ANSYS CFD – SESSION1

Jurij Gregorc, University of Ljubljana, Faculty of Mechanical Engineering

9/2022

Univerza v Ljubljani

TU WIEN TECHNISCHE UNIVERSITÄT WIEN

CINECA

VSB TECHNICAL UNIVERSITY OF OSTRAVA | IT4INNOVATIONS NATIONAL SUPERCOMPUTING CENTER

Co-funded by the Erasmus+ Programme of the European Union

This project has been funded with support from the European Commission.
This publication [communication] reflects the views only of the author, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

Agenda

Jurij Gregorc, University of Ljubljana, FME

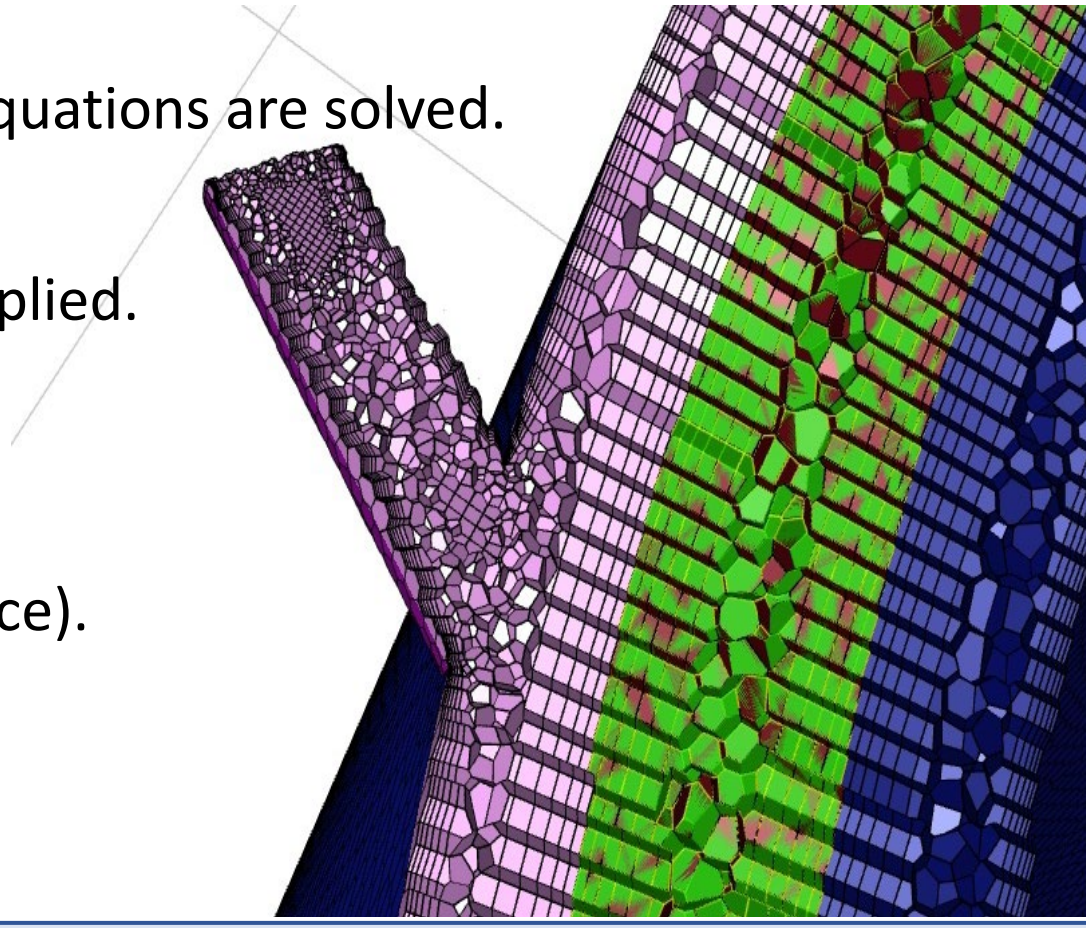
- Why meshing?
- Building blocks and terminology
- Mesh types
- Cell types
- Mesh quality
- Hands on: building CFD mesh for simulation of laminar flow in 2D U-bend channel

Why meshing?

