



École Polytechnique Fédérale de Lausanne
Wind Engineering and Renewable Energy Laboratory

Environmental Transport Phenomena

Introduction to the Project

Teachers

Prof. Fernando Porté Agel
Dr. Benoît Couzry

Teaching Assistants

Mohammad Jamshidmofid
Marwa Souaiby
Matteo Cirillo
Amr Khedr
Xiao Hu

Fall 2025



Contents

Introduction and Project Structure

Part I: Tutorial

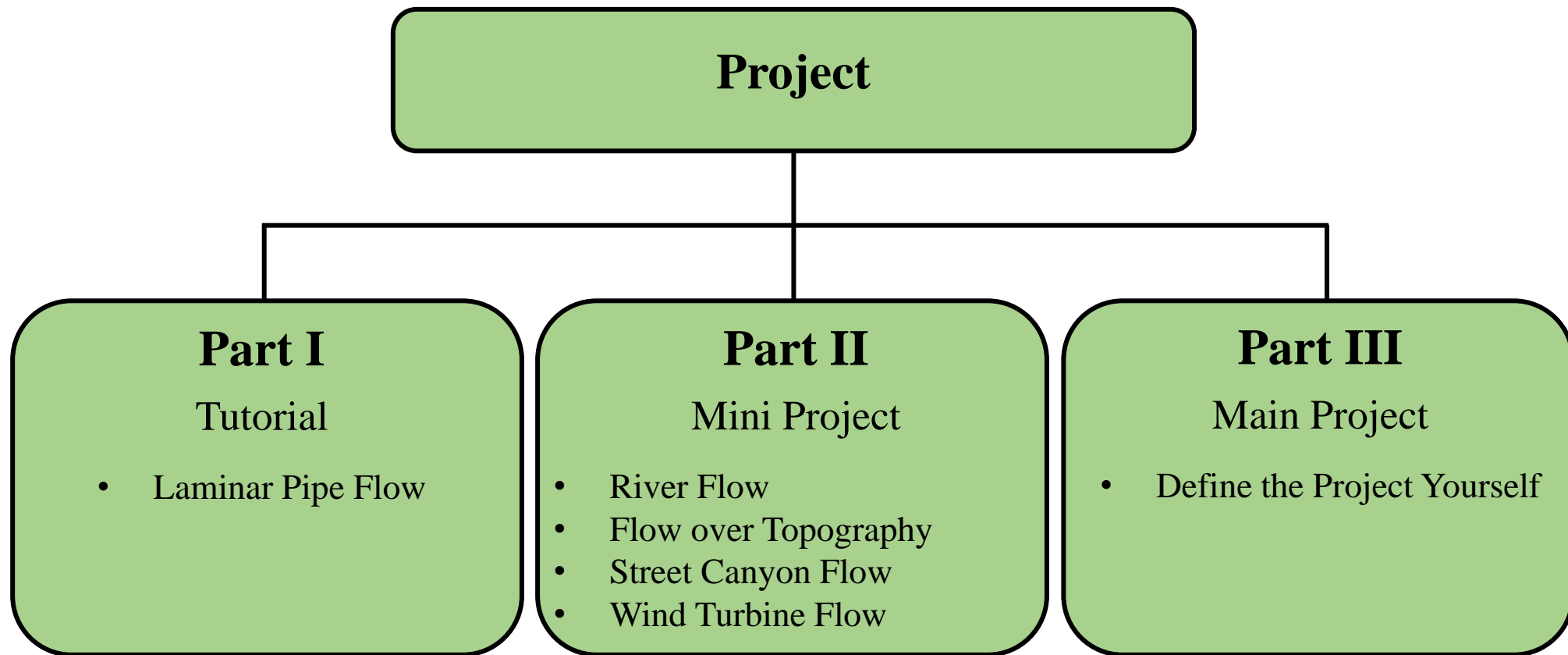
Part II: Mini Project

Part III: Main Project

Final Notes

Introduction and Project Structure





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

❖ **Step-by-step tutorial 10%**

❖ **Mini Project 20%**

❖ **Main project 70%**

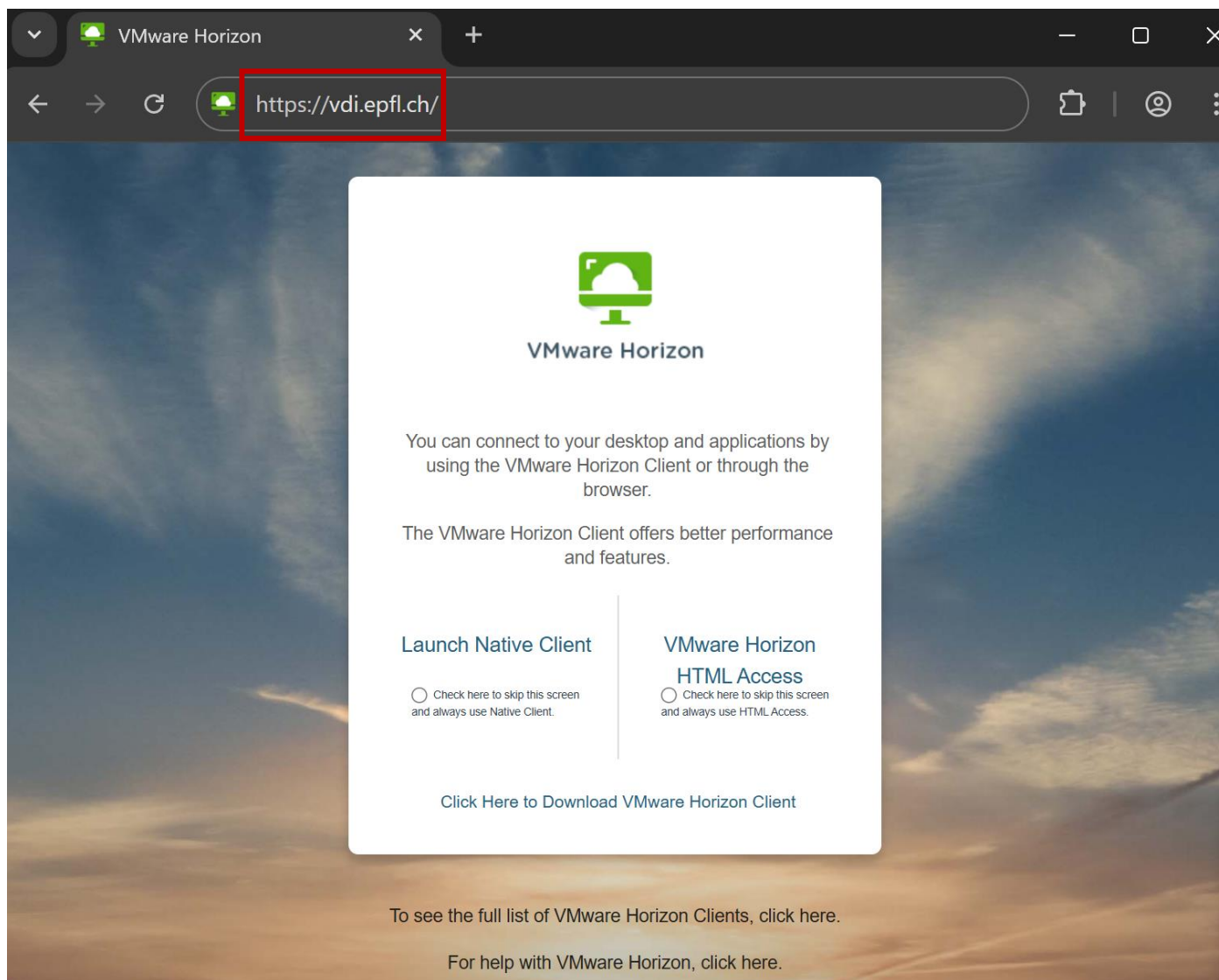
- **10%:** A clearly stated research question for the main project
- **30%:** Comprehensive description of your model, featuring: logical geometry to represent the problem, appropriate mesh to discretize the geometry, numerical methods well-suited to the problem, key results appropriately post-processed
- **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



We will use the flow-modelling software **Ansys 2021 R1** for all parts of the project




No need to install Ansys on your laptop! The **VDI** has everything you need. You can access it via vdi.epfl.ch
(VMware Horizon Client/HTML, ENAC-SSIE-WIN)



VMware Horizon

<https://vdi.epfl.ch/>


VMware Horizon

You can connect to your desktop and applications by using the VMware Horizon Client or through the browser.

The VMware Horizon Client offers better performance and features.

Launch Native Client

Check here to skip this screen and always use Native Client.

VMware Horizon HTML Access

Check here to skip this screen and always use HTML Access.

[Click Here to Download VMware Horizon Client](#)

To see the full list of VMware Horizon Clients, [click here](#).

For help with VMware Horizon, [click here](#).

The screenshot displays the VMware Horizon web client interface. The browser address bar shows the URL `vdi.epfl.ch/portal/webclient/#/launchitems`. The interface features a search bar and several utility icons. A grid of desktop icons is shown, each with a star icon in the top right corner and a three-dot menu icon in the bottom right corner. The icon for 'ENAC-SSIE-WIN' is highlighted with a red border.

