

NCBJ/DUZ/UZ2 Seminar:

# Making CFD simulation

- Part 1/2 -

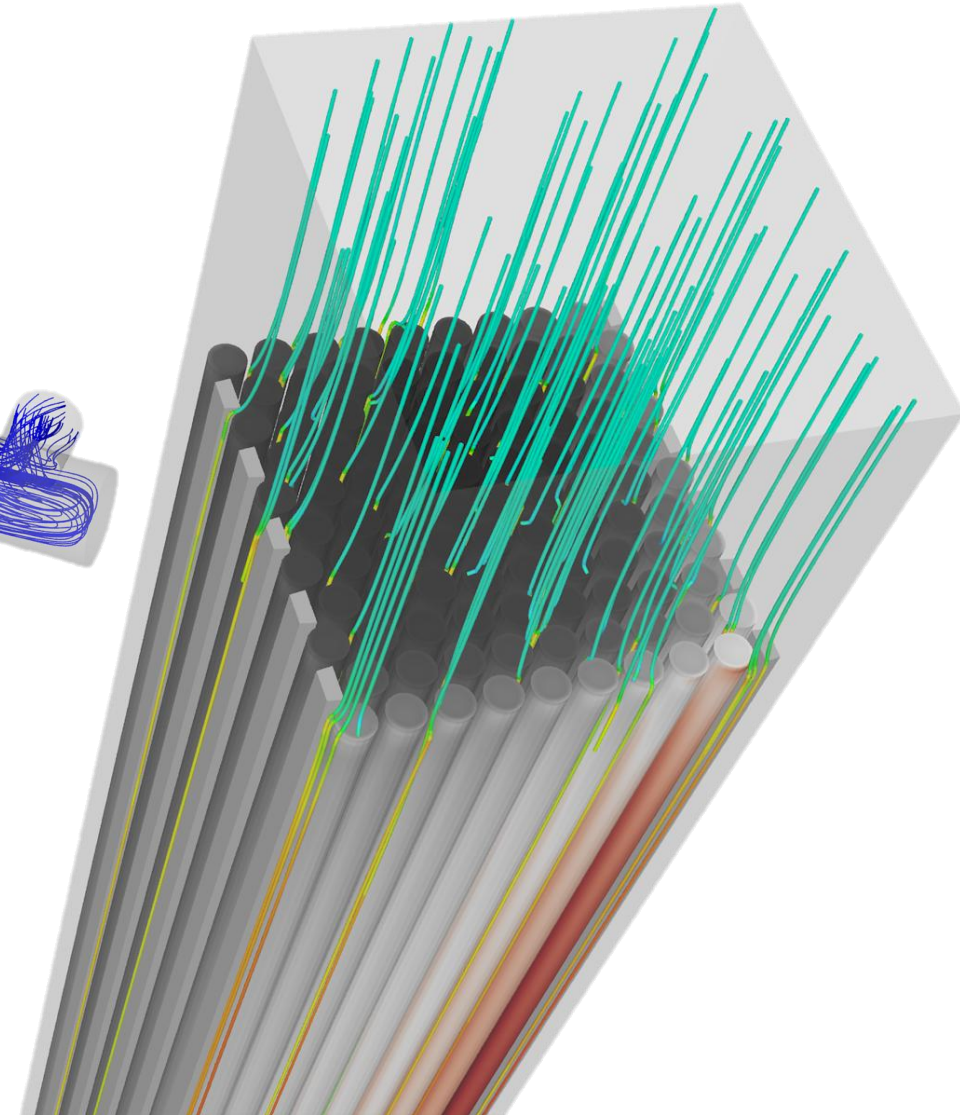
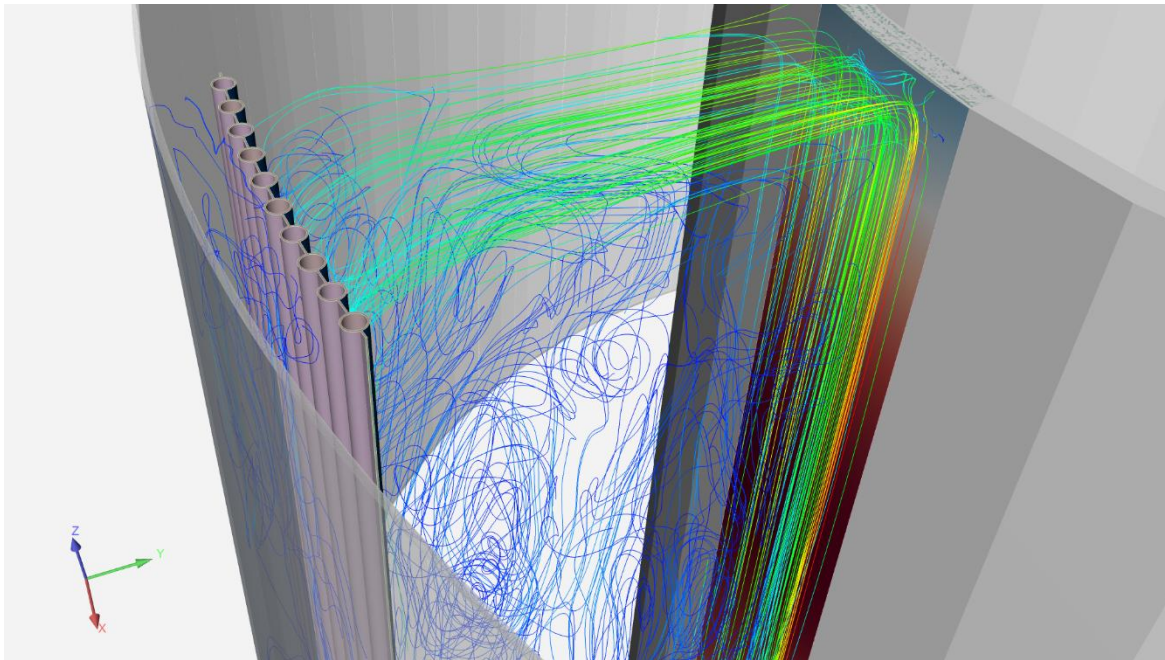
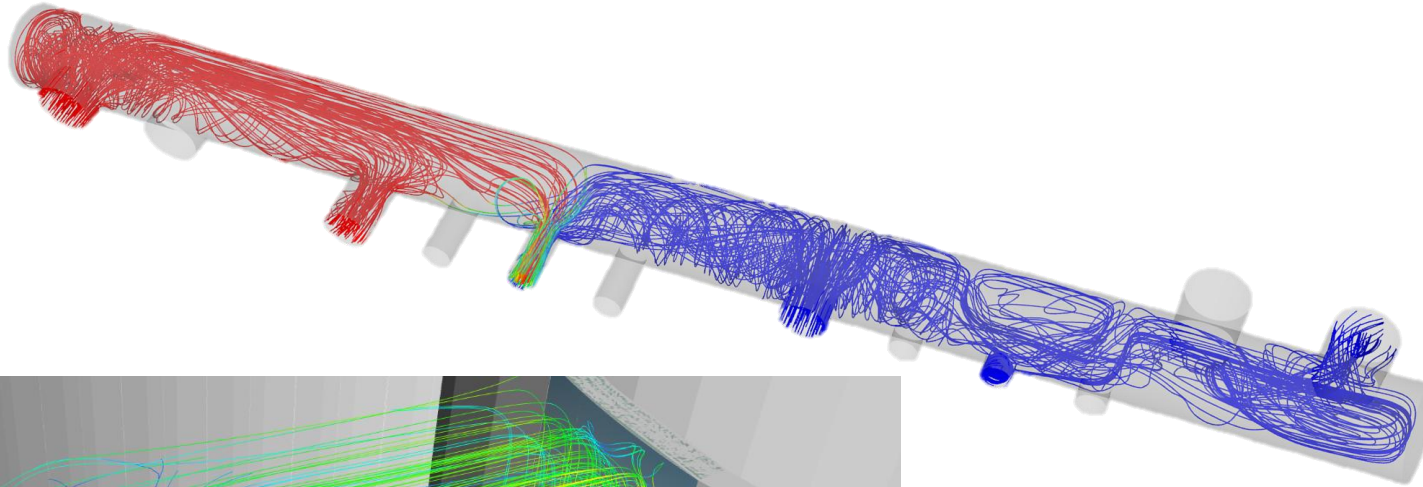
Piotr Prusiński  
Tomasz Früboes

30.03.2026

# Outline

- What CFD stands for?
- CFD types and differences
- CFD case step-by-step
- Sensitivity study

# CFD = Colours For Directors?



# Computational Fluid Dynamics – CFD

- Numerical technique for solving mass (and heat) transport in real geometries based on fundamental conservation laws (or even *ab initio*):

- Continuity (Mass):

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0$$

- Momentum:

$$\frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho u \times u) = -\nabla p + \nabla \cdot \tau + \rho g$$

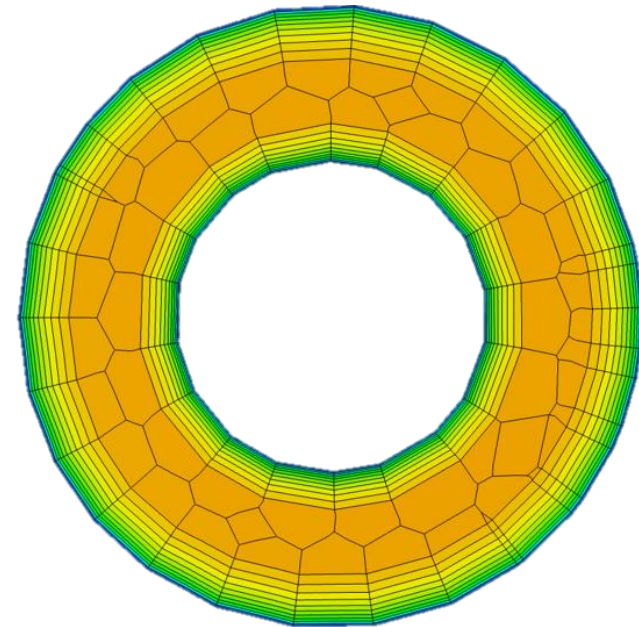
- Energy/Heat:

$$\frac{\partial(\rho E)}{\partial t} + \nabla \cdot (u(\rho E + p)) = \nabla \cdot (k\nabla T + u \cdot \tau) + S_h$$

