Budapest University of Technology and Economics
Department of Aeronautics, Naval Architecture and Railway Vehicles
Budapest, H-1111, Sztoczek street 6. building J. 4th floor, Hungary
Tel.: +36-1-463-1922, Fax: +36-1-463-30-80

Computational Heat Transfer and Fluid Dynamics

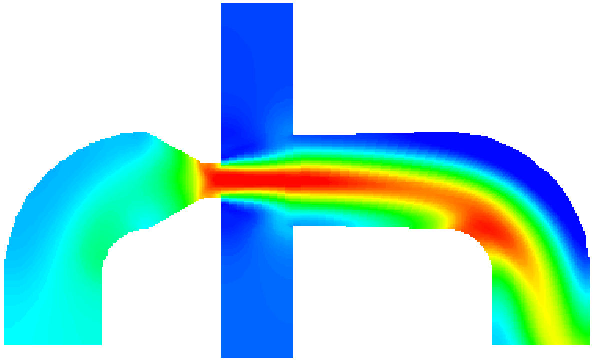
BMEKOVRM606

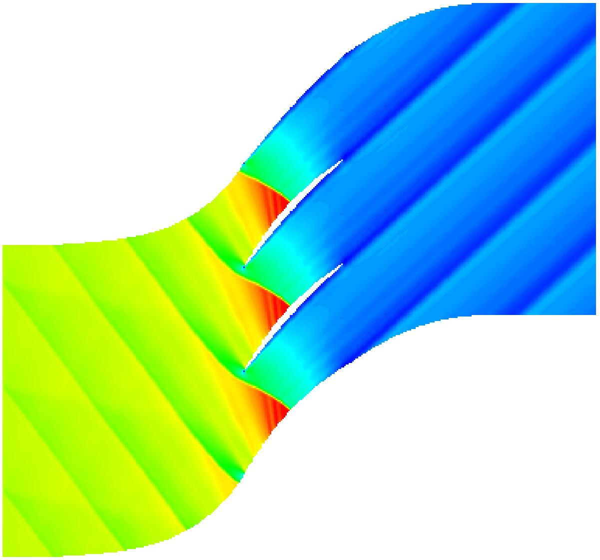
Dr. Árpád Veress
associate professor

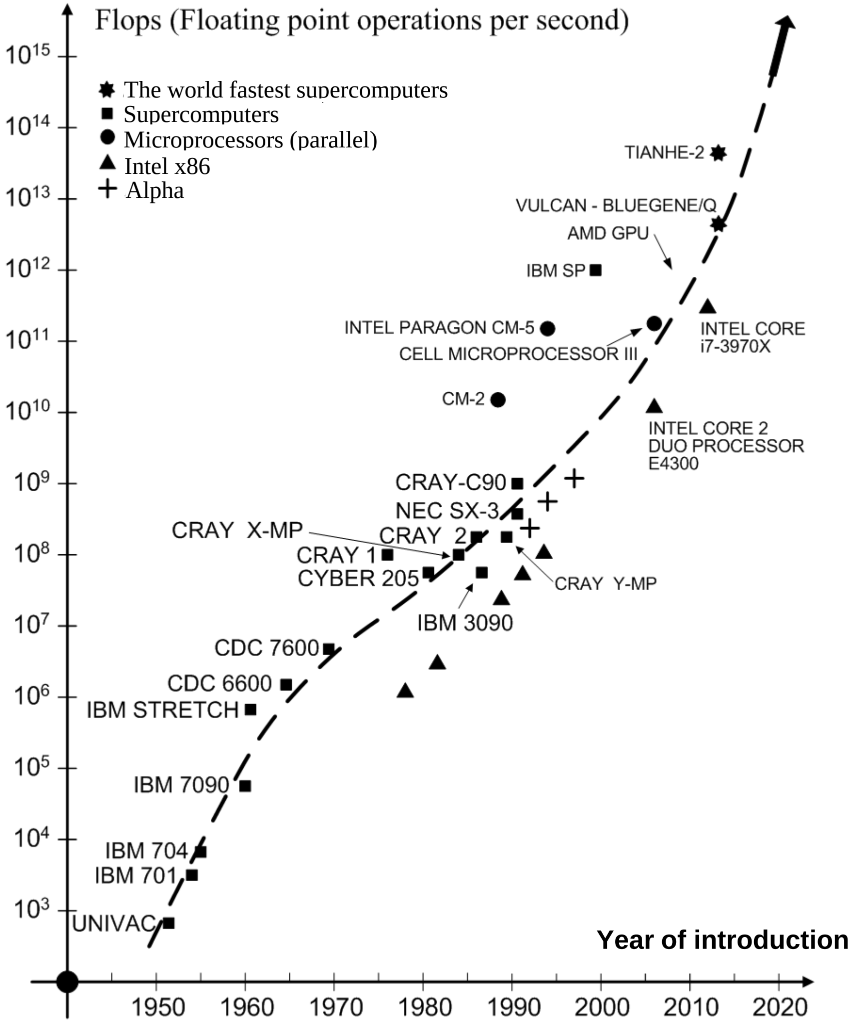
Budapest, 07-09-2020

Introduction

Actuality



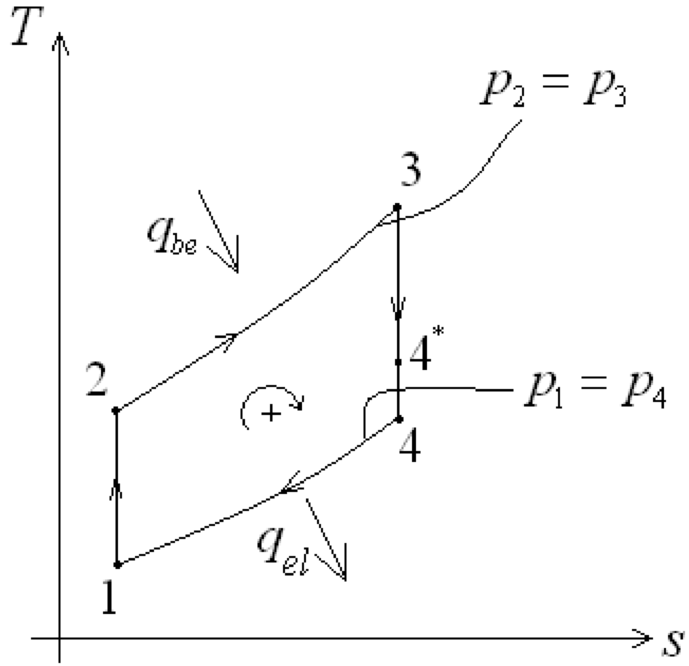


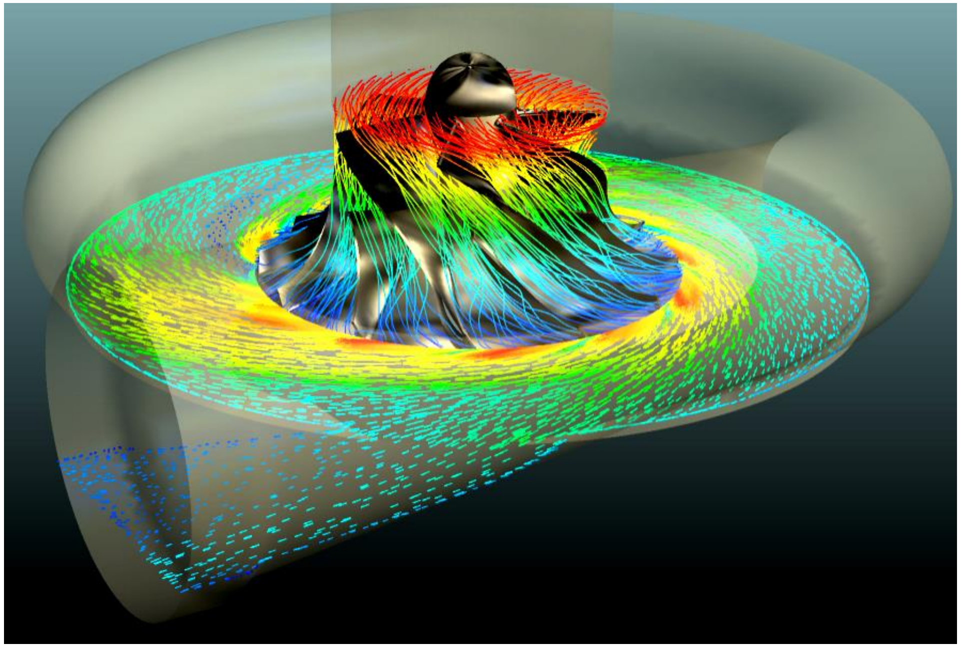

Flops (Floating point operations per second)
The world fastest supercomputers
Supercomputers
Microprocessors (parallel)
Intel x86
Alpha
TIANHE-2
VULCAN - BLUEGENE/Q
AMD GPU
IBM SP
INTEL PARAGON CM-5
CELL MICROPROCESSOR III
INTEL CORE i7-3970X
CM-2
INTEL CORE 2 DUO PROCESSOR E4300
CRAY-C90
NEC SX-3
CRAY X-MP
CRAY 2
CRAY 1
CYBER 205
CRAY Y-MP
IBM 3090
CDC 7600
CDC 6600
IBM STRETCH
IBM 7090
IBM 704
IBM 701
UNIVAC
Year of introduction
10^{15}
10^{14}
10^{13}
10^{12}
10^{11}
10^{10}
10^{9}
10^{8}
10^{7}
10^{6}
10^{5}
10^{4}
10^{3}
1950
1960
1970
1980
1990
2000
2010
2020

Approaches for Modelling and Conditions for Applications

Description of physical processes - basic approaches of mathematical models in space

1. Concentrated parameter type **2. Distributed parameter type**





Thermodynamics, Heat Transfer and Fluid Dynamics → Possibilities of the application of the governing equations