Icon Label	OS/Environment
ENAC-SSIE-Ubuntu-20-04	Ubuntu-20-04
ENAC-SSIE-WIN	WIN
IC-CO-IN-SC-INJ-2025-Fall	INJ-2025-Fall
IC-CO-IN-SC-MA-2025-Fall	MA-2025-Fall
IC-Win11-MA	Win11-MA
IC-Windows-Cours2	Windows-Cours2
SB-IPHYS-LINUX	IPHYS-LINUX
SB-LINUX	LINUX
SB-WINDOWS	WINDOWS
STI-EDA-LABS-L4	EDA-LABS-L4
STI-EDA-LABS-L4-BKP	EDA-LABS-L4-BKP
STI-Windows11	Windows11

Information

**Informations importantes !
Important Information !**

Les machines VDI sont RÉINITIALISÉES à chaque démarrage, il est donc impératif de **sauvegarder vos fichiers uniquement sur votre espace personnel** (Lecteur Z:) et de ne rien stocker sur le Bureau !
Tout fichier stocké en local sera définitivement perdu !

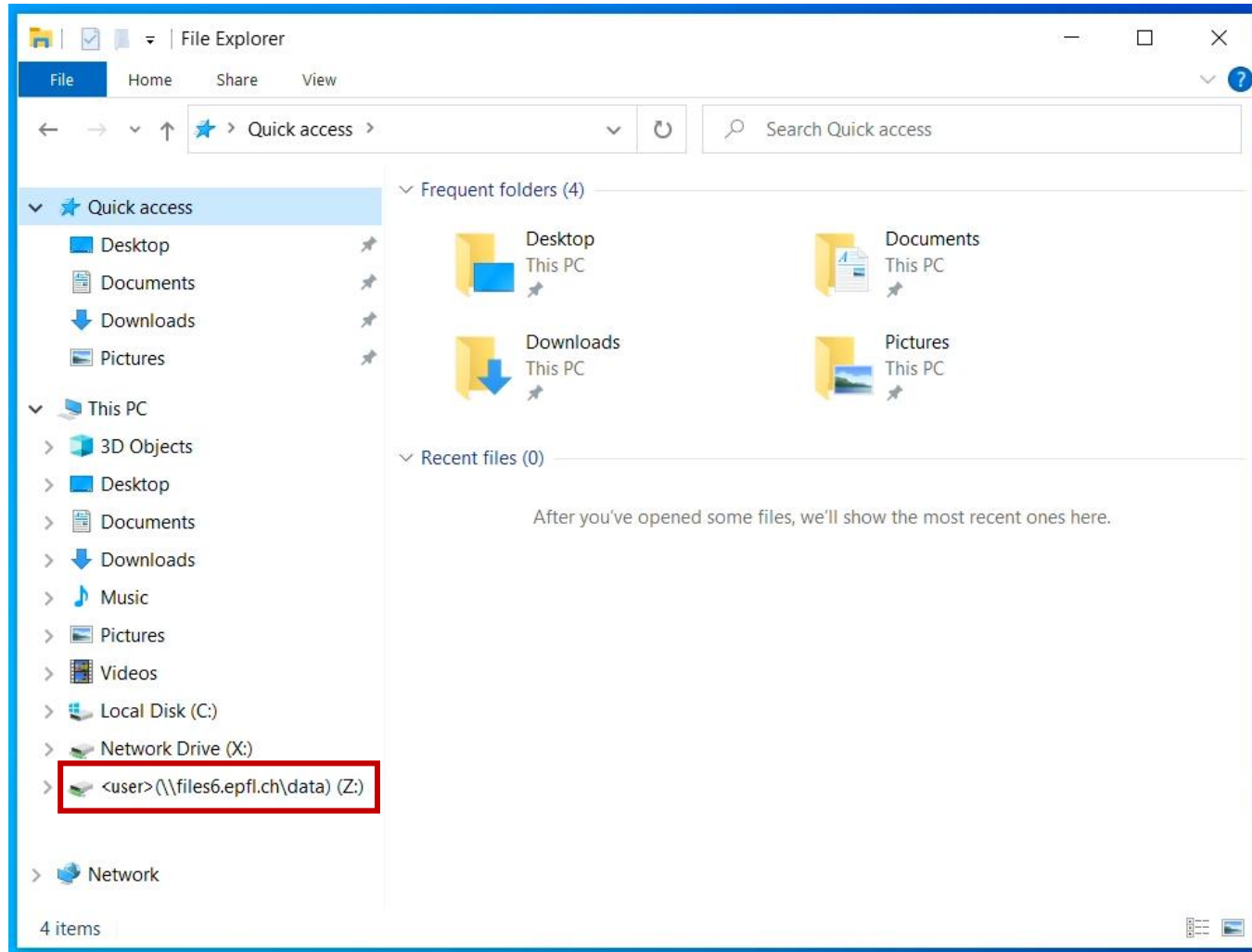
En cas de montage manuel du Lecteur Z : voir [<https://mynas.epfl.ch/>](https://mynas.epfl.ch/)

As VDI machines are RESET on each start-up, it is imperative to **save your files only on your personal space** (Z: Drive) and not to store anything on the Desktop !
Any file stored locally will be definitively lost !