- Think of CFD as of Digital Twin of a real experimental setup

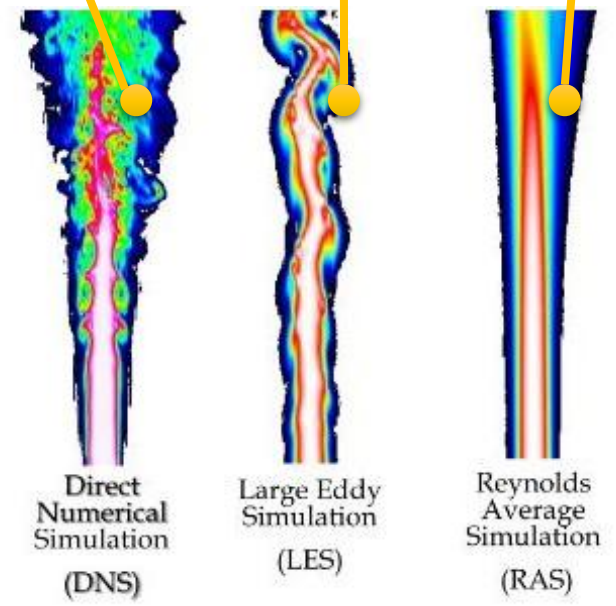
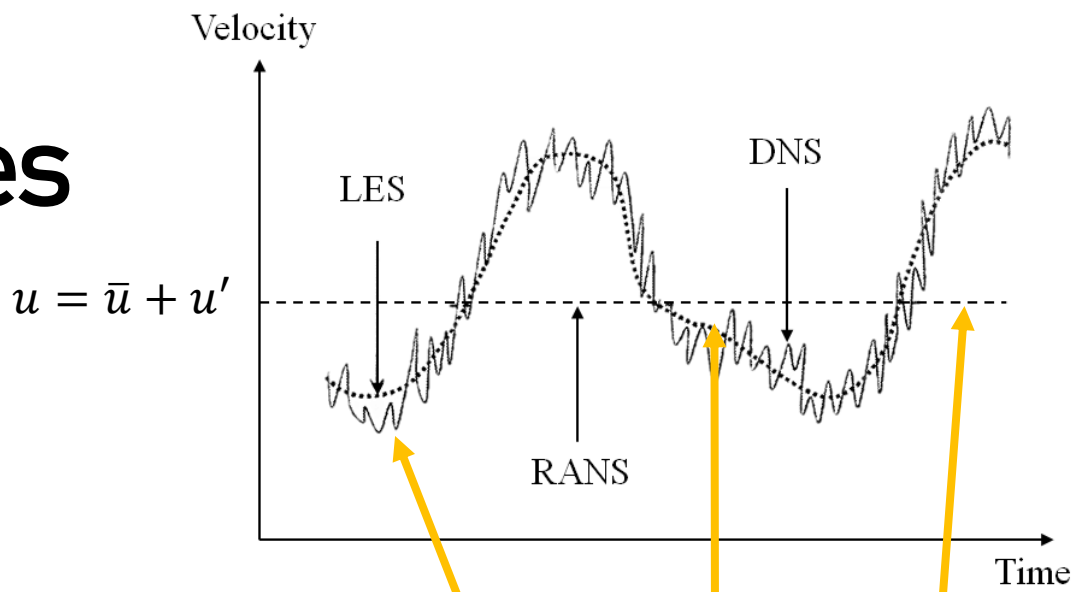
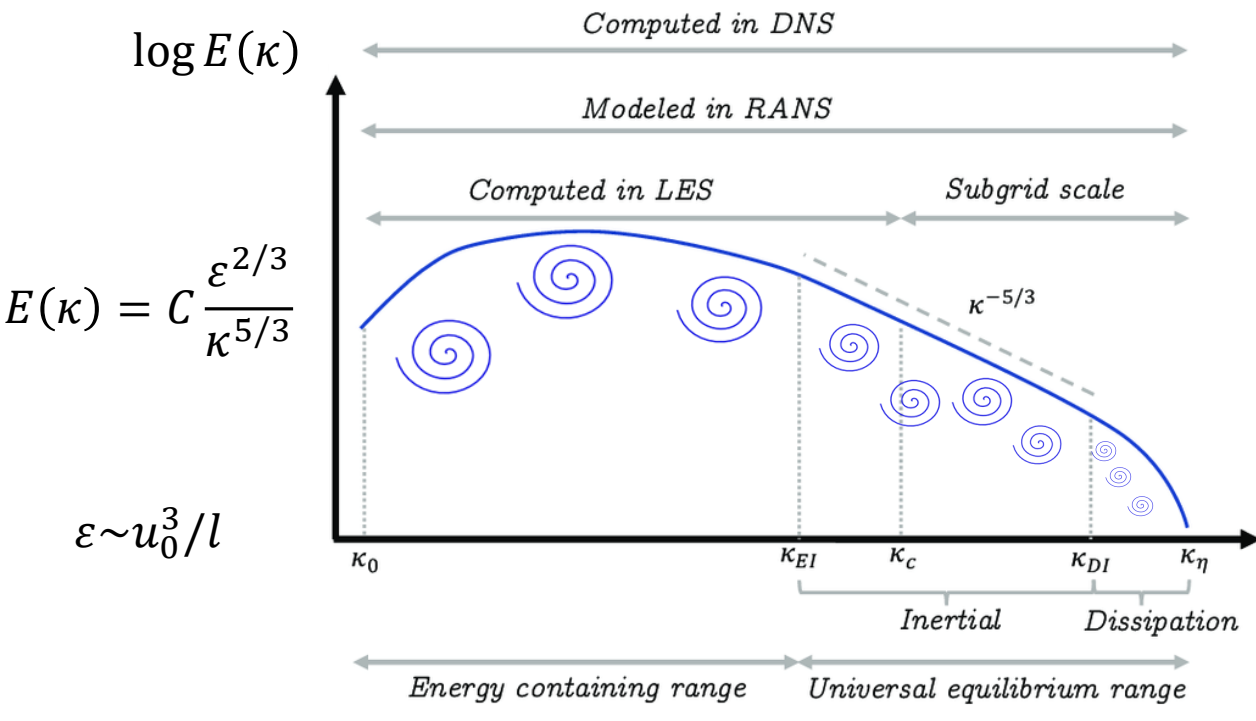
# Spatial discretization = meshing

- We DO NOT solve ONE SET of governing EQUATIONS for the whole geometry
- We DO solve NUMBERS of SETS of EQUATIONS at discrete locations in such geometry in an iterative process.

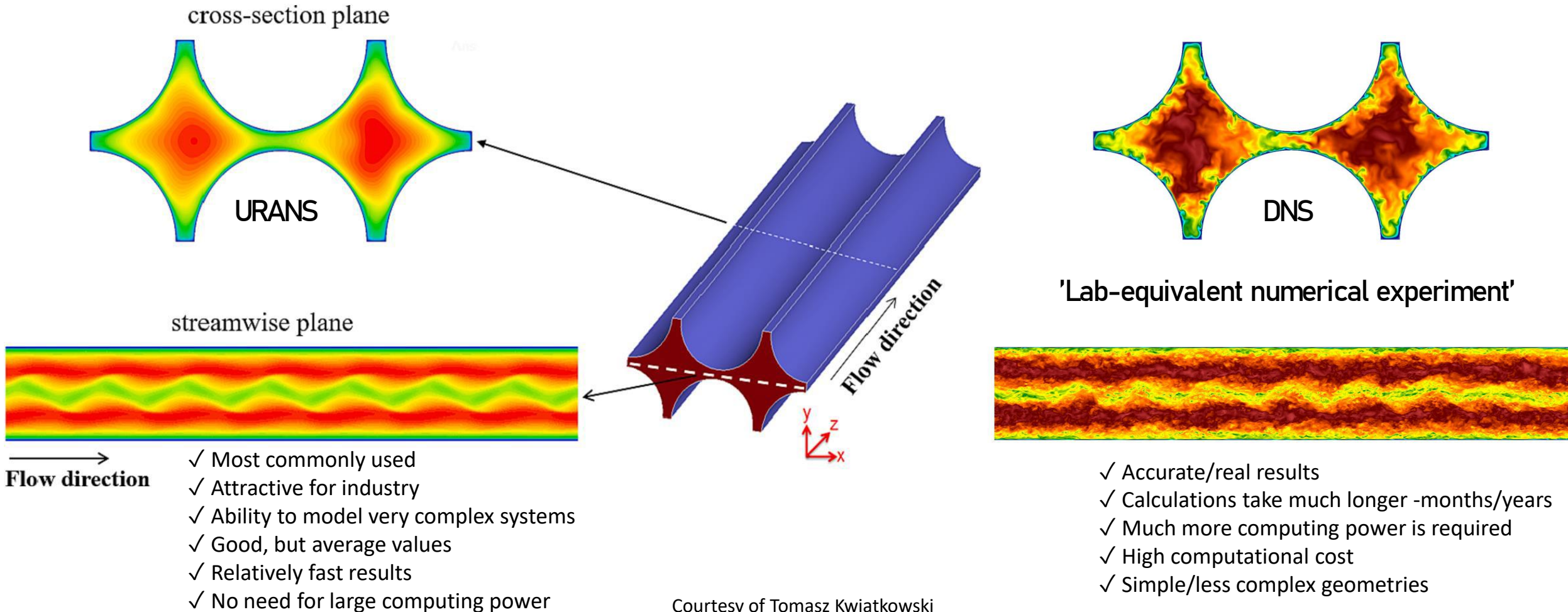


# RANS, LES, DNS = CFD types

- Most common division



# NCBJ's experience



# Continuity equation - explained

- Continuity (Mass):

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0$$

How much mass accumulated through the time?

