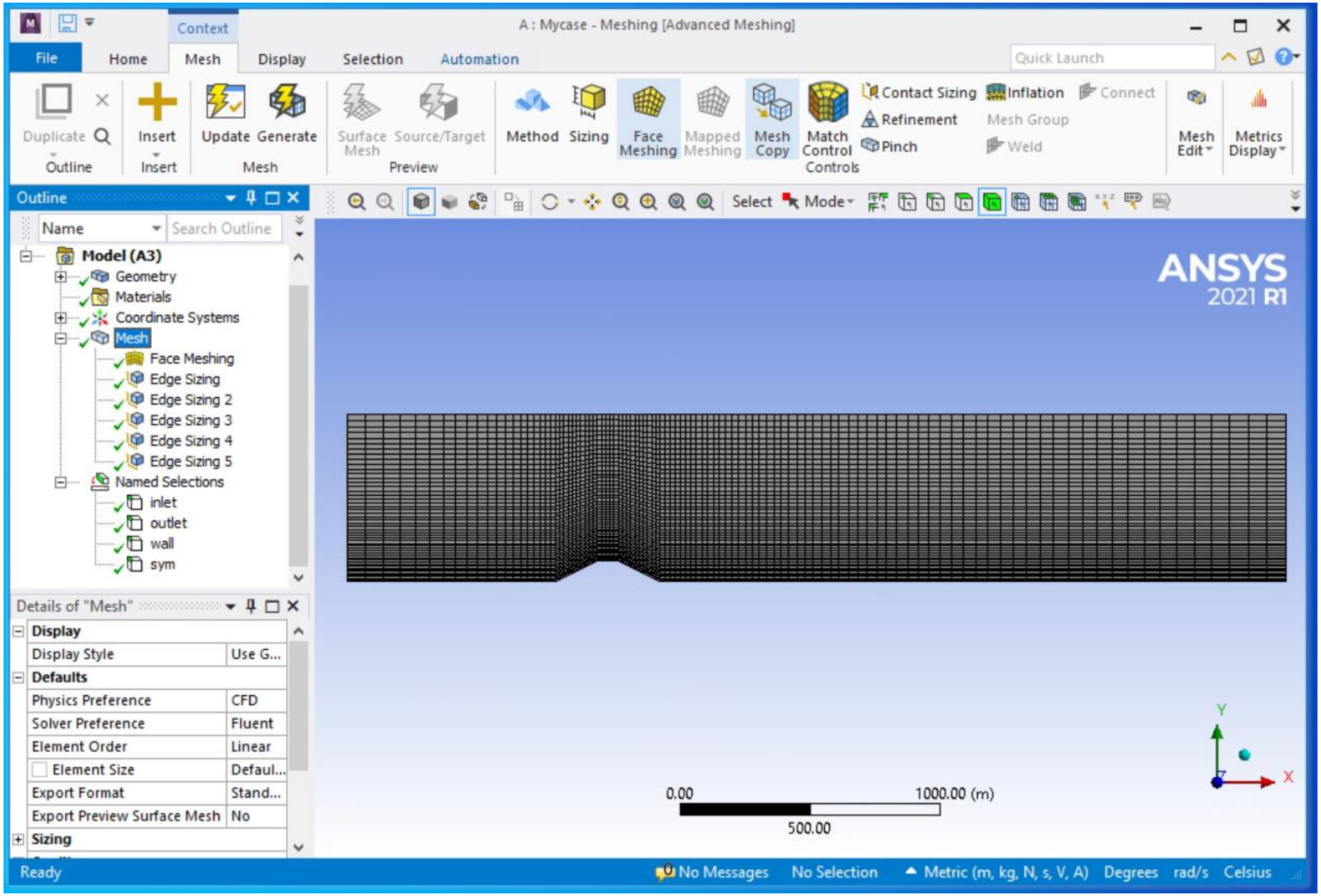




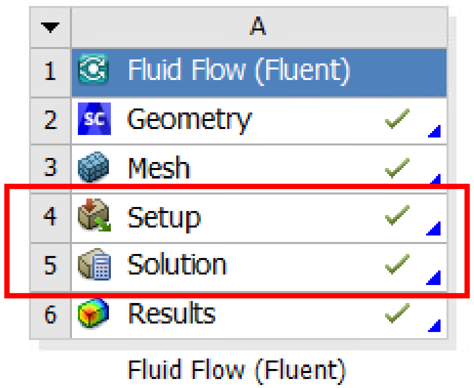
Introduction to the project: The 5 steps of a CFD project