Approaches for Modelling and Conditions for Applications

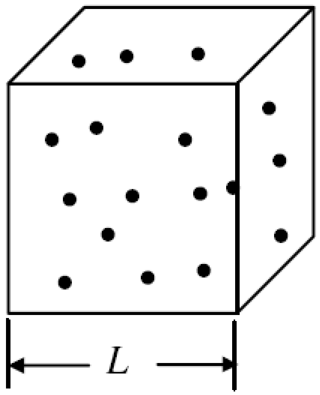
Description of physical processes - basic approaches for mathematical modelling:

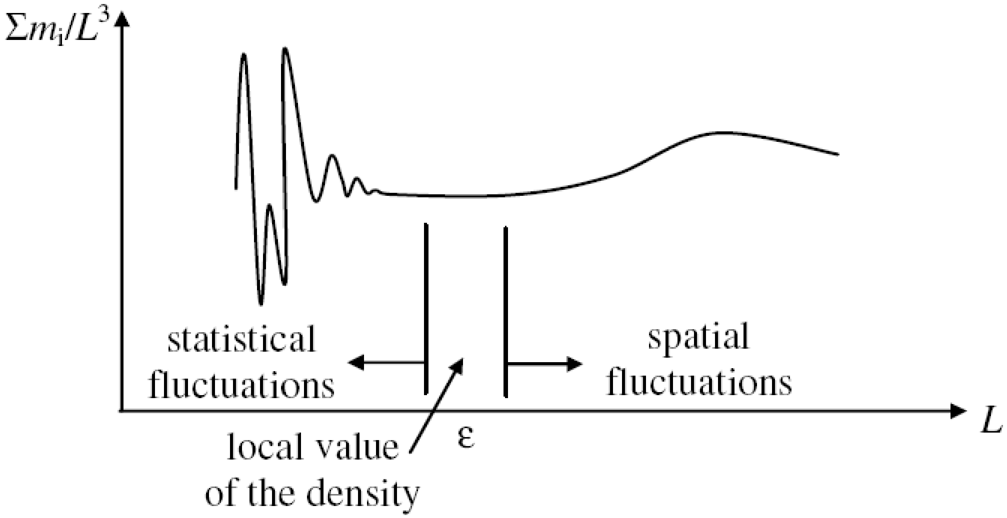
1. Statistical Physics (based on statistical mechanics, kinetic theory of gases)

2. Continuum Mechanics

Definition of Length scale

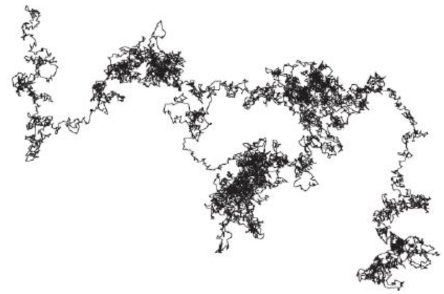
$$\rho \equiv \lim_{L \to \varepsilon} \frac{\sum m_i}{L^3}$$

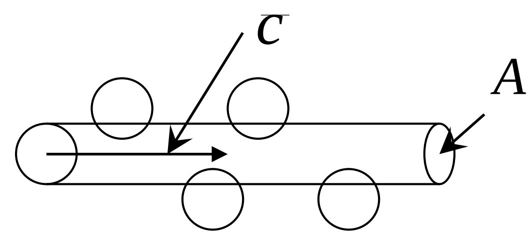




Approaches for Modelling and Conditions for Applications

Robert Brown – 1827
Brownian motion





Length scale - mean free path of the molecule

Swept volume of a molecule by unit time: $\dot{V} = \bar{c}A \quad [m^3 / s]$

Then, the number of collisions with other molecule by unit time:

$$\dot{n} = n\dot{V} \quad [db / m^3][m^3 / s] = [db / s]$$

The average time between two collisions: $t' = 1/\dot{n} = 1/(n\dot{V}) = 1/(n\bar{c}A) \quad [s]$

The average path length between two collisions (mean free path of the molecule):

$\lambda = \bar{c}t' = 1/(nA)\,[m]$ For air in case of standard conditions: $n = 2{,}7e19\left[\frac{db}{cm^3}\right]$

$A = 1e-15\left[cm^2\right]$

In 160 Km far from the ground: $\lambda = 80[m]$

In shock waves: $\lambda = 1[\mu m] = 1e-4[cm]$

$\lambda = 3{,}7e-5[cm]$

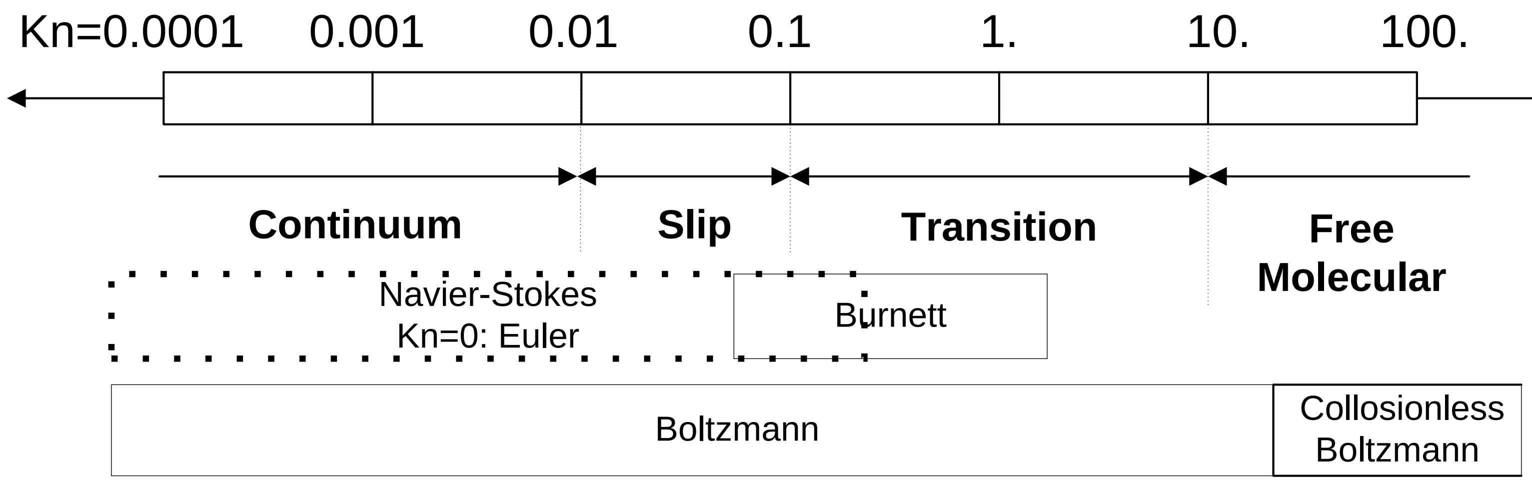
Approaches for Modelling and Conditions for Applications

Categorization of the Flow by Local Knudsen Number

1. Statistical Physics (based on statistical mechanics, kinetic theory of gases)

2. Continuum Mechanics

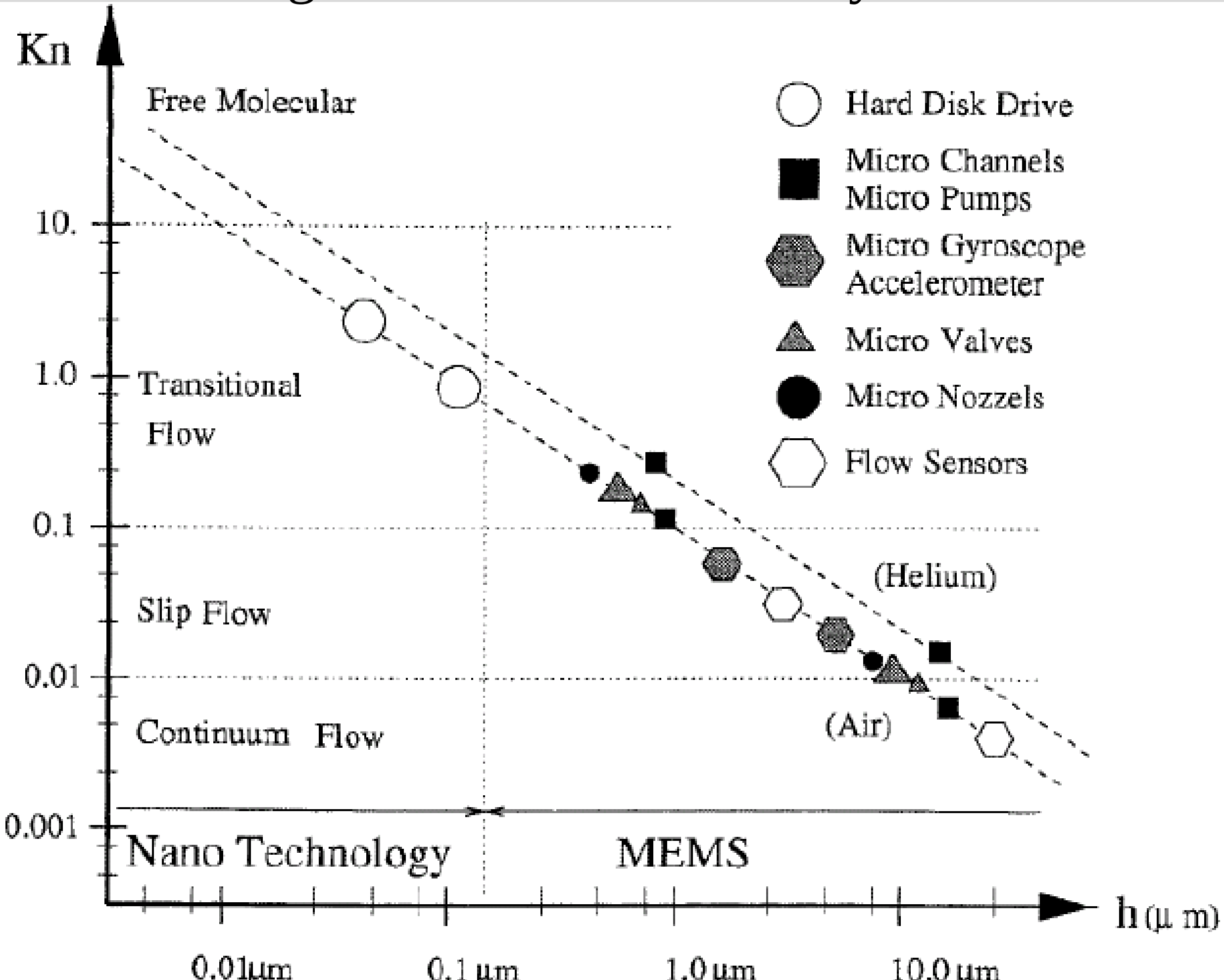
The importance of the local Knudsen number in case of decision about using continuum mechanics based approaches.