Jurij Gregorc, University of Ljubljana, FME

Why do we need mesh/grid?

- Designates the “elements” on which the flow equations are solved.
- Gives discrete representation of the geometry.
- Has entities (groups of cells) where b.c.’s are applied.



The mesh has impact on:

- Rate of convergence (or even lack of convergence).
- Solution accuracy.
- CPU time required.

Building blocks

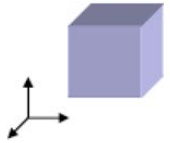
- Geometry is the starting point of any CFD study

  Geometry can be:

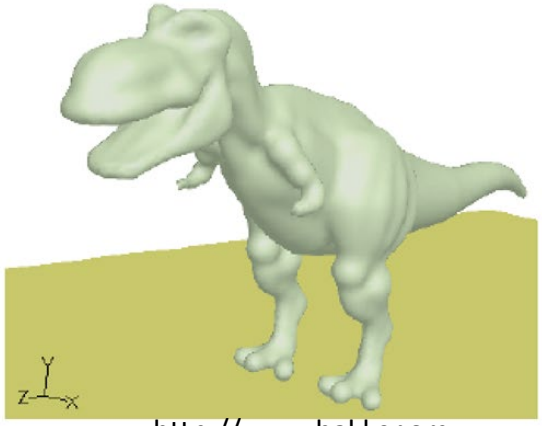
  - Basic or complex
  - Imported form CAD
  - Created using bottom-up/top-down/hybrid approach

  Is composed of:

  - Volumes
  - Surfaces
  - Curves
  - points



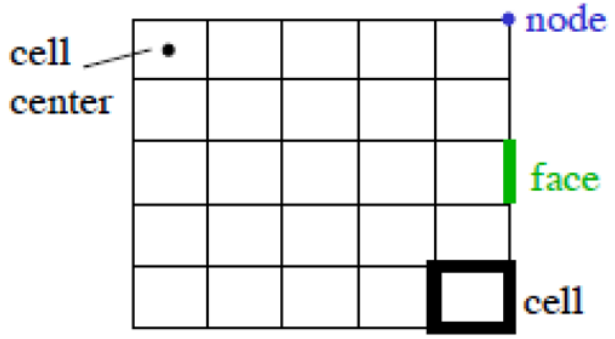
geometry for a "cube"