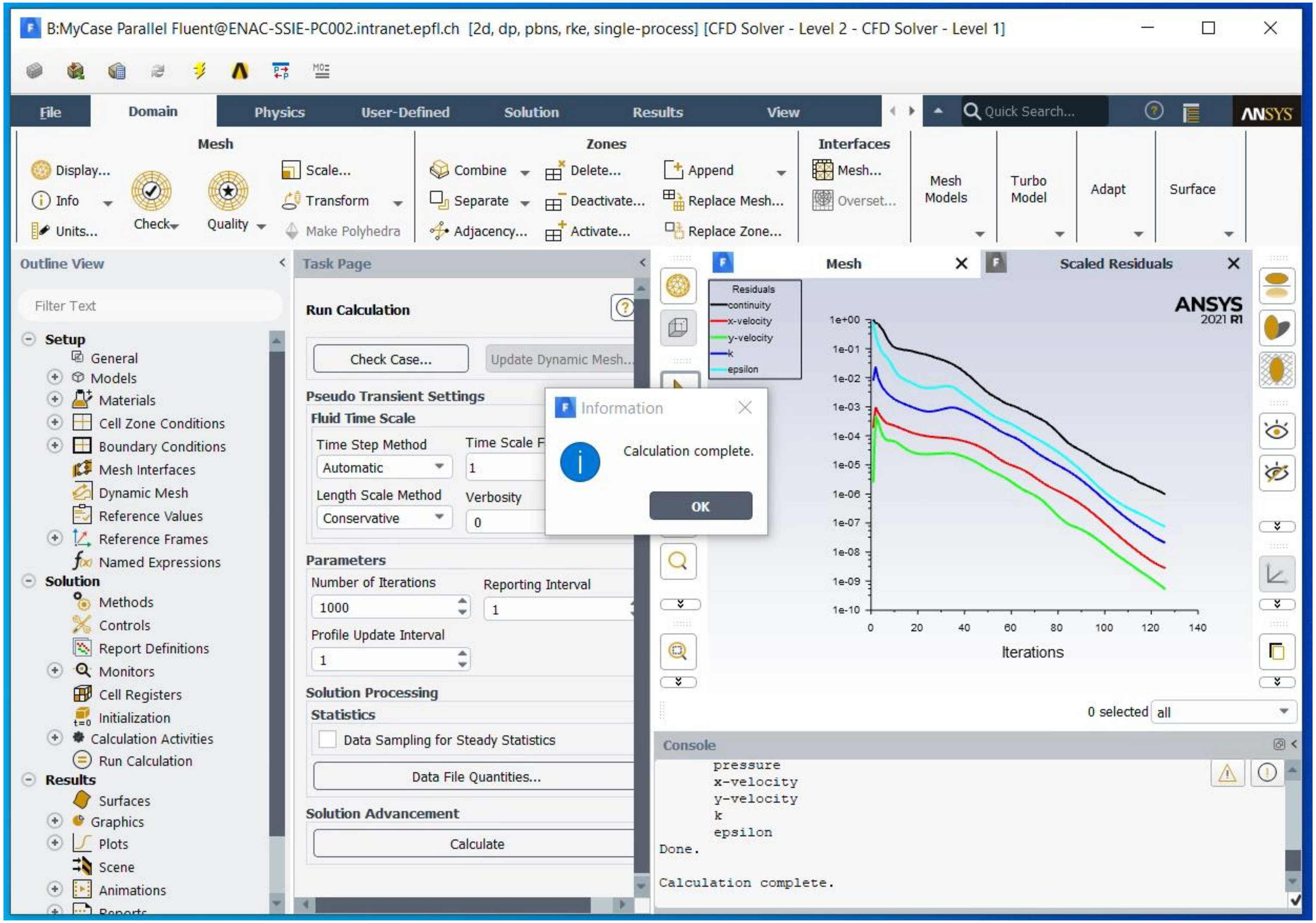
	A
1	Fluid Flow (Fluent)
2	Geometry ✓
3	Mesh ✓
4	Setup ✓
5	Solution ✓
6	Results ✓

Fluid Flow (Fluent)