$$\mathrm{Kn} = \frac{\lambda}{L} = \frac{3{,}7e-7[m]}{0{,}03[m]} << 0{,}01$$

Approaches for Modelling and Conditions for Applications

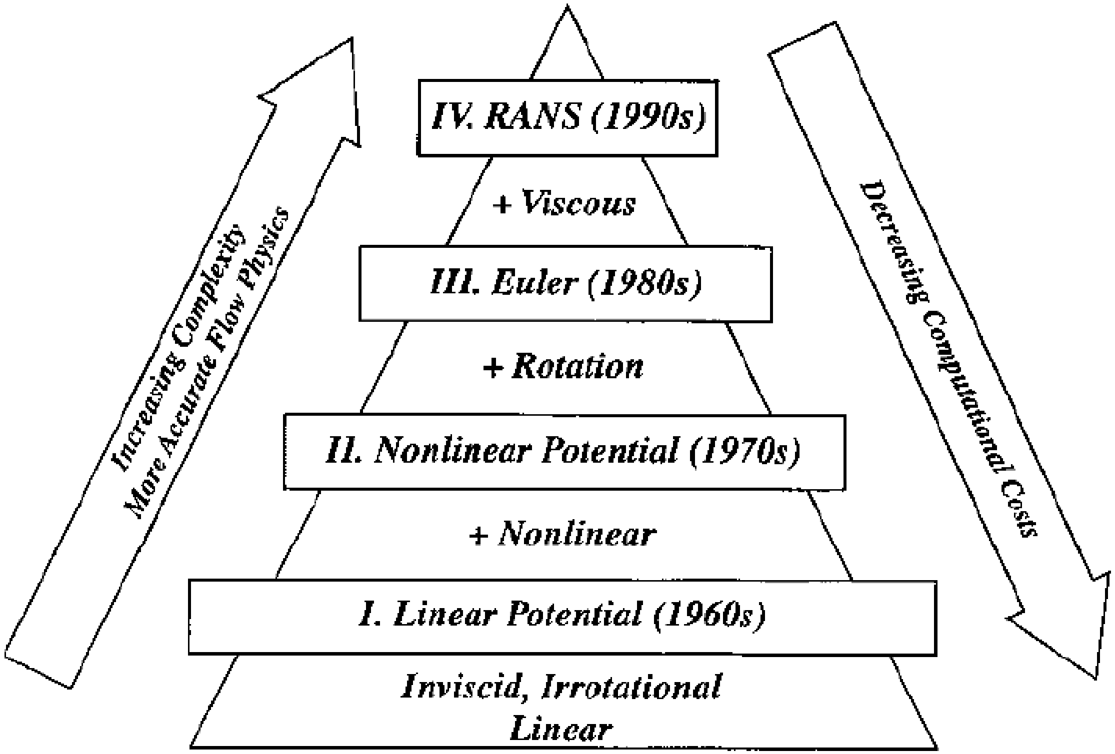
Categorization of the Flow by Local Knudsen Number



Source: **Xiao-Jun Gu and David R. Emerson: Application of the Moment Method in the Slip and Transition Regime for Microfluidic Flows, RTO-EN-AVT-194,** http://ftp.rta.nato.int/public//PubFullText/RTO/EN/RTO-EN-AVT-194///EN-AVT-194-11.pdf (2013.09.01.)

Mathematical Models of Flow – Governing Equations

Approaches for Modelling and Conditions for Applications



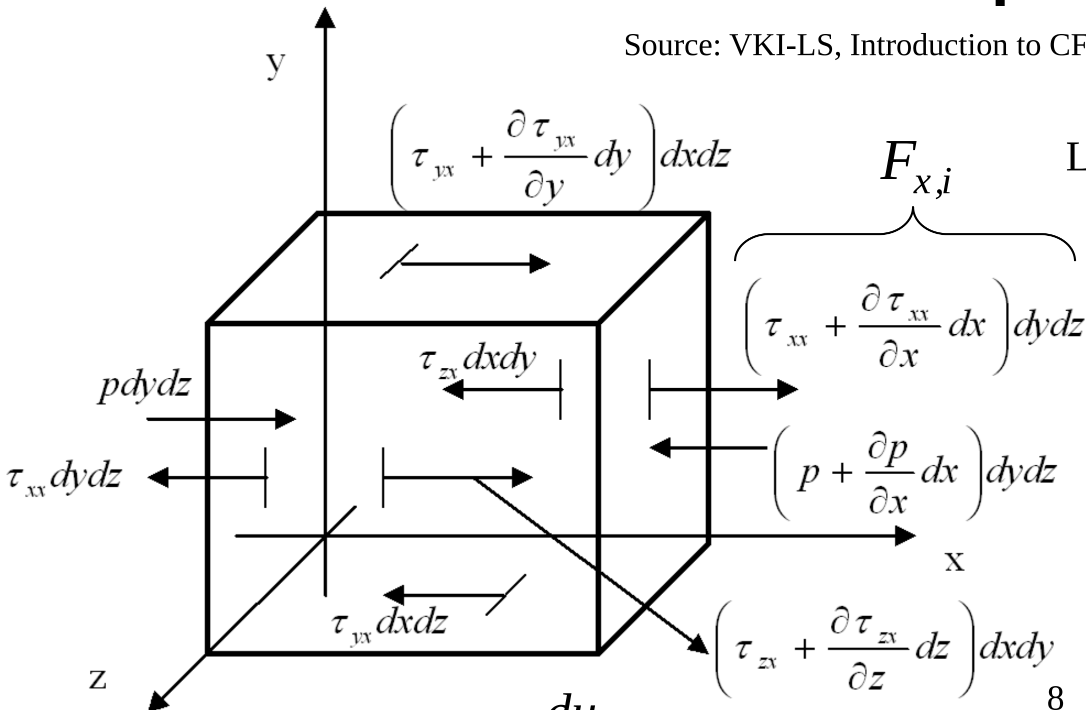
Hierarchy of Fluid Flow Models

Source: Antony Jameson: A perspective on computational algorithms for aerodynamic analysis and design, Progress in Aerospace Sciences, Volume 37, Issue 2, February 2001, Pages 197–243

http://aero-comlab.stanford.edu/Papers/SEVILLE.pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; Navier-Stokes Equations

Source: VKI-LS, Introduction to CFD



Lagrangian and Eulerian specifications

<u>Newton II. law in x direction:</u>

$$\boxed{ma_x = m\frac{du}{dt} = \sum_{i=1}^{8} F_{x,i}}$$

$$m = \rho dxdydz$$

$$\boxed{m\frac{du}{dt} = dx\,dy\,dz\rho\frac{du}{dt} = \sum_{i=1}^{8} F_{x,i}}$$

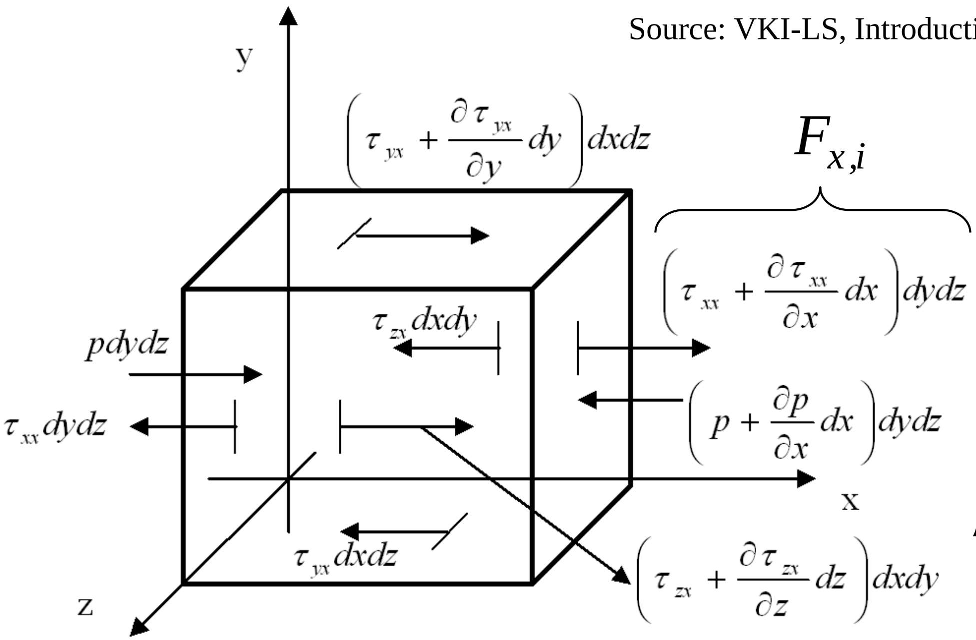
$$\rho\frac{du}{dt} = \rho\frac{\partial u}{\partial t} + \rho\overline{V}\overline{\nabla}u$$

<u>in x direction:</u>

$$\boxed{dx\,dy\,dz\overbrace{\left(\frac{\partial(\rho u)}{\partial t} + \frac{\partial\left(\rho u^2\right)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} + \frac{\partial(\rho uw)}{\partial z}\right)}^{\rho\frac{du}{dt}} = \overbrace{\left(-\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z}\right)dx\,dy\,dz}^{\sum_{i=1}^{8} F_{x,i}}}$$

Flow Modelling by Means of Continuum Mechanics; Navier-Stokes Equations

Source: VKI-LS, Introduction to CFD



$$\rho \frac{du}{dt} = \rho \frac{\partial u}{\partial t} + \rho \overline{V}\,\overline{\nabla} u$$