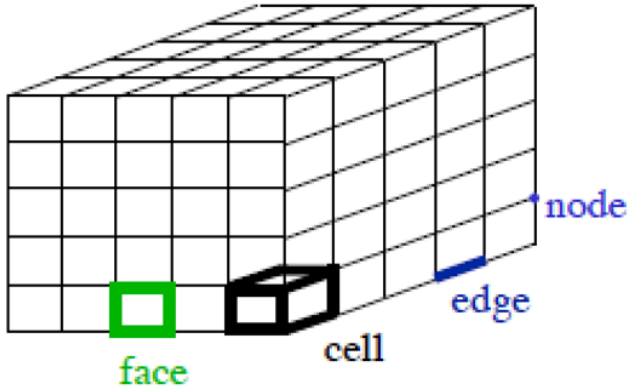
http://www.bakker.org

Mesh types

Jurij Gregorc, University of Ljubljana, FME



2D computational grid



3D computational grid

http://www.bakker.org

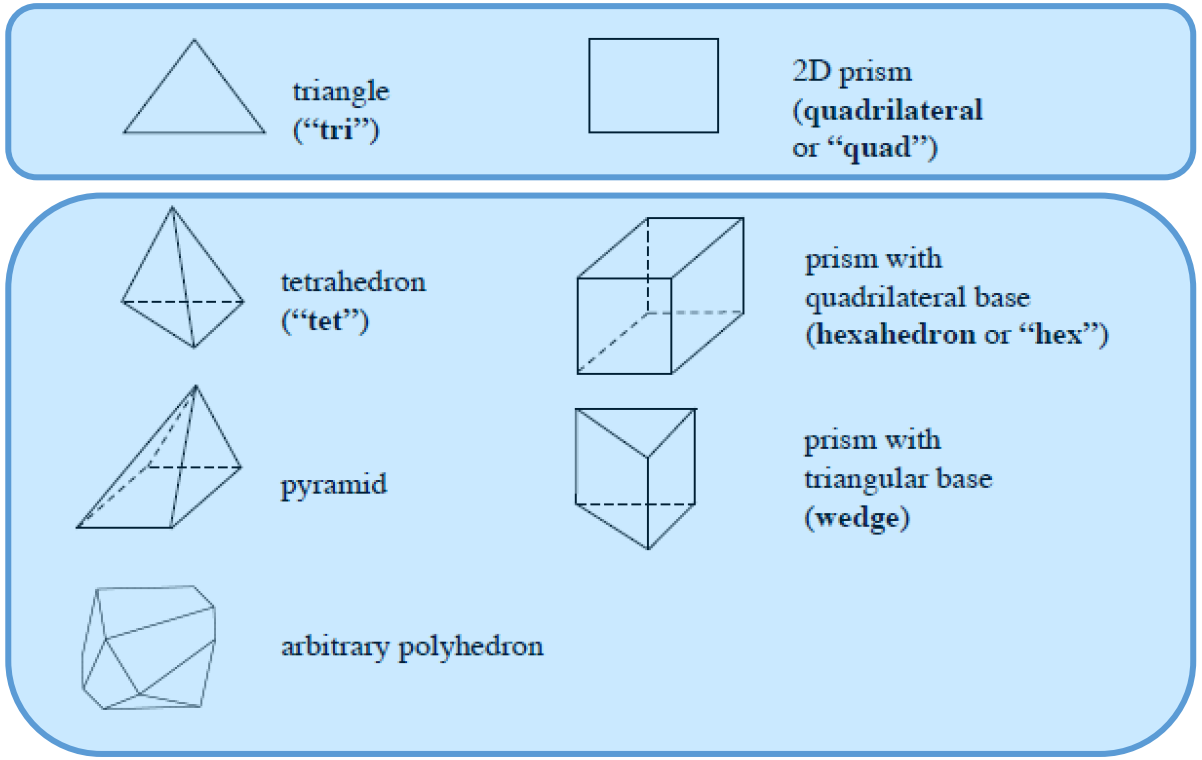
Mesh terminology

- Cell – basic volume unit of the mesh
- Node – point when sides of different cells meet
- Cell center – center of the cell
- Edge – boundery of face
- Face – boundary of cell
- Zone – group of cells, faces, nodes
- Domain – group of zones