In case of Z : Drive manual mount, see [<https://mynas.epfl.ch/>](https://mynas.epfl.ch/)

Pour toute question / remarque / problème sur ce Pool VDI - contactez 1234@epfl.ch <mailto:1234@epfl.ch>
For any question / remark / problem on this VDI Pool - contact 1234@epfl.ch <mailto:1234@epfl.ch>

Ok



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. Set up the solver and run the simulation**
- 4. Extract the results**

0. Analyze the flow problem

1. Design the geometry and the flow domain

2. Generate the mesh

3. Set up the solver and run the simulation

4. Extract the results



1



2

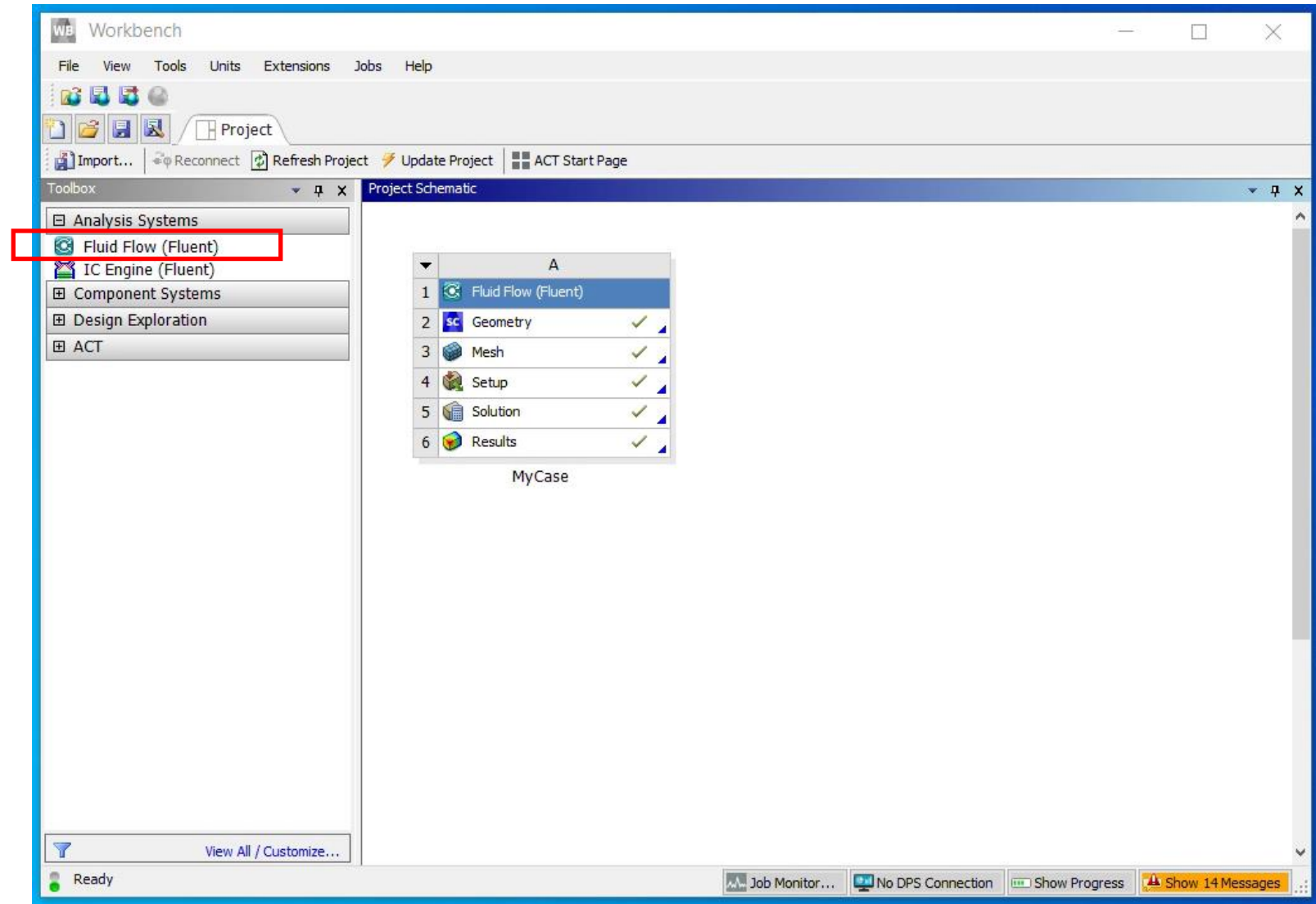


3



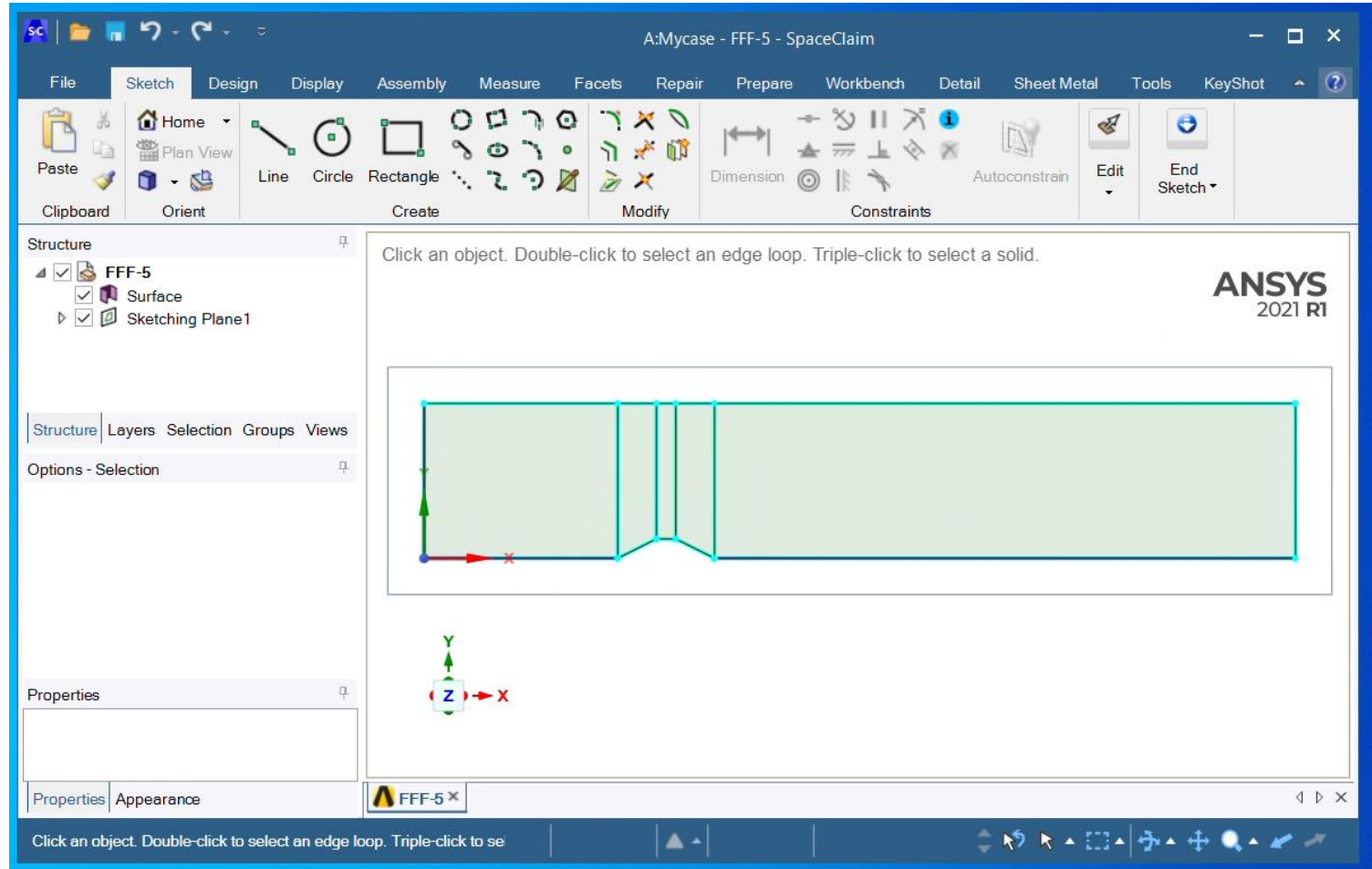
4





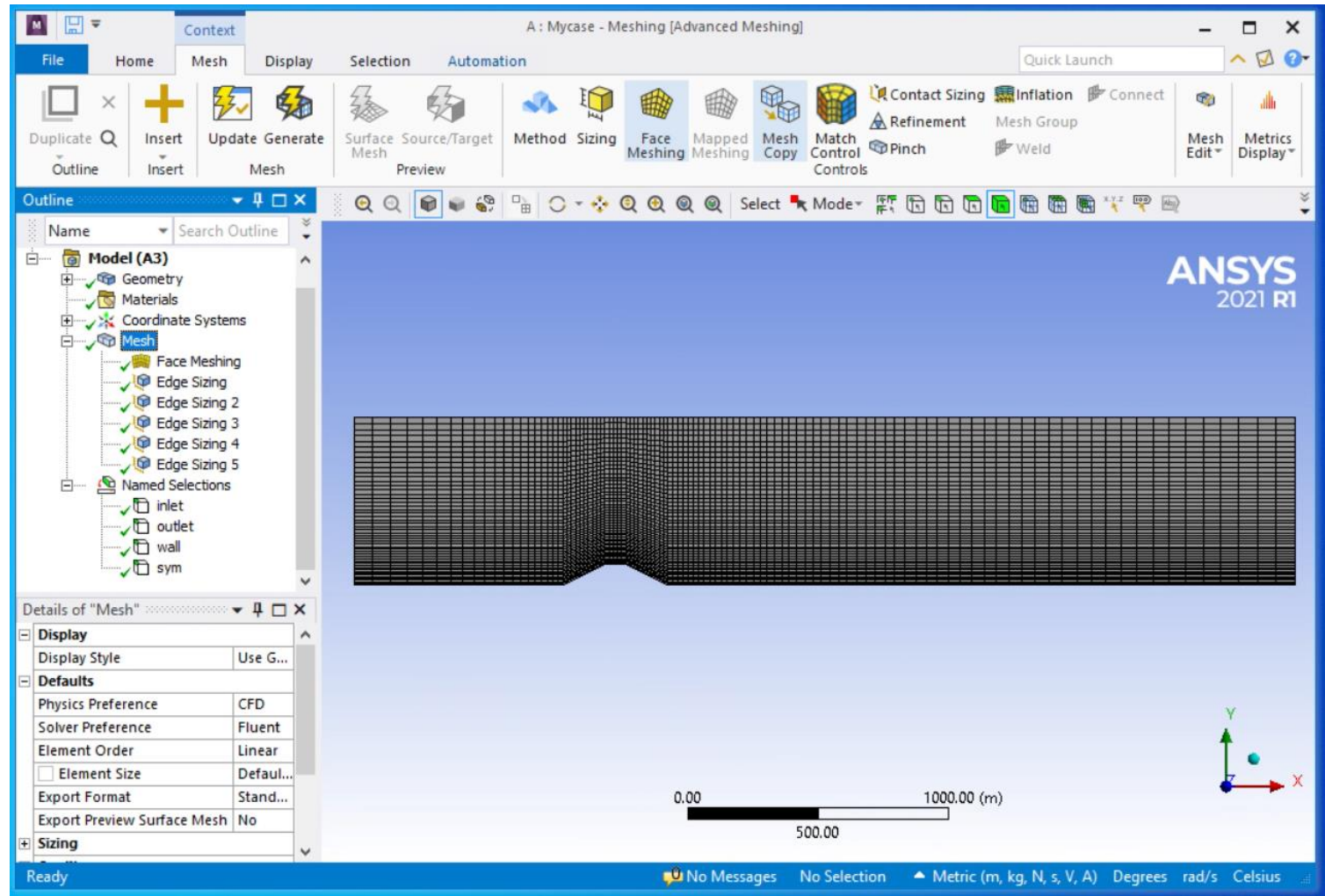
A	
1	Fluid Flow (Fluent) ✓
2	Geometry ✓
3	Mesh ✓
4	Setup ✓
5	Solution ✓
6	Results ✓

Fluid Flow (Fluent)