<u>As:</u>

$$\rho \frac{\partial u}{\partial t} = \frac{\partial(\rho u)}{\partial t} - u \frac{\partial \rho}{\partial t}$$

<u>and:</u>

$$\rho \overline{V}\,\overline{\nabla} u = \overline{\nabla}\left(\rho u \overline{V}\right) - u \overline{\nabla}\left(\rho \overline{V}\right)$$

<u>Hence:</u>

$$\rho \frac{du}{dt} = \frac{\partial(\rho u)}{\partial t} - u \overbrace{\left[\frac{\partial \rho}{\partial t} + \overline{\nabla}\left(\rho \overline{V}\right)\right]}^{0 \text{ (from mass cons. law)}} + \overline{\nabla}\left(\rho u \overline{V}\right)$$

$$\boxed{ma_x = m \frac{du}{dt} = \sum_{i=1}^{8} F_{x,i}}$$

<u>in x direction:</u>

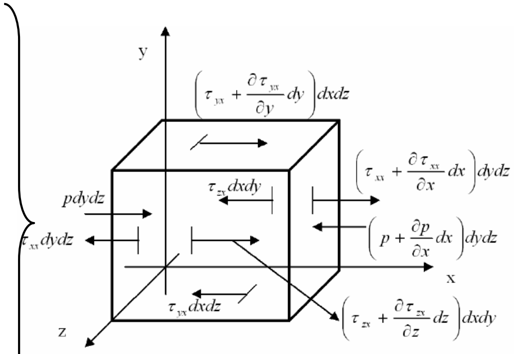
$$\boxed{\overbrace{dx\,dy\,dz\left(\frac{\partial(\rho u)}{\partial t} + \frac{\partial\left(\rho u^2\right)}{\partial x} + \frac{\partial(\rho u v)}{\partial y} + \frac{\partial(\rho u w)}{\partial z}\right)}^{\rho \frac{du}{dt}} = \overbrace{\left(-\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z}\right) dx\,dy\,dz}^{\sum_{i=1}^{8} F_{x,i}}}$$

Flow Modelling by Means of Continuum Mechanics; Navier-Stokes Equations

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \longrightarrow \frac{\partial(\rho u)}{\partial x}=0 \xrightarrow[V]{\iiint} \iint_A \rho u dA=\rho u A=Constant$$

$$\boxed{F=ma}$$

$$\left.\begin{aligned}
&\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho u^2+p)}{\partial x}+\frac{\partial(\rho uv)}{\partial y}+\frac{\partial(\rho uw)}{\partial z}=\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{yx}}{\partial y}+\frac{\partial \tau_{zx}}{\partial z}\\
&\frac{\partial(\rho v)}{\partial t}+\frac{\partial(\rho vu)}{\partial x}+\frac{\partial(\rho v^2+p)}{\partial y}+\frac{\partial(\rho vw)}{\partial z}=\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{zy}}{\partial z}\\
&\frac{\partial(\rho w)}{\partial t}+\frac{\partial(\rho wu)}{\partial x}+\frac{\partial(\rho wv)}{\partial y}+\frac{\partial(\rho w^2+p)}{\partial z}=\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z}
\end{aligned}\right\}$$



$$\boxed{P=Fv}$$

(T)

$$\frac{\partial(\rho E)}{\partial t}+\frac{\partial(\rho uH)}{\partial x}+\frac{\partial(\rho vH)}{\partial y}+\frac{\partial(\rho wH)}{\partial z}=\frac{\partial\left(u\tau_{xx}+v\tau_{xy}+w\tau_{xz}+k\,\partial T/\partial x\right)}{\partial x}+$$

$$+\frac{\partial\left(u\tau_{yx}+v\tau_{yy}+w\tau_{yz}+k\,\partial T/\partial y\right)}{\partial y}+\frac{\partial\left(u\tau_{zx}+v\tau_{zy}+w\tau_{zz}+k\,\partial T/\partial z\right)}{\partial z}$$

$$E=c_v T+\frac{|\vec{V}|^2}{2}$$

$$H=c_p T+\frac{|\vec{V}|^2}{2}$$

$$p=\rho R T$$

Flow Modelling by Means of Continuum Mechanics; Navier-Stokes Equations

The conditions in physical point of view for the application of the nonlinear partial differential system of equations introduced in the previous slide:

- Compressible, ideal gas in a relative stationary system,
- Newtonian continuum fluid, which can be either laminar or turbulent,
- Homogeneous (one material), isotropic material,
- Taking account of transient processes,
- Flow, in which the friction is included between fluid layers sliding on each other,
- Free from any force field (e.g.: gravity and electromagnetic field),
- Free from any sources and sink,
- Conservation form → discontinuities (<u>contact discontinuities</u>, <u>slip lines</u> and <u>shock waves</u>) are handled.

$$v_{n,1} - v_{n,2} = [v_n] = 0; [p] = 0; \quad [\rho] \neq 0; [v_t] = 0;$$

$$[v_n] = 0; [p] = 0; \quad [\rho] \neq 0; [v_t] \neq 0;$$

$$[v_n] \neq 0; [p] \neq 0; \quad [\rho] \neq 0; [v_t] = 0;$$

CFD

Computational Fluid Dynamics

What is the CFD (Computational Fluid Dynamics)?

- The CFD stands for Computational Fluid Dynamics, which is a way of modelling flows with the help of a computer using principles found in math and physics.
- It is one of the most important key elements of modern development processes today.
- With the help of CFD one can develop more cost effective, more environmental friendly and safer vehicles, products and processes with higher performances and higher efficiencies.
- It can be effective tool for simulating processes, which are expensive or cannot be implemented.
- Beside verification and plausibility check, the validity of simulations can be checked by experiments or others (e.g.: benchmarks).

The Advantages of CFD

- The results of the simulations give back the results of the measurements within less than 5-10 % in the 80 % of the industrial applications.
- It can be used in the full life-time cycle of the products.
- Significant amount of cost, time and capacity can be saved by using CFD.
- The physical processes and so the root cause of the problems can be recognized more easily and faster due to the possibilities of wide visualization techniques (e.g.: parameter distributions, streamlines, velocity vectors in any arbitrary cross sections).
- More physics (e.g.: fluid dynamics, heat transfer, structural mechanics, electromagnetics) can be coupled together at reasonable computational costs.

The Advantages of CFD

- It can be used where the measurement is difficult, would influence the investigated process or it is impossible (e.g.: Mars Mission).
- The numerical simulation can be parameterized, easily reproduced and automated.
- The simulations can be combined with optimization algorithms.
- The whole development process cannot be based only on calculations, validation is required.

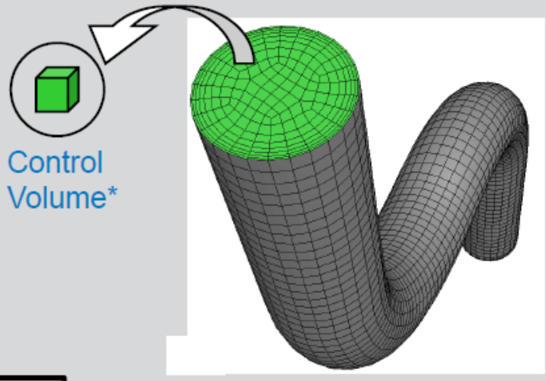
Main Application Area

- Vehicle industry (aerodynamics (internal, external), engine operation, air conditioning),
- Aerospace engineering,
- Safety (prediction of fire- and smoke-spreading, modelling of detonation and other hazard phenomena),
- Turbomachinery,
- Environmental protection,
- Production and operational process in heavy, light, chemistry and food industry
- Building industry (heating, cooling, air conditioning, drain and piped water)
- Weather forecast, climate models
- Astronomy

Summary of Basic Operation of CFD

How Does CFD Work?

- **ANSYS CFD solvers are based on the finite volume method**
  - Domain is discretized into a set of control volumes
  - General conservation (transport) equations for mass, momentum, energy, species, etc. are solved on this set of control volume



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

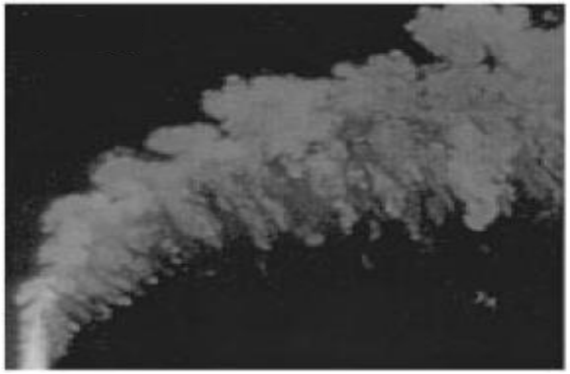
Equation	ϕ
Continuity	1
X momentum	*u*
Y momentum	*v*
Z momentum	*w*
Energy	*h*

  - Partial differential equations are discretized into a system of algebraic equations
  - All algebraic eqations are then solved numerically to render the solution field

Forrás: Introduction to ANSYS CFX, Lecture 02 – Introduction to CFD, CFX-Intro_14.0_L02_IntroCFD_CFX.pdf (2013.09.01.)