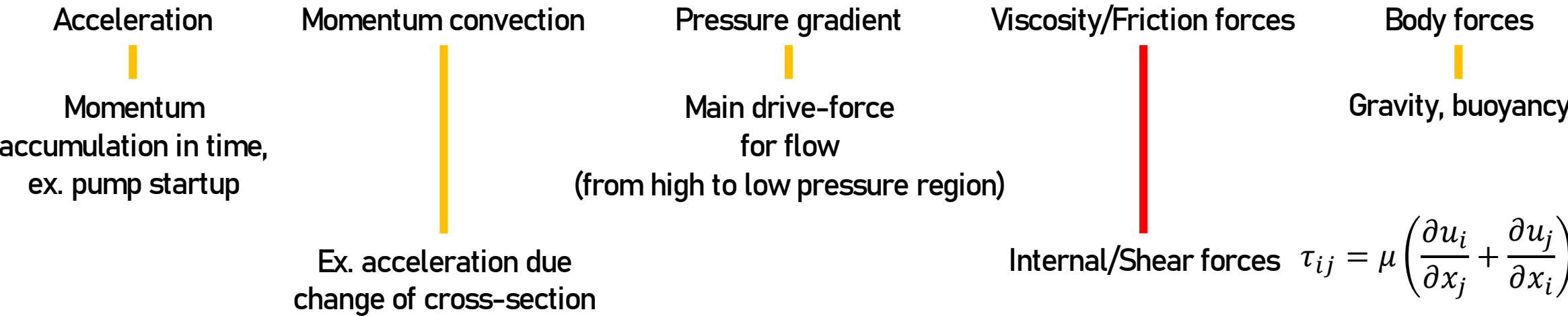
How much mass transported through a control volume borders?

Usually:  $\nabla = \left( \frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \right)$

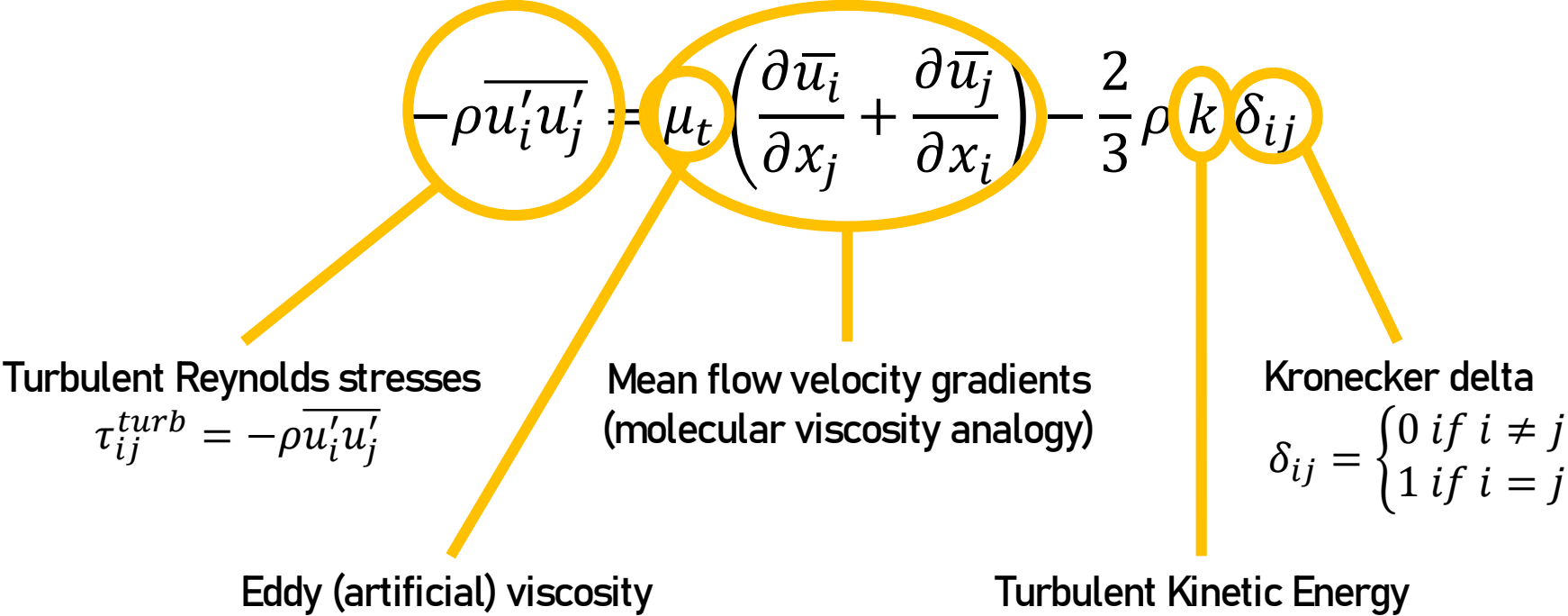
# Momentum equation - explained

• Momentum:

$$\frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho u \times u) = -\nabla p + \nabla \cdot \tau + \rho g$$



# Boussinesq hypothesis



# Effectively, when using RANS

- Instead of solving this one:

$$\tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$$

- We are modeling with this one:

$$\tau_{ij}^{total} = (\mu + \mu_t) \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij}$$

- But the problem is Turbulent Kinetic Energy:

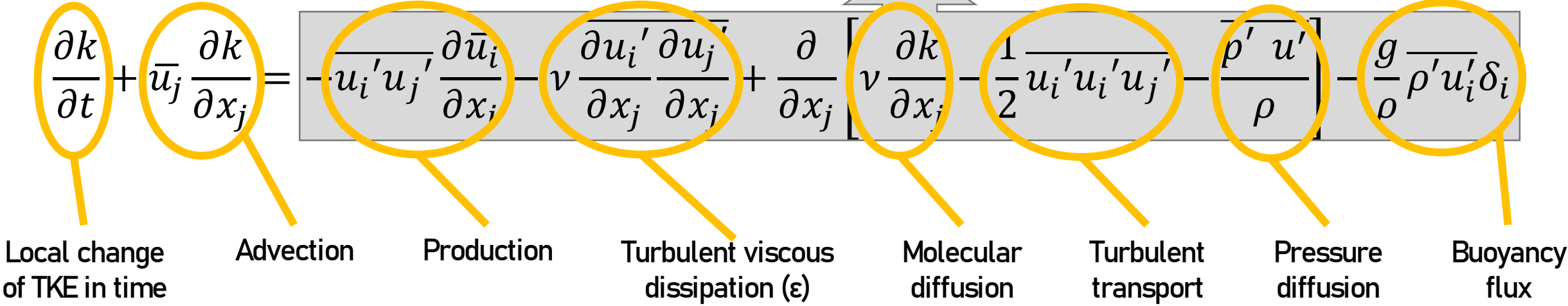
$$k = \frac{1}{2} \left( \overline{(u'_i)^2} + \overline{(u'_j)^2} + \overline{(u'_k)^2} \right)$$

# Essence of RANS turbulence modelling

- The general form of the resolved TKE equation is derived from sum of products of Navier-Stokes equations with fluctuating velocities:

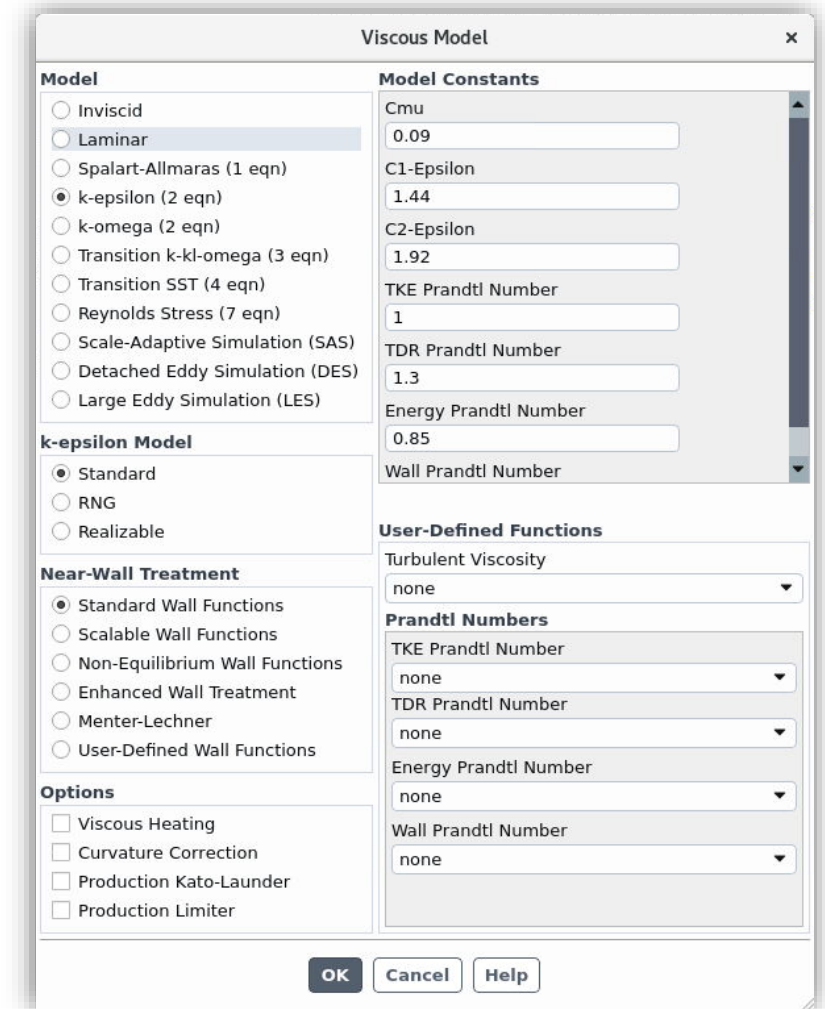
$$\nu = \frac{\mu}{\rho}$$

None of them known, each term has to be modelled in RANS



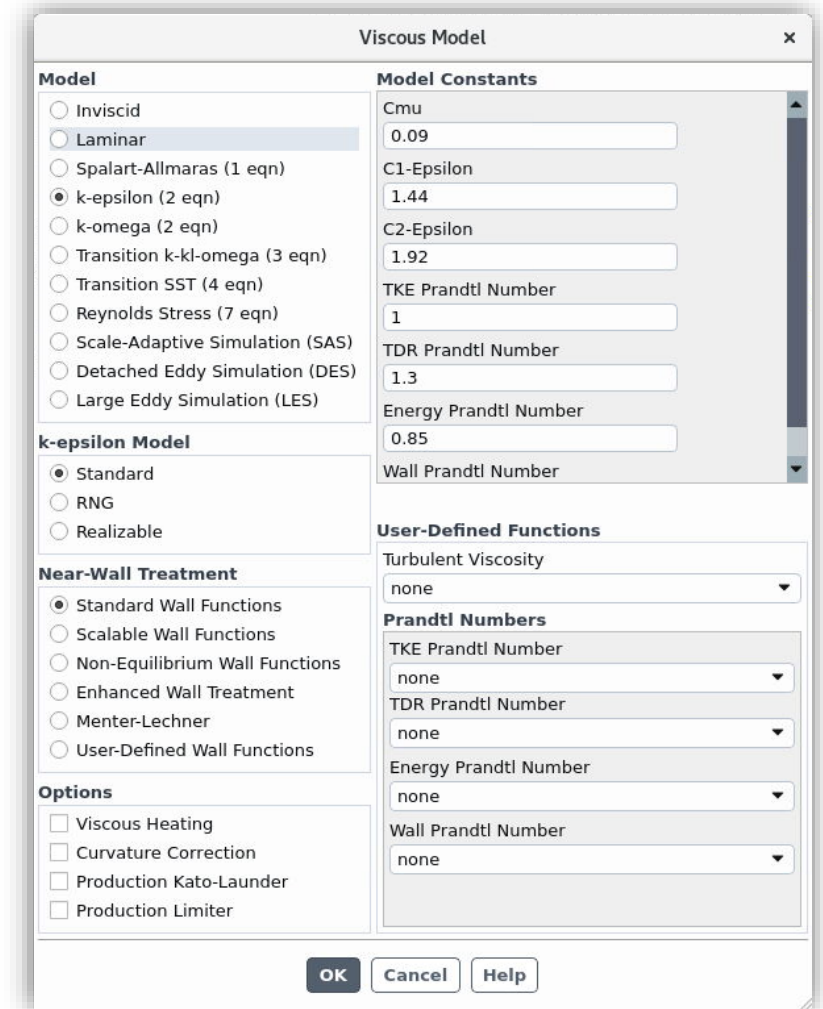
# Turbulence models

- The choice of turbulence model appropriate to the physics in question is always a matter of engineering judgement.
- It is usually based on answers to following:
  - What is the general focus of the model?  
What are the strongest and weakest points?
  - Does the model was calibrated against real life example similar to the one in question?
- The final choice is mostly motivated with engineer's past experiences.



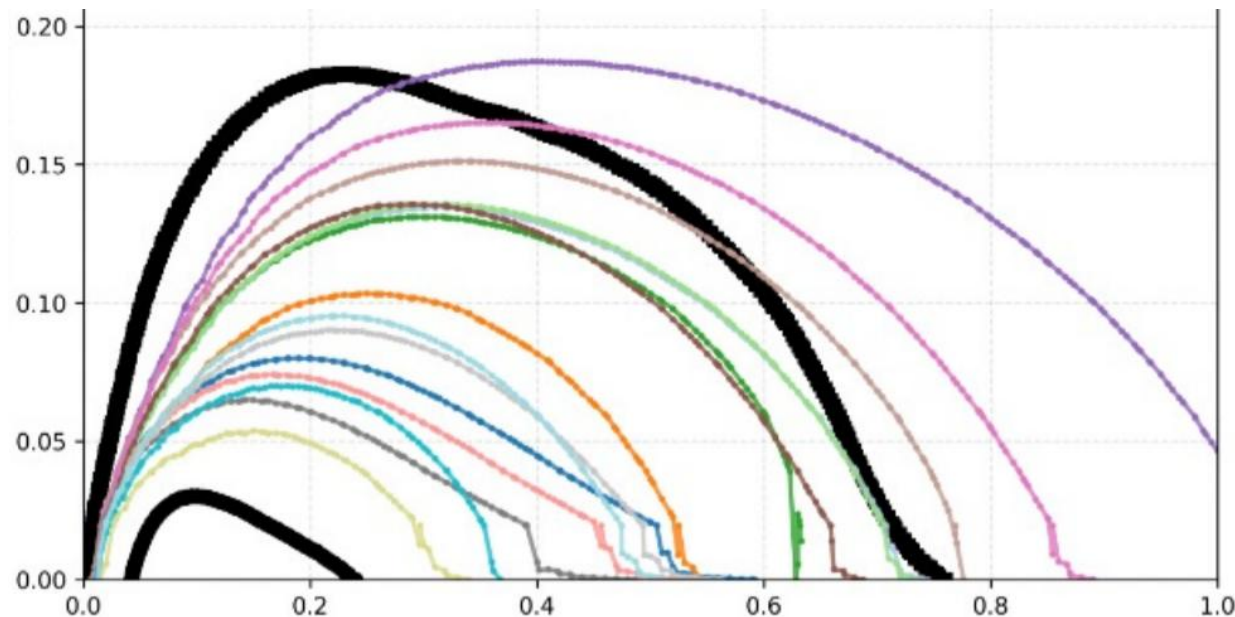
# Turbulence models in ANSYS Fluent

- ANSYS Fluent provides plenty of models, submodels and their derivations with some default options.
- User is allowed to combine them (almost) freely, including changes in some model constants or by adding user's own functions instead.
- Such a customization seems attractive, but adds several biases and assumptions, creating a deviation from the original calibrated model of general purpose. That requires V&V.



# Which one is correct?

- How do we know the simulation output is correct?
- Usually, we don't know\*, but we can estimate the impact of turbulence model choice on final results – we call this sensitivity study.



# ANSYS Fluent isn't an easy tool

- The choice of turbulence model is just one source of headache. There are few others related when using ANSYS Fluent particularly.
- Until 2-3 years back, Fluent offered only two ways of setting case configuration, i.e. via:
  - GUI - Graphical User Interface
    - Although, it seems most attractive approach, yields quite a lot of problem when the targeted simulation requires more hardware than we have locally or it takes an hour to load the data to RAM just to check/change settings.
  - TUI - Text User Interface (based on dedicated/limited syntax)
    - Preferred option when sensitivity study is a major topic – with some significant issues.

# ANSYS Fluent TUI - example

- Definition of Inlet boundary conditions (BCs):
  - Interactively we answer questions as below:

```
; /define/boundary-conditions> mass-flow-inlet
; (inlet)
; zone id/name [inlet] inlet
; Reference Frame: Absolute [yes] yes
; Mass Flow Specification Method: Mass Flow Rate [yes] yes
; Use Profile for Mass Flow Rate? [no] no
; Mass Flow Rate (in [kg/s]) [0] 0.6234
; Use Profile for Supersonic/Initial Gauge Pressure? [no] no
; Supersonic/Initial Gauge Pressure (in [Pa]) [0] 0
; Direction Specification Method: Direction Vector [yes] no
; Direction Specification Method: Normal to Boundary [no] yes
; Turbulent Specification Method: K and Epsilon [no] no
; Turbulent Specification Method: Intensity and Length Scale [no] no
; Turbulent Specification Method: Intensity and Viscosity Ratio [yes] no
; Turbulent Specification Method: Intensity and Hydraulic Diameter [no] yes
; Turbulent Intensity (in [%]) [5] 4.0377199
; Hydraulic Diameter (in [m]) [1] 0.02
```

- But we can write the same as a one-liner:

```
/define/boundary-conditions/mass-flow-inlet inlet y y n 0.6234 n 0 n y n n n y 4.0377199 0.02
```

- Now to show the essence of the problem in a soft way, let's imagine we turned on energy equation at some point (it's not 'on' by default) to account for temperature field, then we need to add an appropriate change in the inlet BC:

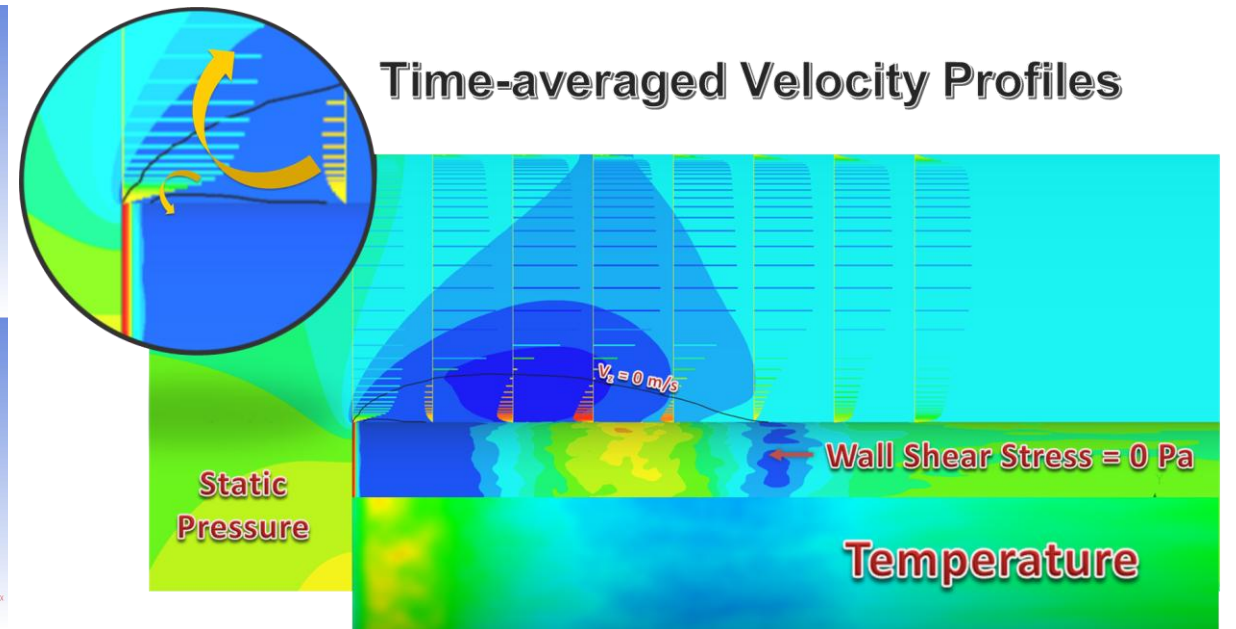
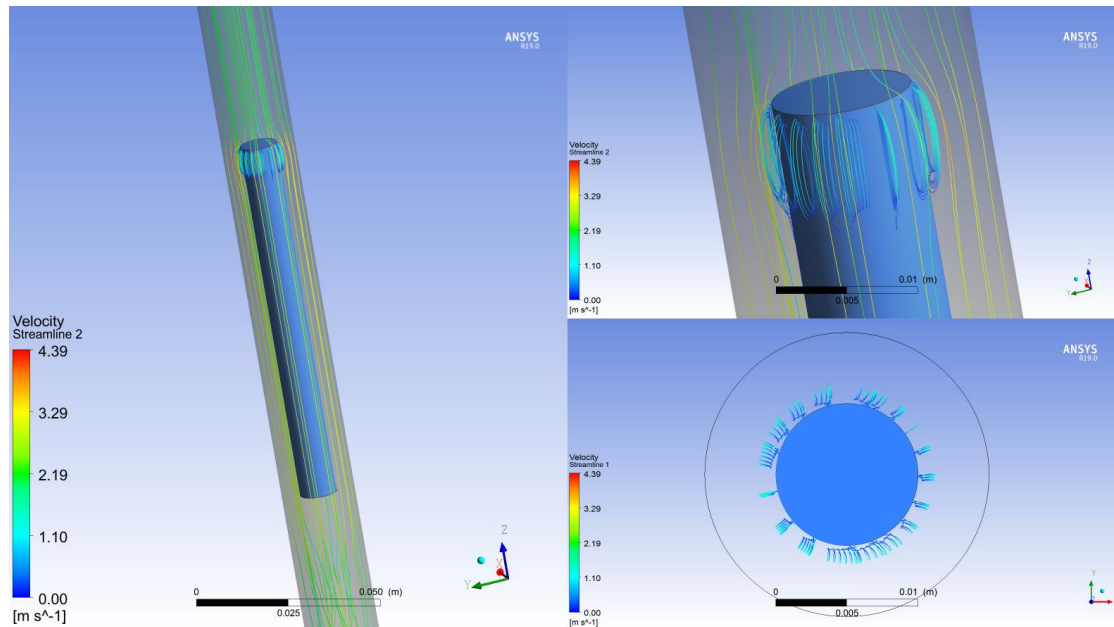
```
/define/boundary-conditions/mass-flow-inlet inlet y y n 0.6234 n 313 n 0 n y n n n y 4.0377199 0.02
```

# ANSYS Fluent TUI - issues

- TUI syntax is neither obvious nor fully documented. It is also confusing (or better to say frustrating) as some of the definitions rely heavily on some general case settings, you may need to change at some point.
- The definition may change either in number of arguments and/or their order! With no easy option to call the list of current requested arguments, with no if, while and any other loops to fill available arguments with possible values.
- Nightmare, you can compare only with the user experience of any early stage open-source code, with every new release bringing change in the layout or naming of your input deck. Imagine an input deck of 1000 lines!
- Fun fact: TUI syntax is more than 40 years old!

# The problem in question

- Boundary layer detachment is an enemy, we try to avoid in aerodynamics...
- It is hard to predict correctly even for the easiest axially symmetric flows.



# CFD solution workflow

- Make CAD geometry (2D/3D)
- Discretize it – generate mesh upon the geometry
- Define material properties and flow conditions
- Choose solution strategy (RANS/LES/DNS)\*
- Run simulation
- Extract and post-process simulation results
- Validation and verification

# The domain and the flow conditions

$$T_{inlet} = 313 \text{ K (40}^\circ\text{C)}$$

$$p_{ref} = 101325 \text{ Pa}$$

$$q''_i = q''_o = 10\,000 \text{ W/m}^2$$

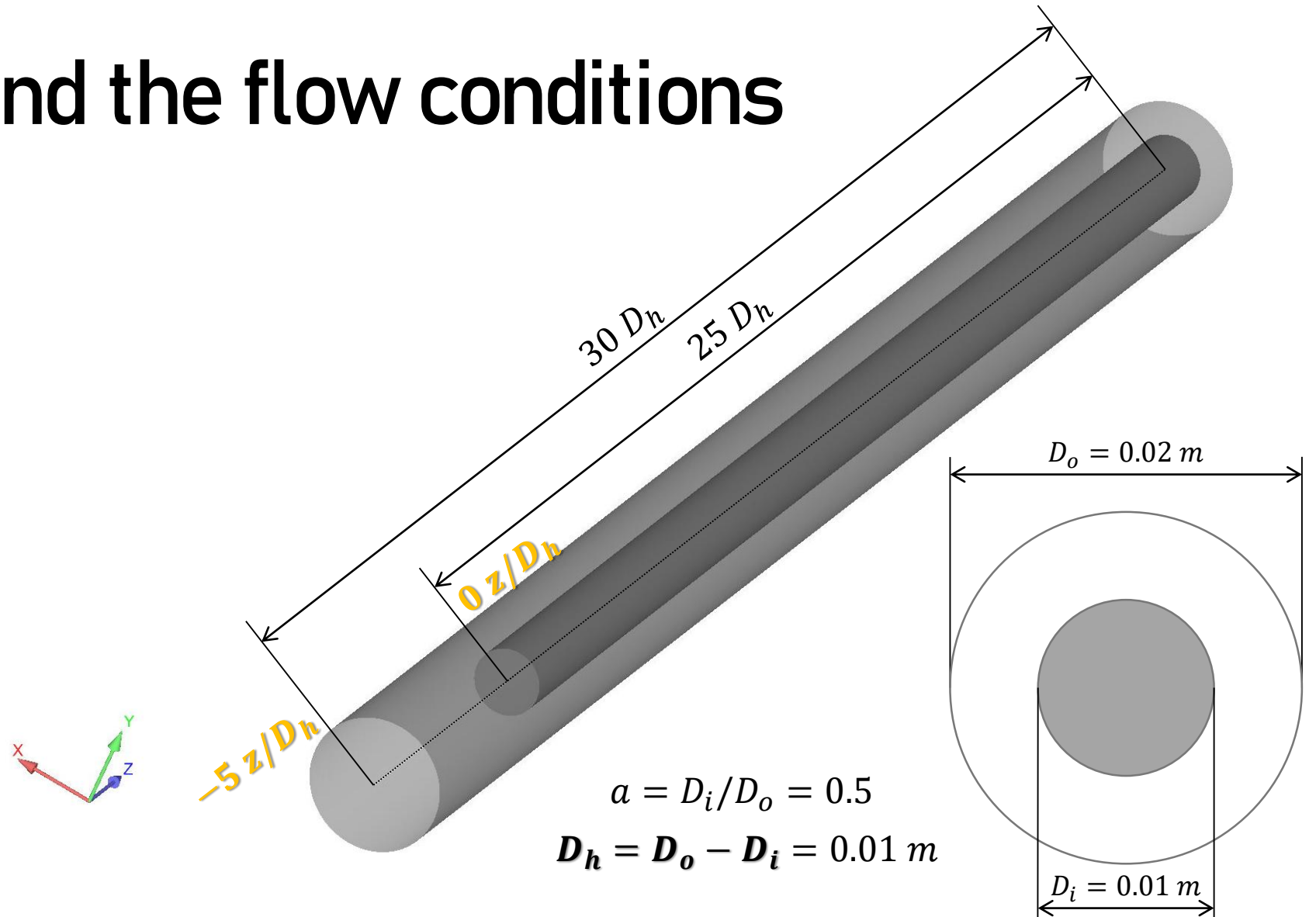
$$U_{inlet} = 0.44, 0.8, 2.0 \text{ m/s}$$