Cell types

Jurij Gregorc, University of Ljubljana, FME

Cell types



Mesh types

- Structured
- Unstructured
- Hybrid

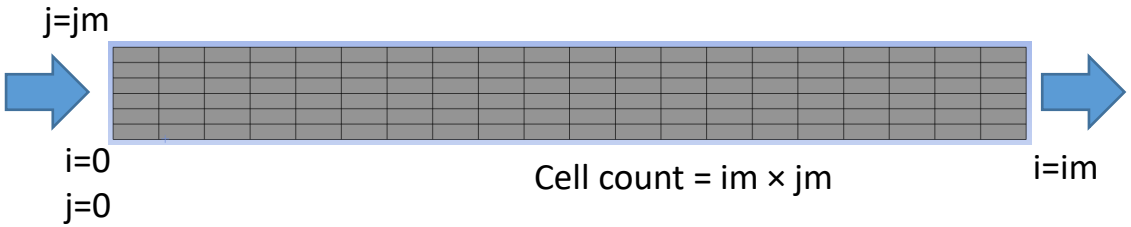
http://www.bakker.org

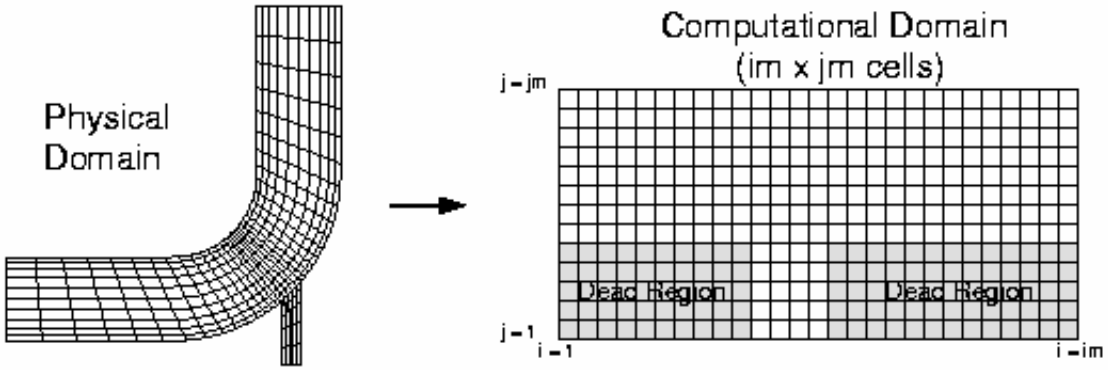
Mesh types

Jurij Gregorc, University of Ljubljana, FME

Structured mesh

- The cells distribution follows the flow topology
- The cells are indexed
- Blocking analogy is used
- Structured meshes are of high quality and very good cell count/volume ratio
- Harder to do (compared to unstructured)
- Limited use for highly complex geometry