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

Fluid Flow (Fluent)



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

Fluid Flow (Fluent)

The screenshot displays the ANSYS Fluent 2021 R1 interface. The main window shows the 'Run Calculation' dialog box with the following settings:

- Time Step Method:** Automatic
- Time Scale Factor:** 1
- Length Scale Method:** Conservative
- Verbosity:** 0
- Number of Iterations:** 1000
- Reporting Interval:** 1
- Profile Update Interval:** 1

The 'Scaled Residuals' plot shows the convergence of residuals over 140 iterations. The y-axis is logarithmic, ranging from $1e-10$ to $1e+00$. The x-axis is 'Iterations' from 0 to 140. The residuals for continuity (black), x-velocity (red), y-velocity (green), k (blue), and epsilon (cyan) all decrease significantly over the iterations.

The 'Console' window at the bottom shows the following output:

```

pressure
x-velocity
y-velocity
k
epsilon
Done.
Calculation complete.
    
```

An 'Information' dialog box is also visible, stating 'Calculation complete.' with an 'OK' button.

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

PSI B6: MyCase - CFD-Post

File Edit Monitor Session Insert Tools Help

Location

Outline Variables Expressions Calculators

- User Locations and Plots
 - Contour 1
 - Contour 2
 - Contour 3
 - Contour 4
 - Default Transform
 - Default Legend View 1
 - Legend 1
 - Legend 2
 - Line 1
 - Line 2
 - Line 3
 - Line 4

Details of Contour 1

Geometry	Labels	Render	View
Domains	All Domains		
Locations	symmetry 1		
Variable	Velocity u		
Range	Global		
Min	-0.381117 [m s ⁻¹]		
Max	2.49897 [m s ⁻¹]		
# of Contours	100		
Advanced Properties			

Apply Reset Defaults

View 1

Velocity u [m s⁻¹]

Turbulence Kinetic Energy [m² s⁻²]