Mathematical Models of Flow - Turbulence

Flow Modelling by Means of Continuum Mechanics; Navier-Stokes Equations; Turbulent Flows



Flow Modelling by Means of Continuum Mechanics; Navier-Stokes Equations; Turbulent Flows

- **The Reynolds number is the criterion used to determine whether the flow is laminar or turbulent**

$$\mathrm{Re}_L = \frac{\rho . U . L}{\mu}$$

- The Reynolds number is based on the length scale of the flow:

  $L = x,\ d,\ d_{hyd},\ \text{etc.}$

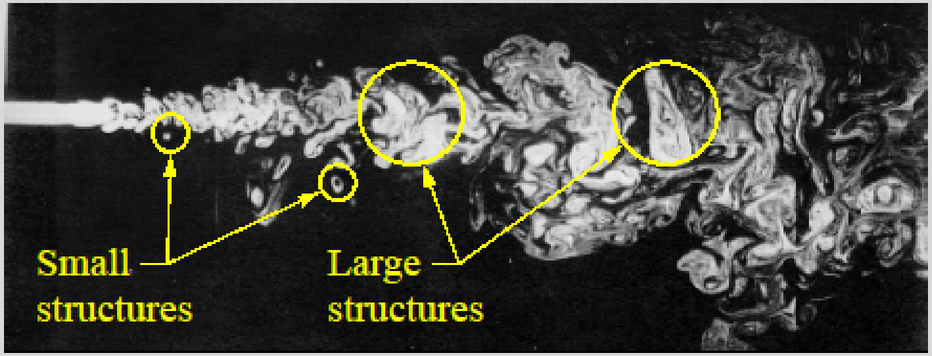
- Transition to Turbulence varies depending on the type of flow:
  - External flow
    - along a surface : $\mathrm{Re}_x > 500\,000$
    - around on obstacle : $\mathrm{Re}_L > 20\,000$
  - Internal flow : $\mathrm{Re}_D > 2\,300$

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; Navier-Stokes Equations; Turbulent Flows -Theory

Turbulent Flow Structures

- **A Turbulent Flow contains a wide range of turbulent eddy sizes Characteristics**
  - Unsteady, tridimensional, irregular, stochastic motion in which transported quantities (mass, momentum, scalar species) fluctuate in time and space
  - Unpredictability in detail
  - Large scale Coherent structures are different in each flow, whereas small eddies are more universal

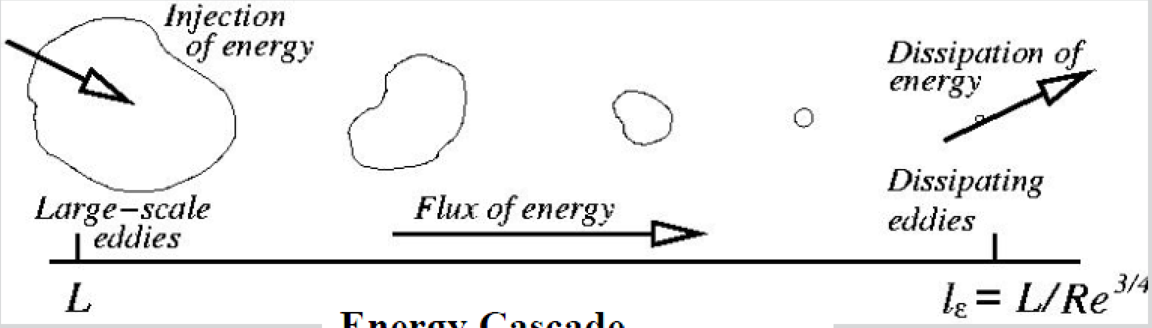


Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; Navier-Stokes Equations; Turbulent Flows -Theory

Turbulent Flow Structures

- **Energy is transferred from larger eddies to smaller eddies (Kolmogorov Cascade)**
  - Large scale contains most of the energy
  - In the smallest eddies, turbulent energy is converted to internal energy by viscous dissipation



Energy Cascade Richardson (1922), Kolmogorov (1941)

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; Navier-Stokes Equations; Turbulent Flows -Theory

Turbulent Flow Structures

- **Characteristics of the Turbulent Structures:**
  - Length scale : l [m]
  - Velocity scale : $\sqrt{k}$ [m/s]
  - Time scale : $\dfrac{l}{\sqrt{k}}$ [s]
  - Shape (non-isotropic larger structures)

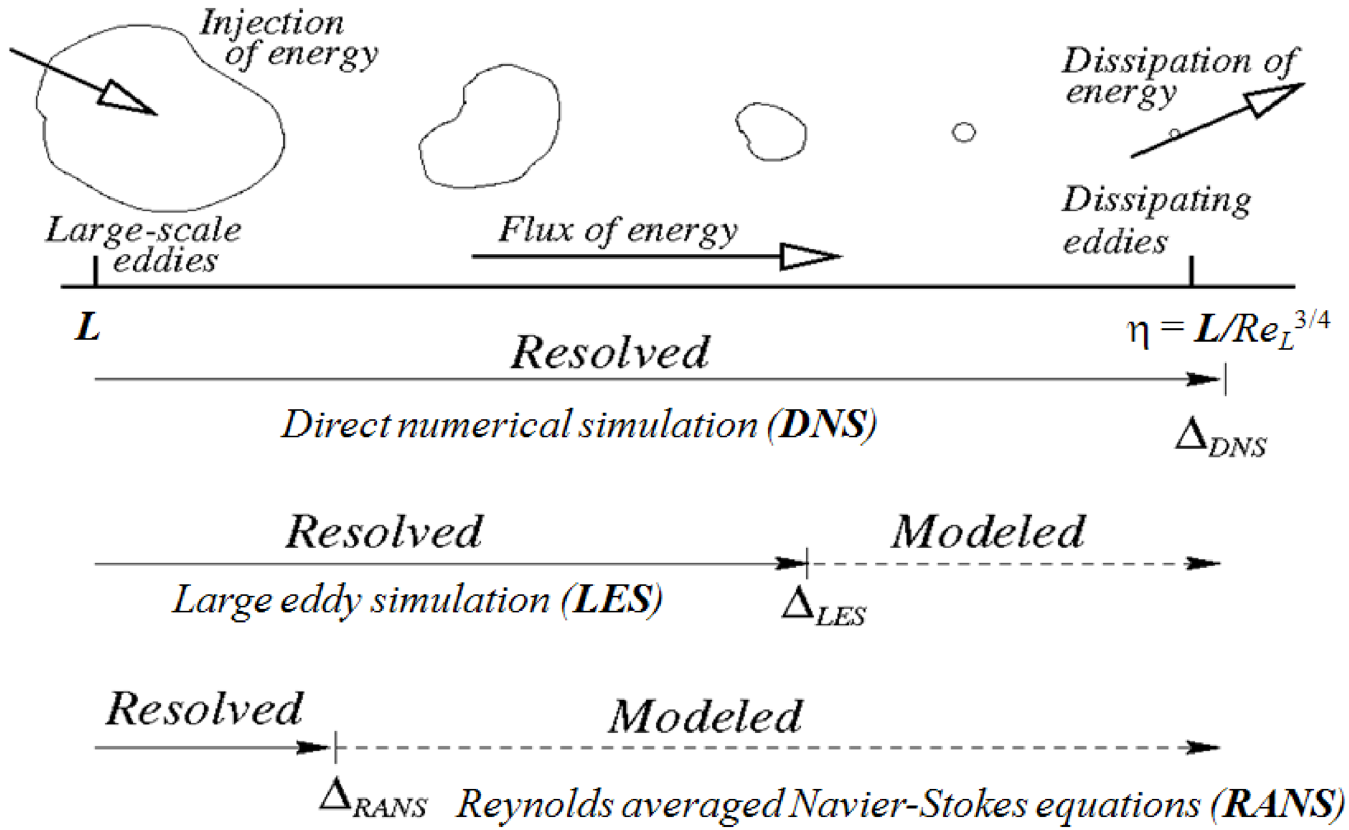
- **Turbulent kinetic energy :** $k = \dfrac{1}{2}\left(\overline{u'^2} + \overline{v'^2} + \overline{w'^2}\right)$ [m^2/s^2]
- **Turbulent kinetic energy dissipation :** ε [m^2/s^3] ~ $k^{3/2}/l$ (dimensional analysis)
- **Turbulent Reynolds :** $Re_t = k^{1/2} . l/\nu \sim k^2/\nu\varepsilon$ [-]
- **Turbulent Intensity :** $I = \dfrac{u'}{U} \approx \dfrac{1}{U}\sqrt{\dfrac{2k}{3}}$ [-]

$$\boxed{u_i(\mathbf{x},t) = \overline{U}_i(\mathbf{x},t) + u_i'(\mathbf{x},t)}$$

Instantaneous component — Time-average component — Fluctuating component

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; NS Equations; Simulation Techniques for Handling Turbulence



Forrás:
Jurij SODJA : Turbulence models in CFD
http://www-f1.ijs.si/~rudi/sola/Turbulence-models-in-CFD.pdf (2013.09.01)

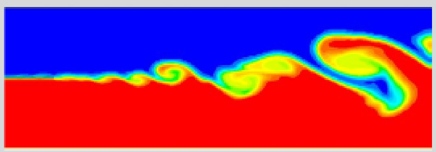
Flow Modelling by Means of Continuum Mechanics; NS Equations; Simulation Techniques for Handling Turbulence

Overview of Computational Approaches

- Different approaches to make turbulence computationally tractable

DNS

(Direct Numerical Simulation)

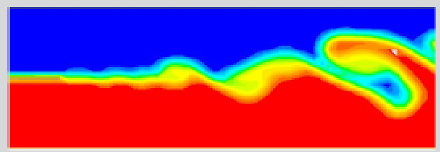


- Numerically solving the full unsteady Navier-Stokes equations
- Resolves the whole spectrum of scales
- No modeling is require
- But the cost is too prohibitive!

Not practical for industrial flows!