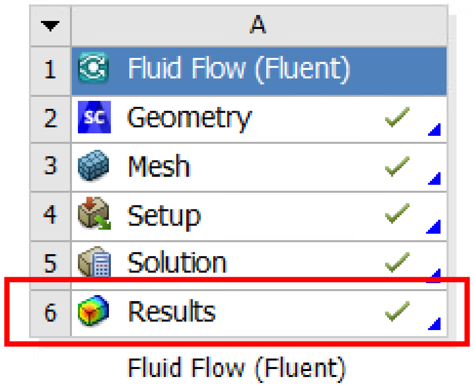


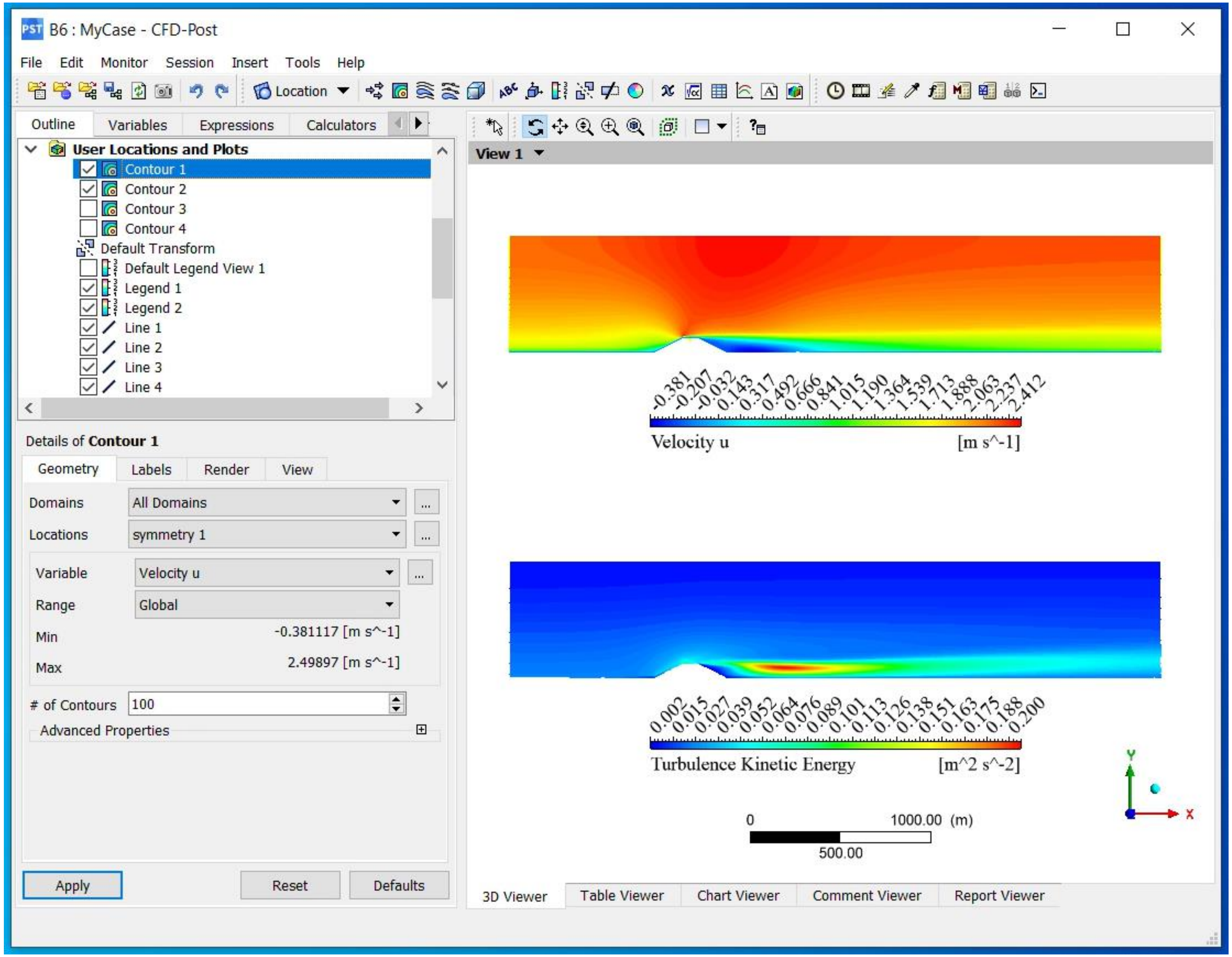
Introduction to the project: The 5 steps of a CFD project





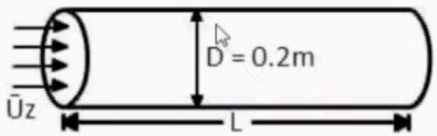
Introduction to the project: The 5 steps of a CFD project





*“Grid generation and **grid quality** are essential elements of the whole discretization process. […] The **accuracy** of the obtained numerical results is **critically dependent on mesh quality**.”* - Hirsch (2007)

Laminar Pipeflow



- Reynolds number of 100
  - $U = 1\ m/s$
  - $\rho = 1\ kg/m^3$
  - $\mu = 2 \times 10^{-3}\ kg/(m\ s)$.
- Geometry
  - $D = 0.2\ m$
  - $L = 8.0\ m$
- Produce
  - Velocity vectors
  - Velocity magnitude contours
  - Velocity profile at the outlet

Introduction to the tutorial

Submit the workbench case file and folder, and the screen capture (presented in the pdf) within a .zip. The submitted .zip must be named as follow:

<StudentLastName>_<StudentFirstName>_Tutorial.zip