$$\rho = 992.18 \text{ kg/m}^3$$

$$\mu = 6.528 \times 10^{-4} \frac{\text{kg}}{\text{m} \cdot \text{s}}$$

$$c_p = 4178.82 \frac{\text{J}}{\text{kg} \cdot \text{K}}$$

$$\lambda = 0.6304 \frac{\text{W}}{\text{m} \cdot \text{K}}$$



# Dimensionless approach

$$a = \frac{D_i}{D_o} = 0.5 \Rightarrow BR = 25\%$$

$$U_{\text{annular}} = U_{\text{inlet}} \frac{D_{\text{outer}}^2}{D_{\text{outer}}^2 - D_{\text{inner}}^2}$$

$$q^* = q''_i / q''_o = 1$$

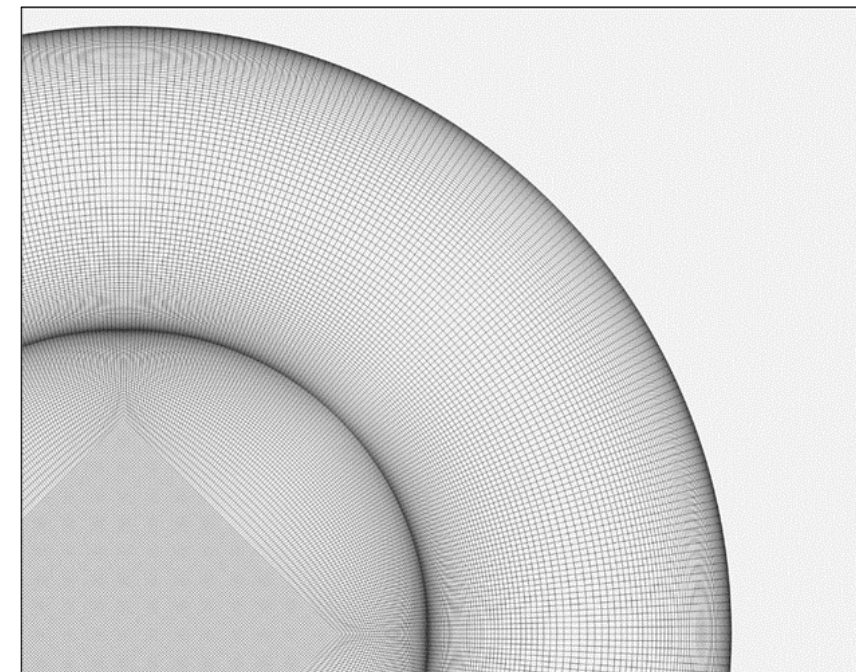
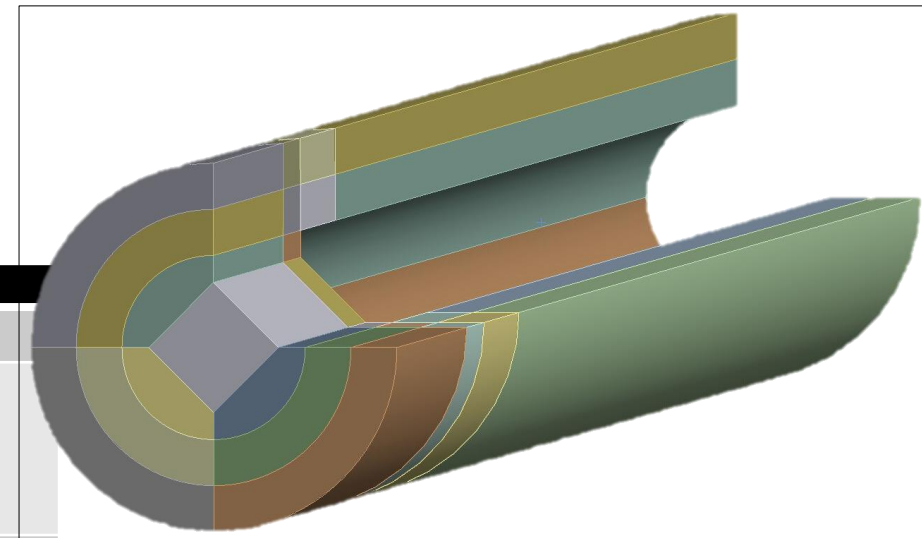
$$Re = \frac{\rho U D_h}{\mu} = 8\,900, 16\,200, 40\,500$$

$$Pr = \frac{c_p \mu}{\lambda} = 4.327$$

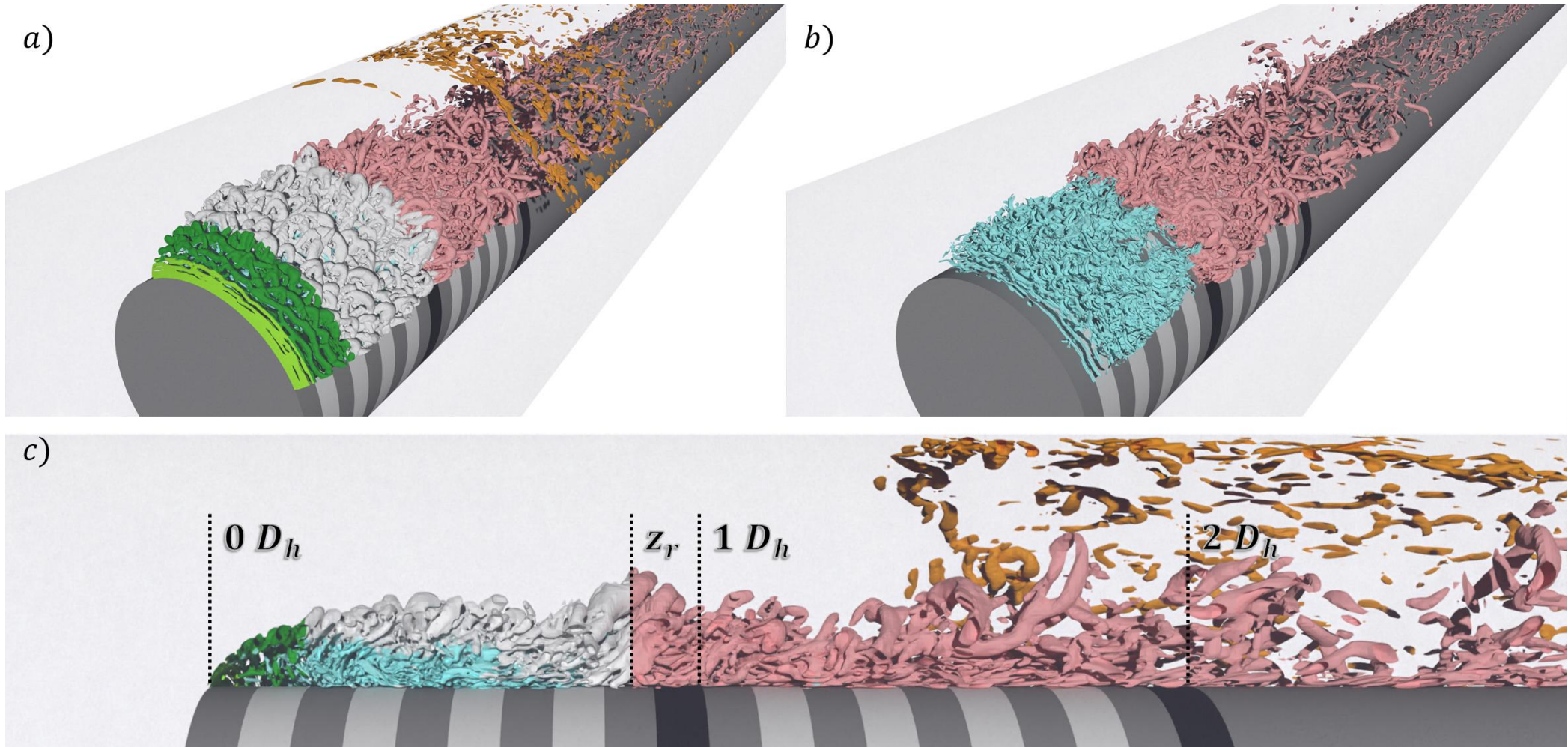
$$Nu = \frac{-\left. \frac{dT}{dr} \right|_w D_h}{T_{\text{wall}} - T_{\text{ref}}} = ?$$