LES

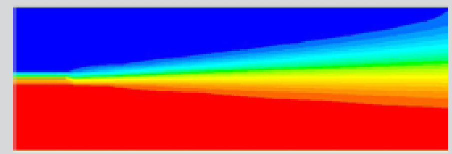
(Large Eddy Simulation)



- Solves the spatially averaged N-S equations
- Large eddies are directly resolved, but eddies smaller than the mesh are modeled
- Less expensive than DNS, but the amount of computational resources and efforts are still too large for most practical applications

RANS

(Reynolds Averaged Navier-Stokes Simulation)



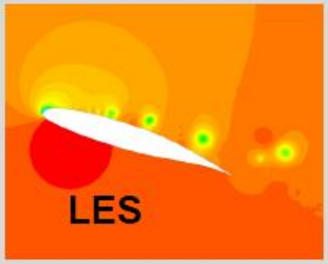
- Solve time-averaged Navier-Stokes equations
- All turbulent length scales are modeled in RANS
- Various different models are available
- This is the most widely used approach for calculating industrial flows

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; NS Equations; Simulation Techniques for Handling Turbulence

RANS Modeling : Justification

- **For most engineering applications it is unnecessary to resolve the details of the turbulent fluctuations**
- **We only need to know how turbulence affect the mean flow**
- **For a turbulence model to be useful it:**
  - must have wide applicability,
  - be accurate,
  - simple,
  - and economical to run,

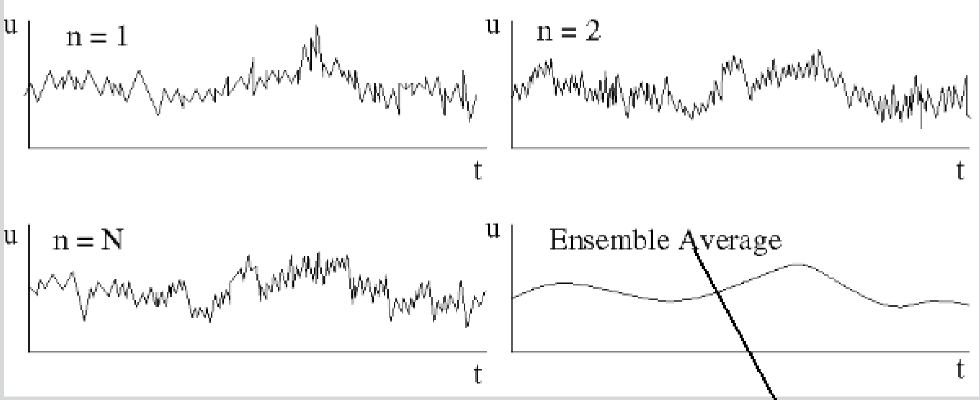


Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence -Theory

RANS Modeling : Justification

- **Fluid properties and velocity exhibit random variations**
  - Statistical averaging results in accountable, turbulence related transport mechanisms.
  - This characteristic allows for turbulence modeling



$$\overline{u_i}(\mathbf{x},t) = \lim_{N\to\infty} \frac{1}{N} \sum_{n=1}^{N} u_i^{(n)}(\mathbf{x},t)$$

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

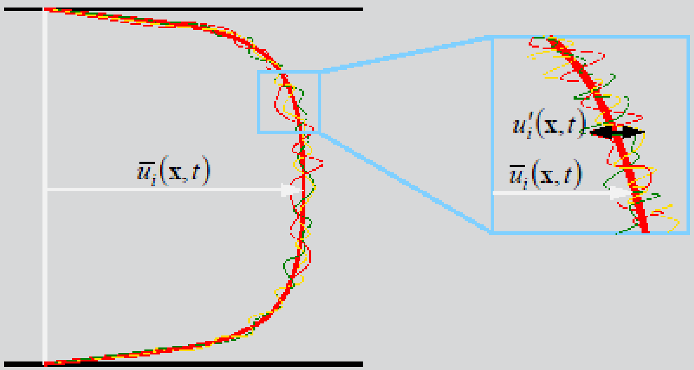
Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence -Theory

RANS Modeling : Averaging

- **Ensemble (time) averaging may be used to extract the mean flow properties from the instantaneous ones**
  - The instantaneous velocity, u_i, is split into average and fluctuating components

$$\boxed{u_i(\mathbf{x},t) = \overline{u}_i(\mathbf{x},t) + u_i'(\mathbf{x},t)}$$

Instantaneous component — Time-average component — Fluctuating component



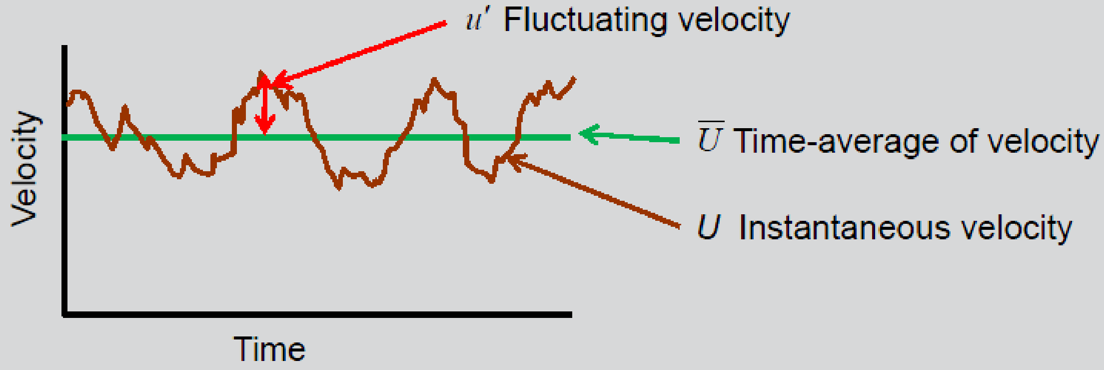
Example: Fully-Developed Turbulent Pipe Flow Velocity Profile

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence -Theory

Mean and Instantaneous Velocities

- If we recorded the velocity at a particular point in the real (turbulent) fluid flow, the instantaneous velocity (*U*) would look like this:



- At any point in time: $U = \overline{U} + u'$
- The time average of the fluctuating velocity u' must be zero: $\overline{u'} = 0$
- BUT, the RMS of u' is not necessarily zero: $\overline{u'^2} \neq 0$
- Note you will hear reference to the turbulence energy, *k*. This is the sum of the 3 fluctuating velocity components: $k = 0.5 * \left(\overline{u'^2} + \overline{v'^2} + \overline{w'^2} \right)$

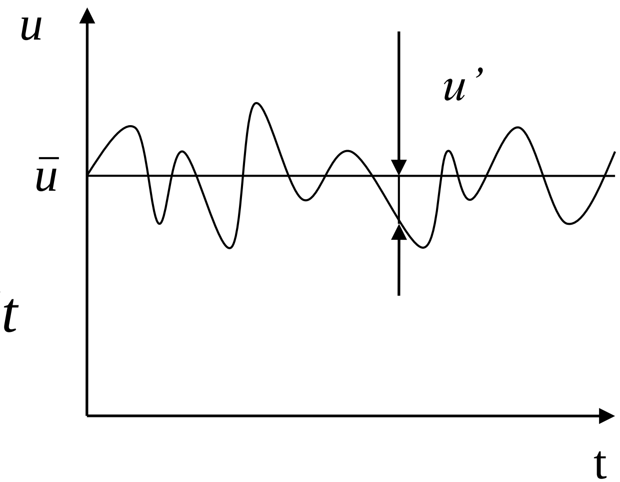
Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Reynolds and Favre Averaging -Theory - DASFLOW

$$u = \bar{u} + u' \quad v = \bar{v} + v'$$

$$\rho = \bar{\rho} + \rho'$$

$$w = \bar{w} + w' \quad p = \bar{p} + p'$$



$$\underbrace{\bar{u} = \frac{1}{\Delta t}\int_{t_0}^{t_0+\Delta t} u\,dt}_{\text{Reynolds Átlagolás}} \qquad \underbrace{\tilde{u} = \frac{1}{\bar{\rho}}\frac{1}{\Delta t}\int_{t_0}^{t_0+\Delta t} (\rho u)\,dt}_{\text{Favre Átlagolás}}$$

Reynolds Átlagolás ⇒ Favre Átlagolás

Nagy sebességű, összenyomható áramlás esetén.

$$u = \tilde{u} + u'' \quad v = \tilde{v} + v'' \quad w = \tilde{w} + w'' \quad p = \bar{p} + p' \quad \rho = \bar{\rho} + \rho'$$

$$h = \tilde{h} + h'' \quad e = \tilde{e} + e'' \quad T = \tilde{T} + T'' \quad q_j = \bar{q}_j + q_j' \quad \bar{q}_j = q_{L\ j}$$

L: laminar transport of the heat

Flow Modelling by Means of Continuum Mechanics; RANS; Reynolds and Favre Averaging -Theory - DASFLOW

$$\frac{\partial\overline{\rho}}{\partial t}+\frac{\partial\overline{\rho}\widetilde{u}}{\partial x}+\frac{\partial\overline{\rho}\widetilde{v}}{\partial y}+\frac{\partial\overline{\rho}\widetilde{w}}{\partial z}=0$$

$$\frac{\partial\overline{\rho}\widetilde{u}}{\partial t}+\frac{\partial\overline{\rho}\widetilde{u}\widetilde{u}}{\partial x}+\frac{\partial\overline{\rho}\widetilde{v}\widetilde{u}}{\partial y}+\frac{\partial\overline{\rho}\widetilde{w}\widetilde{u}}{\partial z}=-\frac{\partial\overline{p}}{\partial x}+\left[\frac{\partial\tau_{xx}^{F}}{\partial x}+\frac{\partial\tau_{xy}^{F}}{\partial y}+\frac{\partial\tau_{xz}^{F}}{\partial z}\right]$$

$$\frac{\partial\overline{\rho}\widetilde{v}}{\partial t}+\frac{\partial\overline{\rho}\widetilde{u}\widetilde{v}}{\partial x}+\frac{\partial\overline{\rho}\widetilde{v}\widetilde{v}}{\partial y}+\frac{\partial\overline{\rho}\widetilde{w}\widetilde{v}}{\partial z}=-\frac{\partial\overline{p}}{\partial y}+\left[\frac{\partial\tau_{yx}^{F}}{\partial x}+\frac{\partial\tau_{yy}^{F}}{\partial y}+\frac{\partial\tau_{yz}^{F}}{\partial z}\right]$$

$$\frac{\partial\overline{\rho}\widetilde{w}}{\partial t}+\frac{\partial\overline{\rho}\widetilde{u}\widetilde{w}}{\partial x}+\frac{\partial\overline{\rho}\widetilde{v}\widetilde{w}}{\partial y}+\frac{\partial\overline{\rho}\widetilde{w}\widetilde{w}}{\partial z}=-\frac{\partial\overline{p}}{\partial z}+\left[\frac{\partial\tau_{zx}^{F}}{\partial x}+\frac{\partial\tau_{zy}^{F}}{\partial y}+\frac{\partial\tau_{zz}^{F}}{\partial z}\right]$$