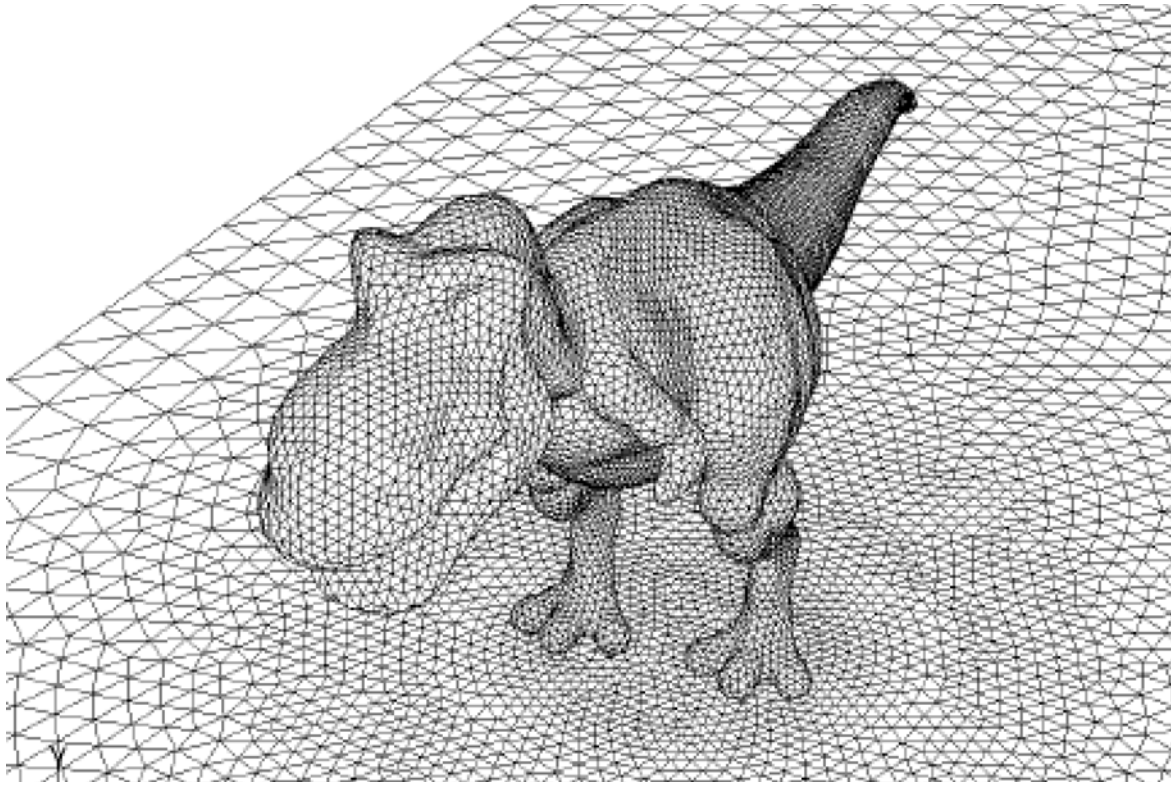




http://www.bakker.org

Mesh types

Jurij Gregorc, University of Ljubljana, FME



http://www.bakker.org

Unstructured mesh

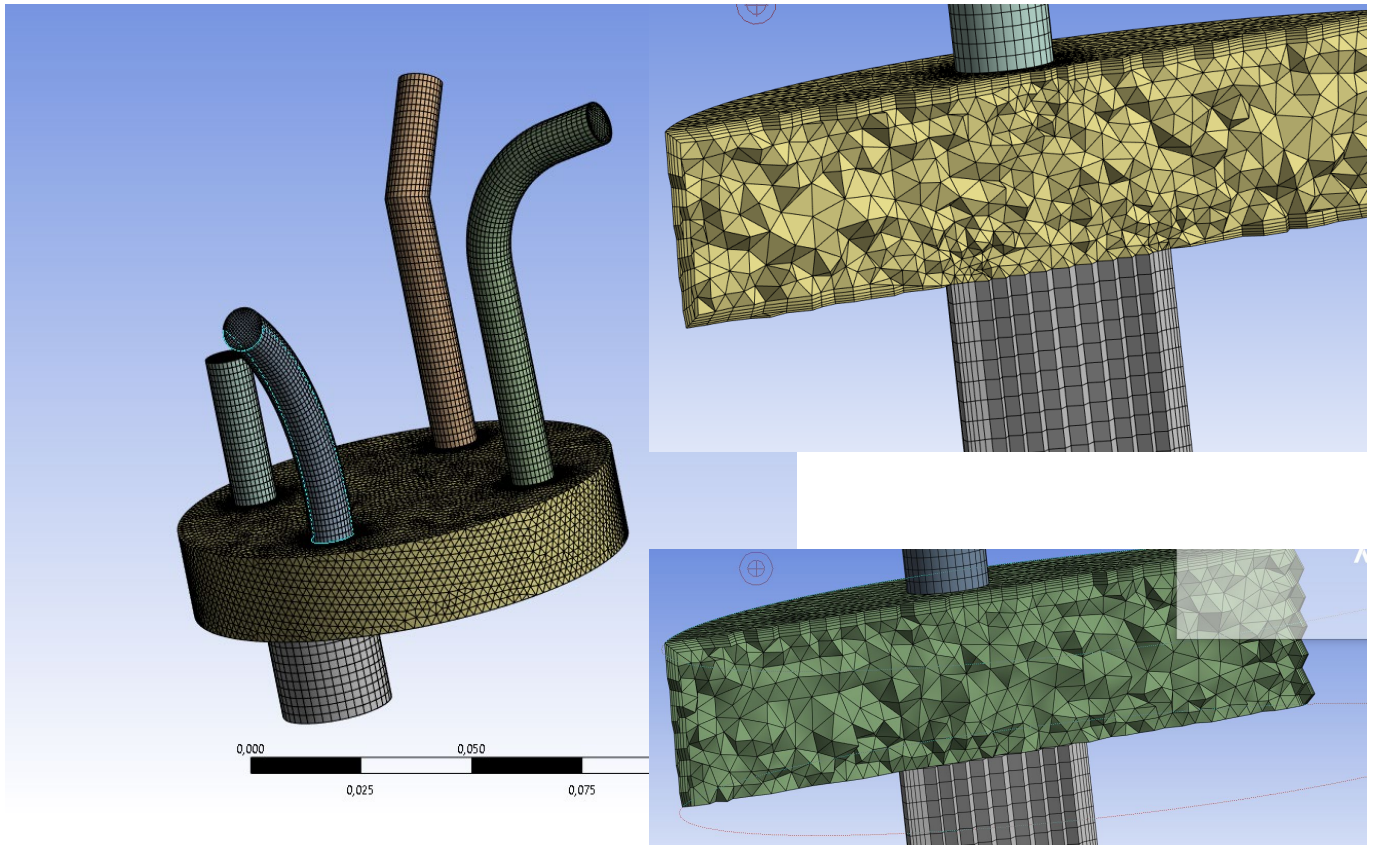
- The cells arbitrarily distributed
- No indexation of cells
- Meshing can be highly automized
- useful for highly complex geometry

Mesh types

Jurij Gregorc, University of Ljubljana, FME

"Hybrid" mesh

- Tetrahedral mesh in complex regions
- Hexahedral mesh in simpler regions
- Conformal vs non-conformal