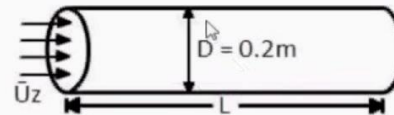
0 1000.00 (m)
500.00

3D Viewer Table Viewer Chart Viewer Comment Viewer Report Viewer

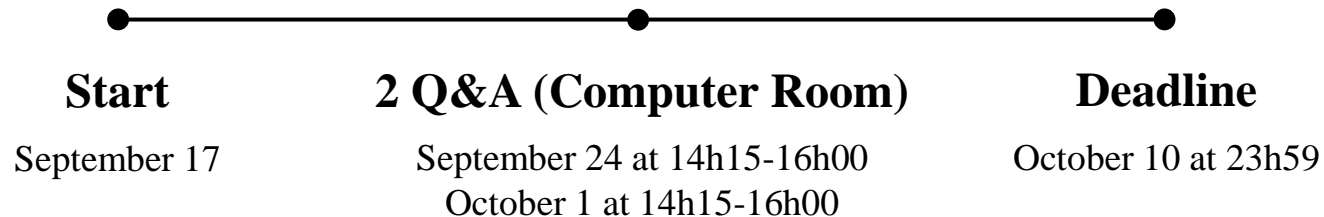
*“Grid generation and **grid quality** are essential elements of the whole discretization process. [...] The **accuracy** of the obtained numerical results is **critically dependent on mesh quality**.” - Hirsch (2007)*

Part I: Tutorial

Laminar Pipeflow



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



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:

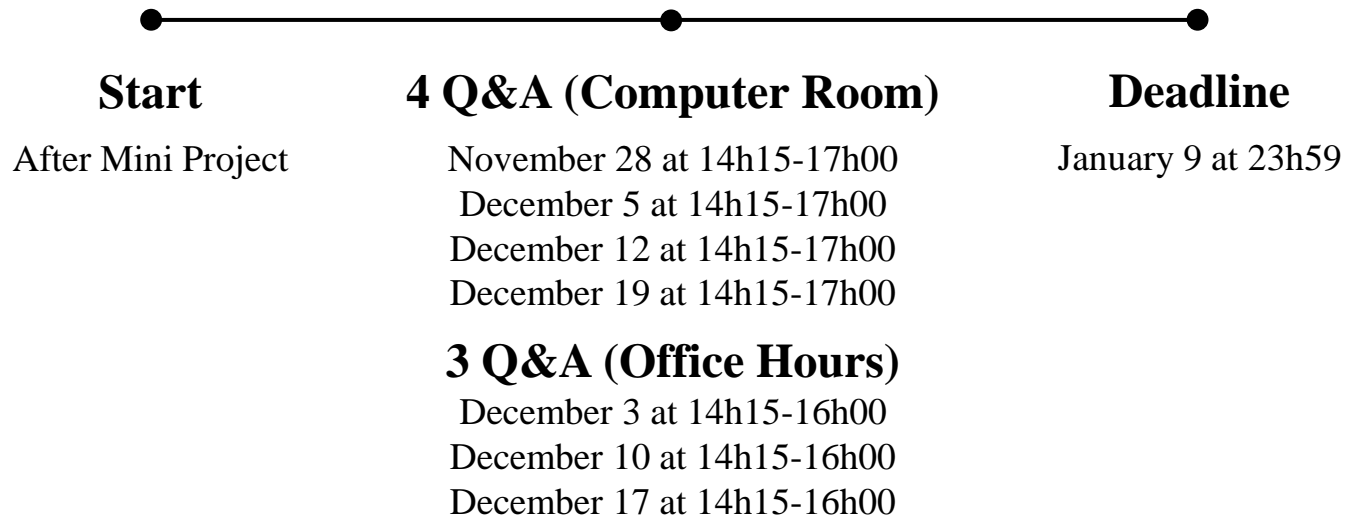
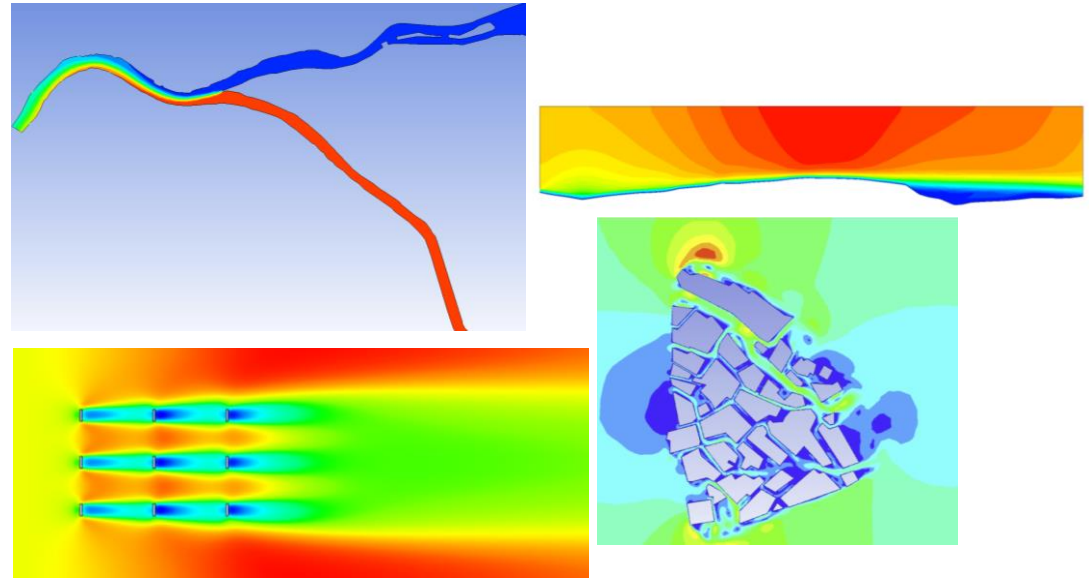
<StudentLastName>_<StudentFirstName>_Tutorial.zip