Flow Modelling by Means of Continuum Mechanics; RANS; Reynolds and Favre Averaging -Theory - DASFLOW

$$\tau_{xx}^F = 2\mu\frac{\partial\tilde{u}}{\partial x} - \frac{2}{3}\mu\nabla^T\tilde{\vec{V}} - \overline{\rho u''u''}$$

$$\tau_{yy}^F = 2\mu\frac{\partial\tilde{v}}{\partial y} - \frac{2}{3}\mu\nabla^T\tilde{\vec{V}} - \overline{\rho v''v''}$$

$$\tau_{zz}^F = 2\mu\frac{\partial\tilde{w}}{\partial z} - \frac{2}{3}\mu\nabla^T\tilde{\vec{V}} - \overline{\rho w''w''}$$

$$\tau_{xy}^F = \tau_{yx}^F = \mu\left(\frac{\partial\tilde{u}}{\partial y} + \frac{\partial\tilde{v}}{\partial x}\right) - \overline{\rho u''v''}$$

$$\tau_{xz}^F = \tau_{zx}^F = \mu\left(\frac{\partial\tilde{u}}{\partial z} + \frac{\partial\tilde{w}}{\partial x}\right) - \overline{\rho u''w''}$$

$$\tau_{yz}^F = \tau_{zy}^F = \mu\left(\frac{\partial\tilde{v}}{\partial z} + \frac{\partial\tilde{w}}{\partial y}\right) - \overline{\rho v''w''}$$

$$\frac{\partial}{\partial t}\left[\bar{\rho}\left(\tilde{e} + \frac{1}{2}\sum_{i=1}^{3}\tilde{u}_i\tilde{u}_i\right) + \frac{1}{2}\sum_{i=1}^{3}\overline{\rho u_i''u_i''}\right] + \sum_{j=1}^{3}\frac{\partial}{\partial x_j}\left[\bar{\rho}\tilde{u}_j\left(\tilde{h} + \frac{1}{2}\sum_{i=1}^{3}\tilde{u}_i\tilde{u}_i\right) + \tilde{u}_j\frac{1}{2}\sum_{i=1}^{3}\overline{\rho u_i''u_i''}\right] =$$

$$= \sum_{j=1}^{3}\frac{\partial}{\partial x_j}\left[-\bar{q}_j - \overline{\rho u_j''h''} + \sum_{i=1}^{3}\overline{\tau_{ji}u_i''} - \frac{1}{2}\sum_{i=1}^{3}\overline{\rho u_j''u_i''u_i''}\right] + \sum_{j=1}^{3}\frac{\partial}{\partial x_j}\left[\sum_{i=1}^{3}\left[\tilde{u}_i\left(\bar{\tau}_{ji} - \overline{\rho u_i''u_j''}\right)\right]\right]$$

Flow Modelling by Means of Continuum Mechanics; RANS; Reynolds and Favre Averaging -Theory - DASFLOW

$$\frac{1}{2}\sum_{i=1}^{3}\overline{\rho u_i'' u_i''} = \overline{\rho} k \tag{1.75}$$

where $k = \frac{1}{2}\left(\overline{u''^2} + \overline{v''^2} + \overline{w''^2}\right)$ is the turbulent kinetic energy per unit mass. The turbulent transport of heat is next:

$$q_{T\,j} = \overline{\rho u_j'' h''} \tag{1.76}$$

Another two terms on the RHS are given by:

- $\sum_{i=1}^{3}\overline{\tau_{ji} u_i''}$: molecular diffusion of turbulent kinetic energy
- $\frac{1}{2}\sum_{i=1}^{3}\overline{\rho u_j'' u_i'' u_i''}$: turbulent transport of turbulent kinetic energy

They represent the transfers between the mean energy and turbulent kinetic energy. The remaining terms are:

- $\overline{\tau}_{ji}$: the laminar part of the stress tensor (elements of $\underline{\underline{\pi}}_L^F$)
- $-\overline{\rho u_i'' u_j''}$: the Favre averaged Reynolds stresses

$$\boxed{\overline{p} = \overline{\rho} R \widetilde{T}}$$

Flow Modelling by Means of Continuum Mechanics; RANS; Reynolds and Favre Averaging -Theory - DASFLOW

This equation is used in the one- and two-equation turbulence models. On the LHS there are the unsteady and the convection terms. The terms on the RHS are:

– $\sum_{j=1}^{3}\sum_{i=1}^{3}\left[-\overline{\rho u_i'' u_j''}\frac{\partial \widetilde{u}_i}{\partial x_j}\right]$: production term, means the rate, at which the kinetic energy is transferred from the mean flow to the turbulence.

– $\sum_{j=1}^{3}\sum_{i=1}^{3}\left[\overline{\tau_{ji}\frac{\partial u_i''}{\partial x_j}}\right] = \overline{\rho}\varepsilon$: dilatation dissipation, the rate, at which the turbulent kinetic energy is converted into thermal energy. 'ε' is the dissipation per unit mass of the turbulent kinetic energy.

– The next two terms are the molecular diffusion and the turbulent transport of the turbulent kinetic energy like in the energy equation.

– $p' u_j''$: pressure diffusion term.

– $\sum_{i=1}^{3}\overline{u_i''}\frac{\partial \overline{p}}{\partial x_i}$: pressure work term.

– $\sum_{i=1}^{3}\overline{p'\frac{\partial u_i''}{\partial x_i}}$: pressure dilatation term.

Many of the relationships in these expressions are not known, only empirical approximations exist to describe them. Some of the approximations are described in the next chapters. It can be adequate for many types of flows of interest to extend (1.79) to be able to handle compressibility.

Flow Modelling by Means of Continuum Mechanics; RANS; Reynolds and Favre Averaging -Theory - DASFLOW

Boussinesq Approximation

Following Boussinesq approximation the Reynolds stresses can be related to the turbulent viscosity and they are modeled in a similar way to the stresses from the mean velocity gradients:

$$-\overline{\rho u_i'' u_j''} = \mu_t \left(\frac{\partial \widetilde{u}_i}{\partial x_j} + \frac{\partial \widetilde{u}_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \left(\mu_t \sum_{k=1}^{3} \frac{\partial \widetilde{u}_k}{\partial x_k} + \overline{\rho} k \right) \tag{1.80}$$

where (for 3 dimensions)

- $i, j, k = 1..3$
- $x_1 = x$, $x_2 = y$, $x_3 = z$, $u_1 = u$, $u_2 = v$, $u_3 = w$
- $\delta_{ij} = \begin{cases} 1 & if \quad i = j \\ 0 & if \quad i \neq j \end{cases}$ the Kronecker's delta
- μ_t is the turbulent or eddy-viscosity
- $k = \frac{1}{2}\left(\overline{u''^2} + \overline{v''^2} + \overline{w''^2} \right)$ is the turbulent kinetic energy per unit mass.

While molecular viscosity is a property of the fluid the turbulent viscosity tries to model viscous effect caused by turbulence. Turbulent viscosity is varying through the flow and has to be determined. The stresses in 3 dimensions after introducing turbulent viscosity are given by:

Flow Modelling by Means of Continuum Mechanics; RANS; Reynolds and Favre Averaging -Theory - DASFLOW

$$\tau^F_{xx} = 2(\mu+\mu_t)\frac{\partial\tilde{u}}{\partial x} - \frac{2}{3}(\mu+\mu_t)\nabla^T\tilde{\vec{V}} - \frac{2}{3}\overline{\rho}k = \frac{2}{3}\mu_{eff}\left(2\frac{\partial\tilde{u}}{\partial x} - \frac{\partial\tilde{v}}{\partial y} - \frac{\partial\tilde{w}}{\partial z}\right) - \frac{2}{3}\overline{\rho}k \tag{1.81}$$

$$\tau^F_{yy} = 2(\mu+\mu_t)\frac{\partial\tilde{v}}{\partial y} - \frac{2}{3}(\mu+\mu_t)\nabla^T\tilde{\vec{V}} - \frac{2}{3}\overline{\rho}k = \frac{2}{3}\mu_{eff}\left(-\frac{\partial\tilde{u}}{\partial x} + 2\frac{\partial\tilde{v}}{\partial y} - \frac{\partial\tilde{w}}{\partial z}\right) - \frac{2}{3}\overline{\rho}k \tag{1.82}$$

$$\tau^F_{zz} = 2(\mu+\mu_t)\frac{\partial\tilde{w}}{\partial z} - \frac{2}{3}(\mu+\mu_t)\nabla^T\tilde{\vec{V}} - \frac{2}{3}\overline{\rho}k = \frac{2}{3}\mu_{eff}\left(-\frac{\partial\tilde{u}}{\partial x} - \frac{\partial\tilde{v}}{\partial y} + 2\frac{\partial\tilde{w}}{\partial z}\right) - \frac{2}{3}\overline{\rho}k \tag{1.83}$$

$$\tau^F_{xy} = \tau^F_{yx} = \mu\left(\frac{\partial\tilde{u}}{\partial y} + \frac{\partial\tilde{v}}{\partial x}\right) + \mu_t\left(\frac{\partial\tilde{u}}{\partial y} + \frac{\partial\tilde{v}}{\partial x}\right) = \mu_{eff}\left(\frac{\partial\tilde{u}}{\partial y} + \frac{\partial\tilde{v}}{\partial x}\right) \tag{1.84}$$

$$\tau^F_{xz} = \tau^F_{zx} = \mu\left(\frac{\partial\tilde{u}}{\partial z} + \frac{\partial\tilde{w}}{\partial x}\right) + \mu_t\left(\frac{\partial\tilde{u}}{\partial z} + \frac{\partial\tilde{w}}{\partial x}\right) = \mu_{eff}\left(\frac{\partial\tilde{u}}{\partial z} + \frac{\partial\tilde{w}}{\partial x}\right) \tag{1.85}$$

$$\tau^F_{yz} = \tau^F_{zy} = \mu\left(\frac{\partial\tilde{v}}{\partial z} + \frac{\partial\tilde{w}}{\partial y}\right) + \mu_t\left(\frac{\partial\tilde{v}}{\partial z} + \frac{\partial\tilde{w}}{\partial y}\right) = \mu_{eff}\left(\frac{\partial\tilde{v}}{\partial z} + \frac{\partial\tilde{w}}{\partial y}\right) \tag{1.86}$$