General guidelines

- Hexahedral mesh will give you superior results to tetrahedral mesh and lower cell count with the same cell size
- Mesh should be dense enough to capture flow phenomena
- Local refinements may be used in areas of high gradients

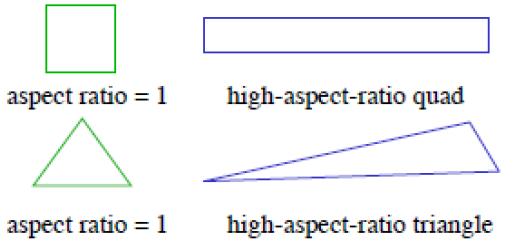
- No mesh is universal it has to be built with application in mind

Mesh quality

Jurij Gregorc, University of Ljubljana, FME

Quality metrics for shape of cells

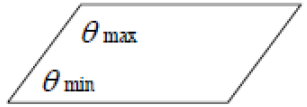
- Aspect ratio



- Skewness $\max\left[\frac{\theta_{max}-\theta_e}{180-\theta_e}, \frac{\theta_e-\theta_{min}}{\theta_e}\right]$

where:

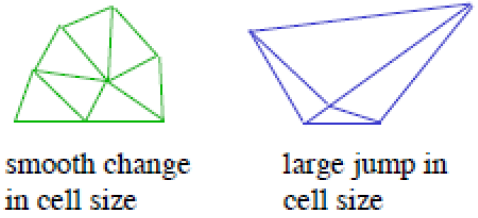
- θ_{max} = largest angle in face or cell.
- θ_{min} = smallest angle in face or cell.
- θ_e = angle for equiangular face or cell.
  - e.g., 60 for triangle, 90 for square.