Part II: Mini Project

Part III: Main Project

Define your project in coordination with your TA and explore new aspects of CFD. The deadline for defining the project is **November 14 at 23:59.**



Final Notes

Environmental Transport Phenomena			
Fall 2025 Schedule			
	Project Session Wednesday 14h15-16h00	Course Session Friday 14h15-16h00	Exercise Session Friday 16h15-17h00
Week 1 8 September - 14 September	—	Introduction / Dimensional Analysis Benoît Crouzy	Introduction to the Project
Week 2 15 September - 21 September	—	Transport Processes Benoît Crouzy	Exercise Set 1
Week 3 22 September - 28 September	Tutorial Q&A (Computer Room)	Diffusion Benoît Crouzy	Exercise Set 2
Week 4 29 September - 5 October	Tutorial Q&A (Computer Room)	Boundary Conditions / Advection–Diffusion Benoît Crouzy	Exercise Set 3
Week 5 6 October - 12 October	—	Advection–Diffusion–Reaction Benoît Crouzy	Exercise Set 4
Week 6 13 October - 19 October	—	Advanced Topics / Diffusion–Driven Patterns Benoît Crouzy	Exercise Set 5
Fall Break 20 October - 26 October			
Week 7 27 October - 2 November	Mini Project Q&A (Computer Room)	Turbulent Flow Fernando Porté-Agel	Exercise Set 6
Week 8 3 November - 9 November	Mini Project Q&A (Computer Room)	Computational Fluid Dynamics Fernando Porté-Agel	—
Week 9 10 November - 16 November	Mini Project Q&A (Office Hours)	Atmospheric Boundary Layer Fernando Porté-Agel	Exercise Set 7
Week 10 17 November - 23 November	—	Turbulent Dispersion in Rivers Fernando Porté-Agel	Exercise Set 8
Week 11 24 November - 30 November	—	Main Project Q&A (Computer Room)	
Week 12 1 December - 7 December	Main Project Q&A (Office Hours)	Main Project Q&A (Computer Room)	
Week 13 8 December - 14 December	Main Project Q&A (Office Hours)	Main Project Q&A (Computer Room)	
Week 14 15 December - 21 December	Main Project Q&A (Office Hours)	Main Project Q&A (Computer Room)	

Remember the Deadlines!

October 10 at 23h59	November 14 at 23h59	January 9 at 23h59
Tutorial	Mini project	Main project
Select the topic of your mini project and make your group	Define the main project	

1. Please note that **during the office hours Q&A, no project session will take place in the computer room.** Instead, students may contact their assigned TA for project-related questions, either by coming directly to the TA's office or by sending an email.

TAs Contact Information:

- Mohammad Jamshidmofid – GR B2 401, mohammad.jamshidmofid@epfl.ch
 - Marwa Souaiby– GR B2 401, marwa.souaiby@epfl.ch
 - Amr Khedr– GR B2 402, amr.khedr@epfl.ch
 - Xiao Hu – GR B2 399, xiao.hu@epfl.ch
 - Matteo Cirillo – GR B2 399, m.cirillo@epfl.ch
2. Use **Ansys 2021 R1** for all parts of the project. Projects done with other versions cannot be evaluated.
 3. You should **submit the tutorial deliverables via Moodle.** For the **mini project and the main project, each group should submit the deliverables by email** to the assigned TA (see the emails above).

4. A questionnaire will be activated in the **General** section of Moodle, where you can indicate your preferred topic for the mini project and the people you would like to work with. Each group may have up to three members.
5. Please **verify your login credentials and your access to the VDI and the computer room**. If you encounter any issues with access, contact the TA (see the email below) as soon as possible.
6. Please **respect the deadlines**. Late submissions will not be accepted.
7. You can find this presentation in the **General** section of Moodle. Please review it again, and if you have any questions, feel free to contact the TA (see the email below).

Good luck and enjoy ETP!

mohammad.jamshidmofid@epfl.ch



Thank you!