# Mesh

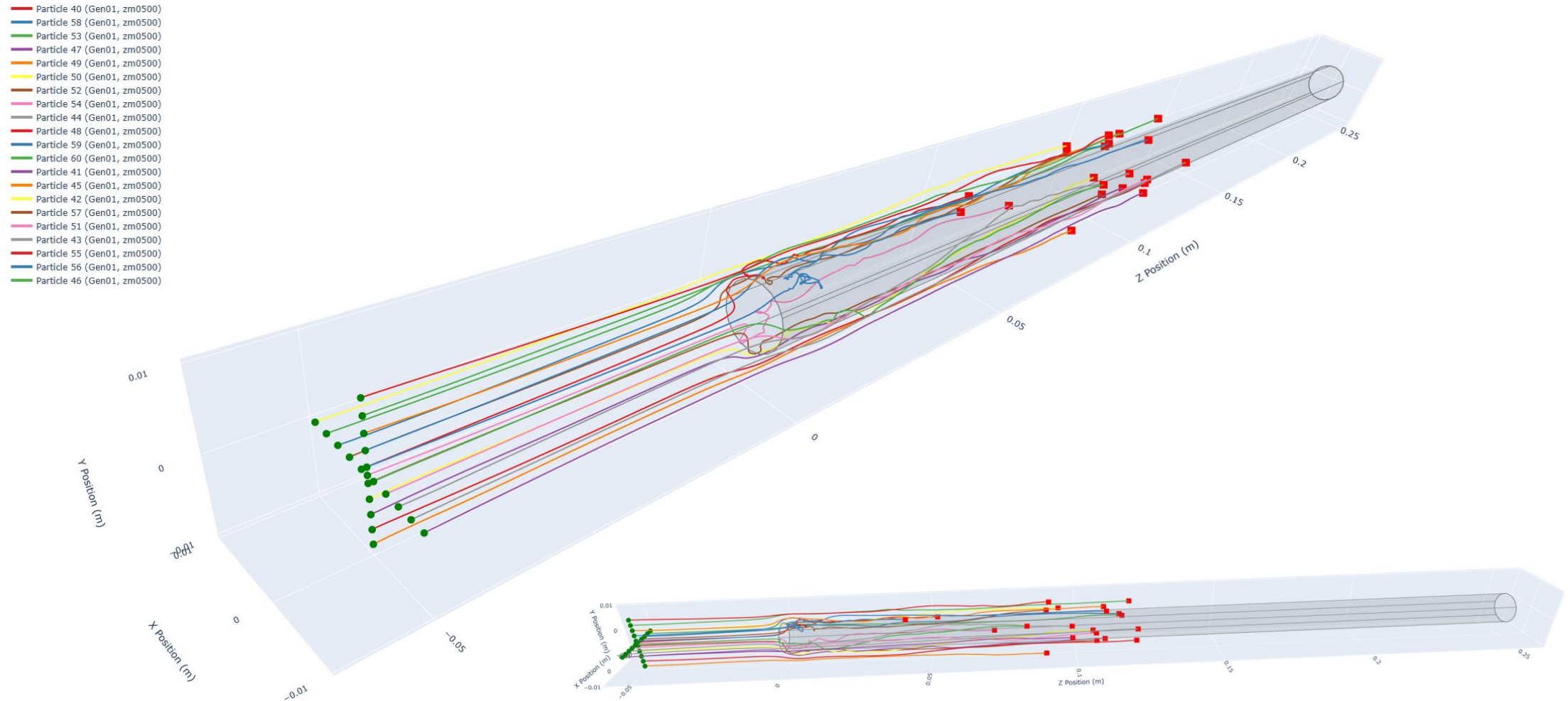
Parameter	Value			
$z/D_h$ range	-5 : -1	-1 : 0	0 : 2	2 : 25
Time step [s]	$10^{-5}$ , $\max(\text{CFL}_{\text{Re}=40500})=2.6$ , $\max(\text{CFL}_{\text{Re}=16200})=1.05$ , $\max(\text{CFL}_{\text{Re}=8900})=0.56$			
Element type	hexahedral			
First element layer height to hydraulic diameter (at both cylindrical walls)	$6.5 \times 10^{-4}$	$6.5 \times 10^{-4}$	$6.5 \times 10^{-4}$	$6.5 \times 10^{-4}$
Number of elements in axial direction - $N_z$	333	250	500	1917
Number of elements in radial direction - $N_r$	235	235	120	120
Number of elements in azimuthal direction - $N_\theta$	480	480	480	480
Total number of elements - $N_{total}$	196 586 400			
Total number of nodes - $N_{nodes}$	197 985 704			
Element growth rate in axial direction	1	1.012	1.006	1
Maximal cell size to hydraulic diameter ratio in axial direction - $\max(\Delta_z)/D_h$	$1.2 \times 10^{-2}$	$1.2 \times 10^{-2}$	$1.26 \times 10^{-2}$	$1.2 \times 10^{-2}$
Element growth rate in radial direction	max 1.052	max 1.052	max 1.052	max 1.052
Maximal cell size to hydraulic diameter ratio in radial direction - $\max(\Delta_r)/D_h$	$1.3 \times 10^{-2}$	$1.3 \times 10^{-2}$	$1.3 \times 10^{-2}$	$1.3 \times 10^{-2}$
Element growth rate in azimuthal direction	1	1	1	1
Maximal cell size to hydraulic diameter ratio in circumferential direction - $\max(\Delta_\theta)/D_h$	$1.3 \times 10^{-2}$	$1.3 \times 10^{-2}$	$1.3 \times 10^{-2}$	$1.3 \times 10^{-2}$



# Flow physics (based on $Re = 40\ 500$ )

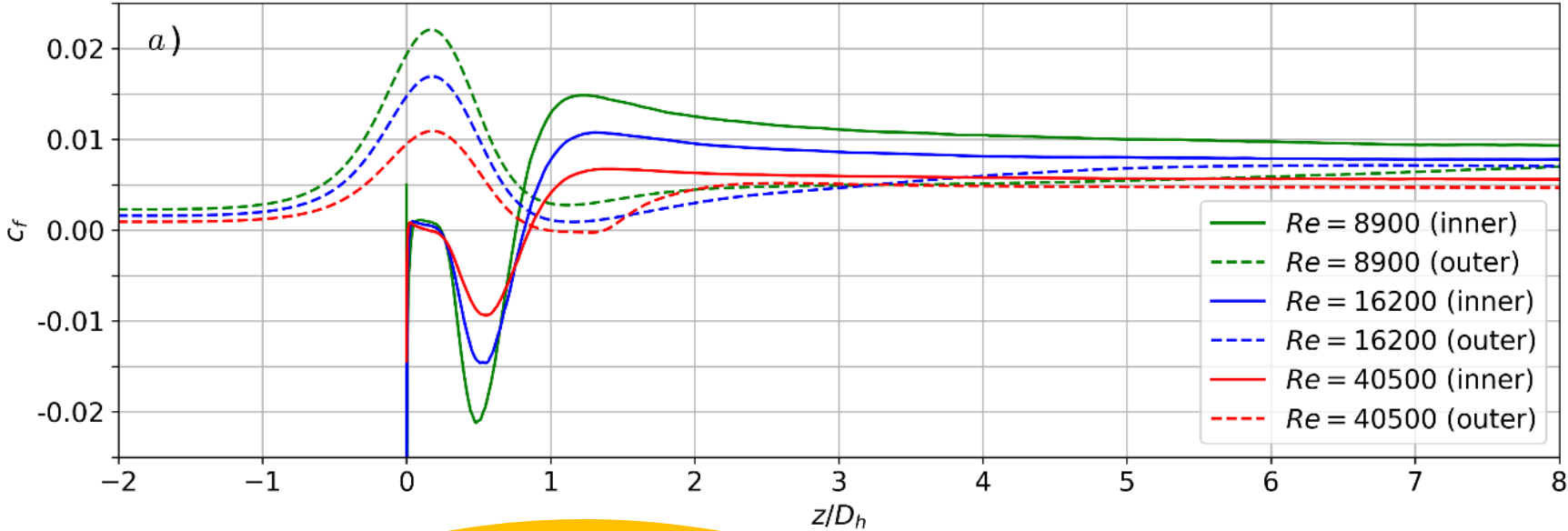


# Flow physics (based on $Re = 40\ 500$ )



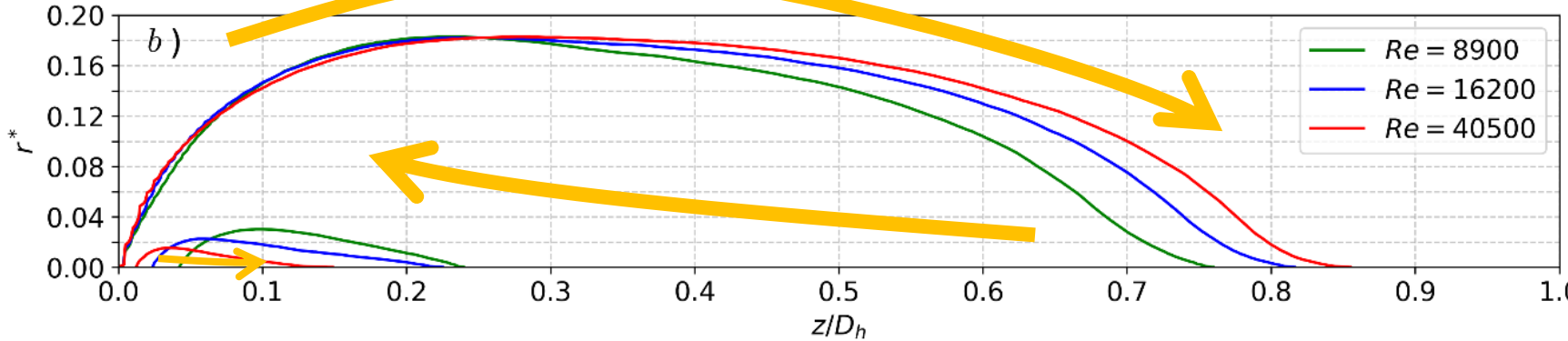
# Does the separation length depend on velocity?