Skewness should not be smaller than 0.85

Quality metrics distribution of cells

- Smoothness

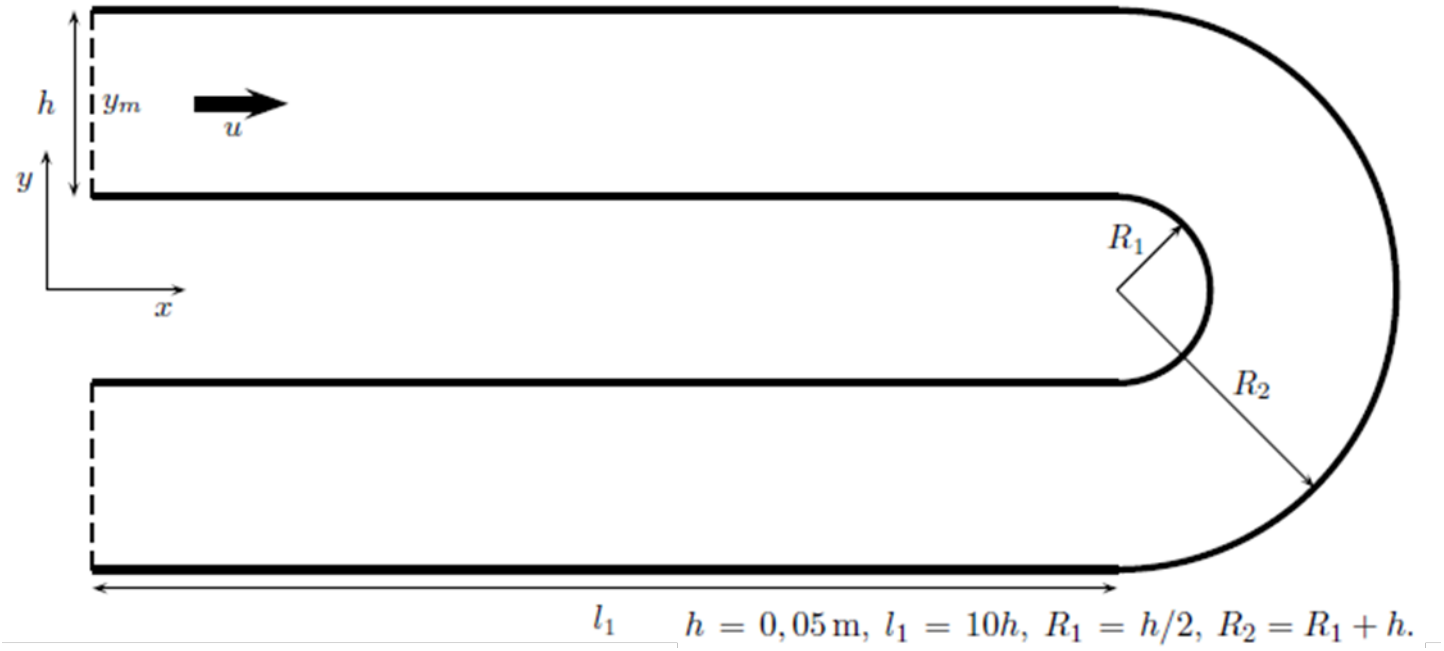


Sudden jump in cell size should be avoided
Growth ratio should not exceed 1.2

http://www.bakker.org

Hands on: 2D u-bend

Jurij Gregorc, University of Ljubljana, FME

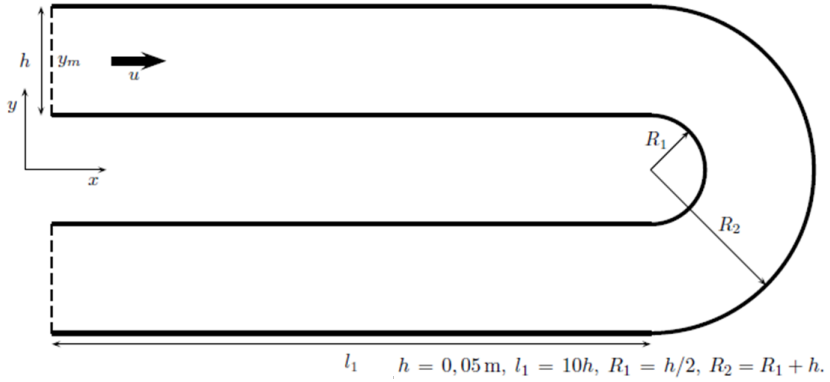


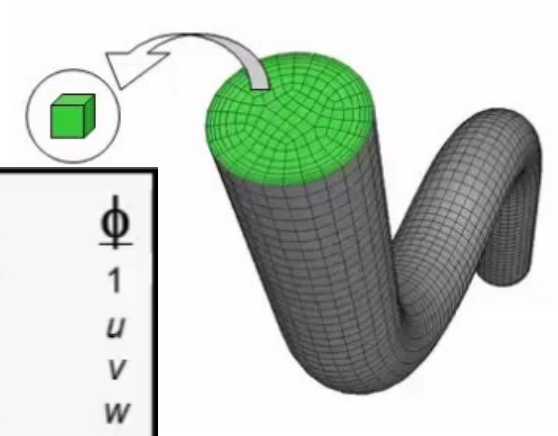
- Steady state, 2D, laminar, isothermal flow of water at standard conditions
  Re=500; ρ=998.2 kg/m^3; ν=1.003×10^{-6} m^{2}s^{-1}
- INLET: Parabolic velocity profile
- OUTLET: Constant ambient pressure

Hands on: 2D u-bend

Jurij Gregorc, University of Ljubljana, FME

$$\underbrace{\frac{\partial}{\partial t}\int_V \rho\phi\, dV}_{\text{Transient}} + \underbrace{\oint_A \rho\phi\mathbf{V}\cdot d\mathbf{A}}_{\text{Convection}} = \underbrace{\oint_A \Gamma_\phi \nabla\phi \cdot d\mathbf{A}}_{\text{Diffusion}} + \underbrace{\int_V S_\phi\, dV}_{\text{Source}}$$





Equation	ϕ
Continuity	1
X momentum	u
Y momentum	v
Z momentum	w
Energy	h

- How many equation do we have to solve (which variables)
- Which terms can be neglected
- Any special consideration for meshing?

End of Session 1

Thank you for your attention!

http://sctrain.eu/

Co-funded by the Erasmus+ Programme of the European Union

This project has been funded with support from the European Commission.
This publication [communication] reflects the views only of the author, and the Commission cannot be held responsible for any use which may be made of the information contained therein.