$$c_f = \frac{\tau_{w,z}}{\frac{1}{2} \rho U_{annular}^2}$$

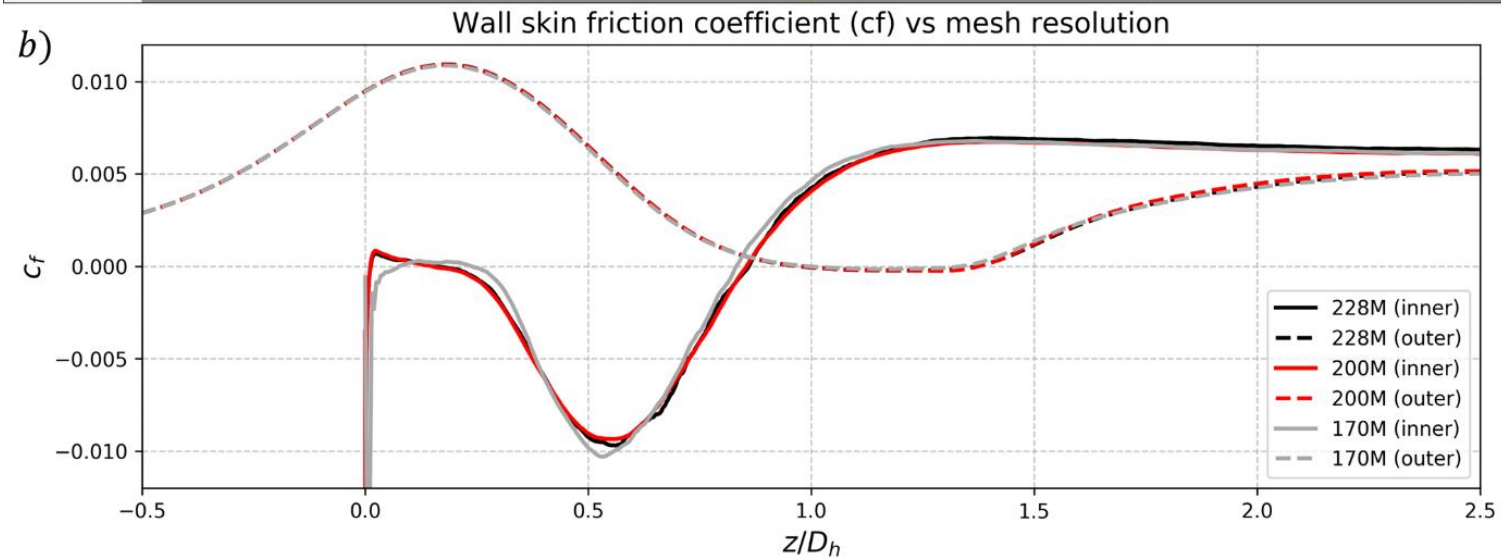
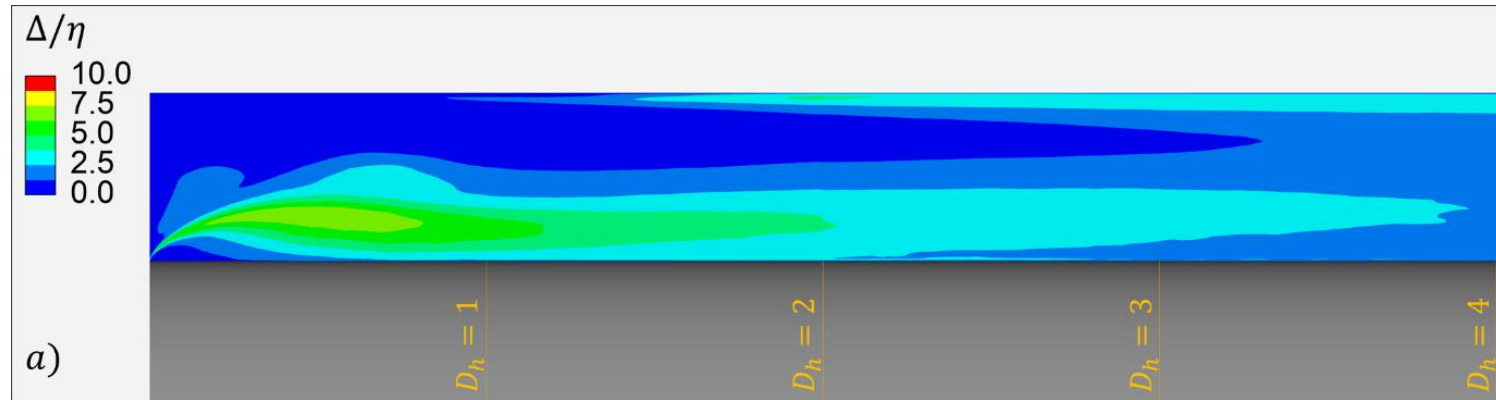


$$r^* = \frac{r - r_{inner}}{r_{outer} - r_{inner}}$$

$$U_z = 0$$

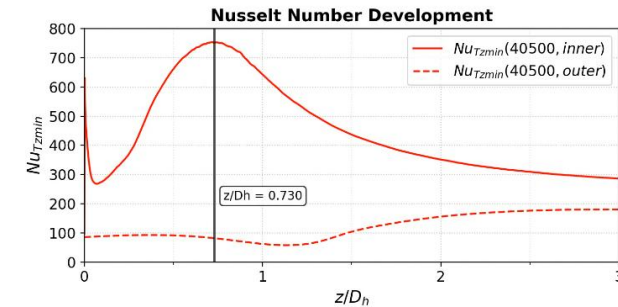
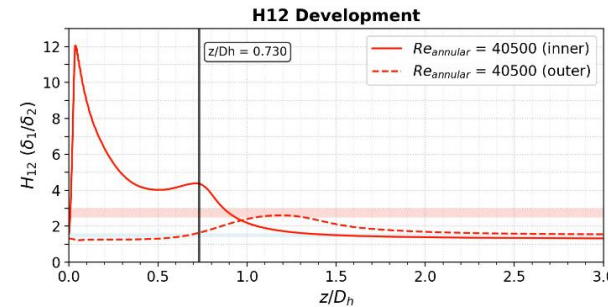
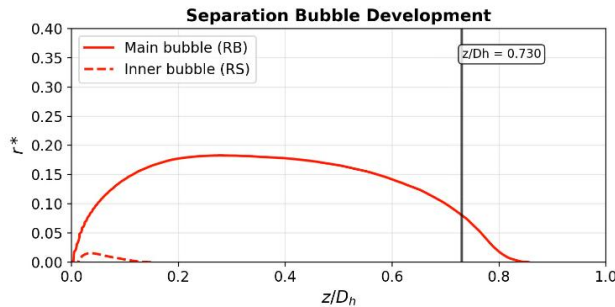
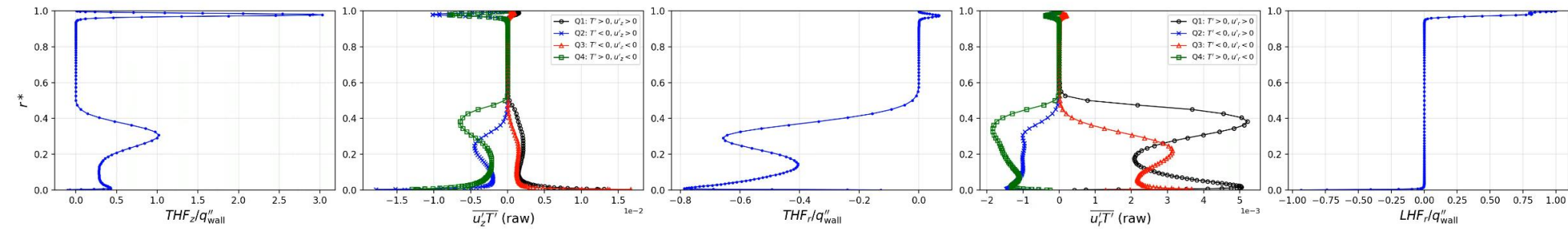
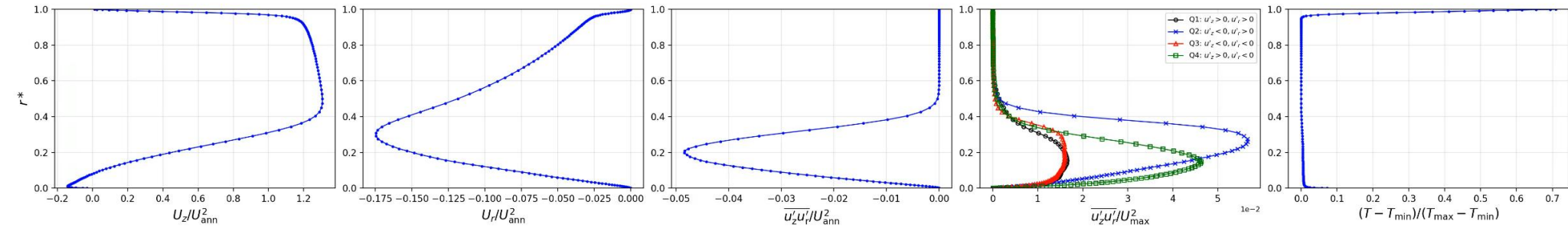


# Why we believe this is true?



# CFD fuels multi-variable analysis

Re = 40500 at z/Dh = 0.73 (Range: 0.000-25.000)



# The End

*Its the time to ask all these questions  
I did not cover with my presentation*

**Piotr Prusiński, MSc. Eng.** | CFD Analysis Group  
Division of Nuclear Energy and Environmental Studies (UZ3)  
Department of Complex Systems (DUZ)  
National Centre for Nuclear Research (NCBJ)  
A. Soltana 7, 05-400 Otwock-Świerk  
e-mail: [piotr.prusinski@ncbj.gov.pl](mailto:piotr.prusinski@ncbj.gov.pl)  
tel: +48 22 273 11 26

**Tomasz Früboes, PhD**  
Software Engineering Division (UZ2)  
Department of Complex Systems (DUZ)  
National Centre for Nuclear Research (NCBJ)  
A. Soltana 7, 05-400 Otwock-Świerk  
e-mail: [tomasz.fruboes@ncbj.gov.pl](mailto:tomasz.fruboes@ncbj.gov.pl)