Here k is the turbulent kinetic energy per unit mass and $\mu_{eff} = \mu + \mu_t$ is the effective viscosity. Boussinesq approximation is assumed in most of the algebraic, one-equation and two-equation turbulence models. For later use the turbulent kinetic energy term is excluded:

Flow Modelling by Means of Continuum Mechanics; RANS; Reynolds and Favre Averaging -Theory - DASFLOW

$$\tau_{ij}^{F} = \begin{cases} 2\mu_{eff}\dfrac{\partial \tilde{u}_i}{\partial x_i} - \dfrac{2}{3}\mu_{eff}\nabla^{T}\tilde{\vec{V}} & if \quad i = j \\ \mu_{eff}\left(\dfrac{\partial \tilde{u}_i}{\partial x_j} + \dfrac{\partial \tilde{u}_j}{\partial x_i}\right) & if \quad i \neq j \end{cases} \tag{1.87}$$

Heat Flux Vector Terms

Changes in the heat flux vector due to averaging process can be expressed by turbulent viscosity.

$$q_{T\,j} = \overline{\rho u_j'' h''} = -\frac{\mu_t c_p}{\mathrm{Pr}_t}\frac{\partial \tilde{T}}{\partial x_j} \tag{1.88}$$

The advantage of this form is that it is similar to the laminar one. The coefficient of thermal conductivity can be written as:

$$k = c_p\left(\frac{\mu}{\mathrm{Pr}} + \frac{\mu_t}{\mathrm{Pr}_t}\right) \tag{1.89}$$

where Pr_t is the turbulent Prandtl number (Prandtl numbers are known or approximated for a given flow, often $\mathrm{Pr} = 0.72$ and $\mathrm{Pr}_t = 0.9$ for air). The heat flux vector comes from the expression

$$\vec{q} = \vec{q}_L + \vec{q}_T = -k\nabla\tilde{T} \tag{1.90}$$

Flow Modelling by Means of Continuum Mechanics; RANS; Reynolds and Favre Averaging -Theory - DASFLOW

Molecular Diffusion and Turbulent Transport Terms

These terms are often handled together as

$$\overline{\tau_{ij} u_i''} - \frac{1}{2}\overline{\rho u_j'' u_i'' u_i''} = \left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j} \qquad (1.91)$$

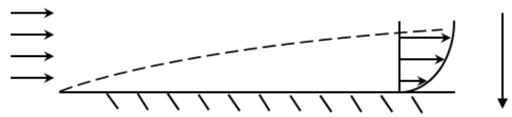
where ' σ_k ' is a scaling factor and ' k ' is the turbulent kinetic energy per unit mass.

Now all the changes resulting of averaging are described.

Flow Modelling by Means of Continuum Mechanics; RANS; Turbulence Modelling -Theory

Introduction to Turbulence Modelling

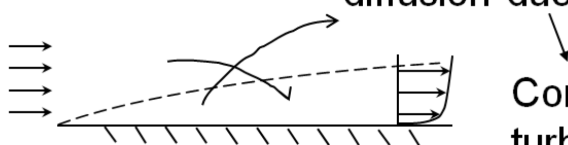
Laminar Flow



Momentum diffusion by viscosity

Turbulent Flow

Additional momentum diffusion due to turbulence



Concept of turbulent (or eddy) viscosity, μ_t

- μ_t is not a fluid property, but depends on level of turbulence in flow
- concept leads to mathematical models to deal with turbulence; each model is an approximation to what is really happening

Forrás: www.tech.plym.ac.uk/sme/dsgn313/CFDNotes06.ppt (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Turbulence Modelling -Theory

RANS Modeling : The Closure Problem

- The Reynolds Stress tensor $R_{ij} = -\rho\overline{u_i' u_j'}$ must be solved
- The RANS models can be closed in two ways:

Reynolds-Stress Models (RSM)

- R_{ij} is directly solved via transport equations (modeling is still required for many terms in the transport equations)

$$\frac{\partial}{\partial t}\left(\rho\overline{u_i' u_j'}\right)+\frac{\partial}{\partial x_k}\left(\rho\,\overline{u}_k\,\overline{u_i' u_j'}\right)=P_{ij}+F_{ij}+D_{ij}^T+\Phi_{ij}-\varepsilon_{ij}$$

- RSM is more advantageous in complex 3D turbulent flows with large streamline curvature and swirl,
- but the model is more complex, computationally intensive, more difficult to converge than eddy viscosity models

Eddy Viscosity Models

- Boussinesq hypothesis
  ➔Reynolds stresses are modeled using an eddy (or turbulent) viscosity, μ_T

$$R_{ij}=-\rho\overline{u_i' u_j'}=\mu_T\left(\frac{\partial\overline{u}_i}{\partial x_j}+\frac{\partial\overline{u}_j}{\partial x_i}\right)-\frac{2}{3}\mu_T\frac{\partial\overline{u}_k}{\partial x_k}\delta_{ij}-\frac{2}{3}\rho k\delta_{ij}$$

- The hypothesis is reasonable for simple turbulent shear flows: boundary layers, round jets, mixing layers, channel flows, etc.

- Note: All turbulence models contain empiricism
  - Equations cannot be derived from fundamental principles
  - Some calibrating to observed solutions and “intelligent guessing” is contained in the models

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Turbulence Modelling -Theory

$$\bar{\rho}\frac{\partial k}{\partial t}+\bar{\rho}\tilde{u}_j\frac{\partial k}{\partial x_j}=\overbrace{-\overline{\rho u_i'' u_j''}\frac{\partial \tilde{u}_i}{\partial x_j}-\beta^*\bar{\rho}k\omega}^{S_k}+\frac{\partial}{\partial x_j}\left[\left(\mu+\sigma^*\mu_t\right)\frac{\partial k}{\partial x_j}\right]$$

$$\bar{\rho}\frac{\partial \omega}{\partial t}+\bar{\rho}\tilde{u}_j\frac{\partial \omega}{\partial x_j}=\overbrace{-\overline{\rho u_i'' u_j''}\alpha\frac{\omega}{k}\frac{\partial \tilde{u}_i}{\partial x_j}-\beta\bar{\rho}\omega^2}^{S_\omega}+\frac{\partial}{\partial x_j}\left[\left(\mu+\sigma\mu_t\right)\frac{\partial \omega}{\partial x_j}\right]$$

$$\mu_t=\bar{\rho}\frac{k}{\omega}\quad \alpha=\frac{13}{25}\quad \sigma^*=\frac{1}{2}\quad \sigma=\frac{1}{2}\quad \beta^*=\beta_0^* f_{\beta^*}\left[1+\xi^* F\left(M_t\right)\right]\quad \beta_0=\frac{9}{125}\quad \beta_0^*=\frac{9}{100}$$

$$\beta=\beta_0 f_\beta-\beta_0^* f_{\beta^*}\xi^* F\left(M_t\right)\quad f_{\beta^*}=\begin{cases}1 & if\quad \chi_k\le 0\\ \dfrac{1+680\chi_k^2}{1+400\chi_k^2} & if\quad \chi_k>0\end{cases}\quad \chi_k=\frac{1}{\omega^3}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j}$$

$$\chi_\omega=\left|\frac{\Omega_{ij}\Omega_{jk}S_{ki}}{\left(\beta_0^*\omega\right)^3}\right|\quad f_\beta=\frac{1+70\chi_\omega}{1+80\chi_\omega}\quad \xi^*=\frac{3}{2}\quad \chi_\omega=0\quad M_{t0}=\frac{1}{4}\quad \varepsilon=\beta^*\omega k\quad l=\frac{k^{1/2}}{\omega}$$

$$F\left(M_t\right)=\left[M_t^2-M_{t0}^2\right]\mathrm{H}\left(M_t-M_{t0}\right)$$

$$\mathrm{H}(x)=\begin{cases}0 & if\quad x\le 0\\ 1 & if\quad x>0\end{cases}\quad S_{ij}=\frac{1}{2}\left(\frac{\partial \tilde{u}_i}{\partial x_j}+\frac{\partial \tilde{u}_j}{\partial x_i}\right)$$

$$M_t^2=\frac{2k}{a^2}\quad \Omega_{ij}=\frac{1}{2}\left(\frac{\partial \tilde{u}_i}{\partial x_j}-\frac{\partial \tilde{u}_j}{\partial x_i}\right)$$

Flow Modelling by Means of Continuum Mechanics; RANS; Turbulence Modelling - DASFLOW - Equations in Conservative Form

$$\frac{\partial}{\partial t}\iint_A U dA + \oint_\Gamma \left[H_n(U)\right] d\Gamma = \oint_\Gamma \left[H_{vn}(U)\right] d\Gamma + \iint_A \left[S(U)\right] dA$$

$$U = \begin{pmatrix} \bar{\rho} \\ \bar{\rho}\tilde{u} \\ \bar{\rho}\tilde{v} \\ \bar{\rho}\tilde{E} \\ \bar{\rho}k \\ \bar{\rho}\omega \end{pmatrix} \qquad H_n(U) = \begin{pmatrix} \bar{\rho}V_n \\ \bar{\rho}\tilde{u}V_n + p^* n_x \\ \bar{\rho}\tilde{v}V_n + p^* n_y \\ \left(\bar{\rho}\tilde{E} + p^*\right)V_n \\ \bar{\rho}V_n k \\ \bar{\rho}V_n \omega \end{pmatrix} \qquad S(U) = \begin{pmatrix} 0 \\ 0 \\ 0 \\ 0 \\ S_k \\ S_\omega \end{pmatrix}$$

Flow Modelling by Means of Continuum Mechanics; RANS; Turbulence Modelling - DASFLOW - Equations in Conservative Form

$$\frac{\partial}{\partial t}\iint U dA + \oint\left[H_n(U)\right]d\Gamma = \oint\left[H_{vn}(U)\right]d\Gamma + \iint\left[S(U)\right]dA$$

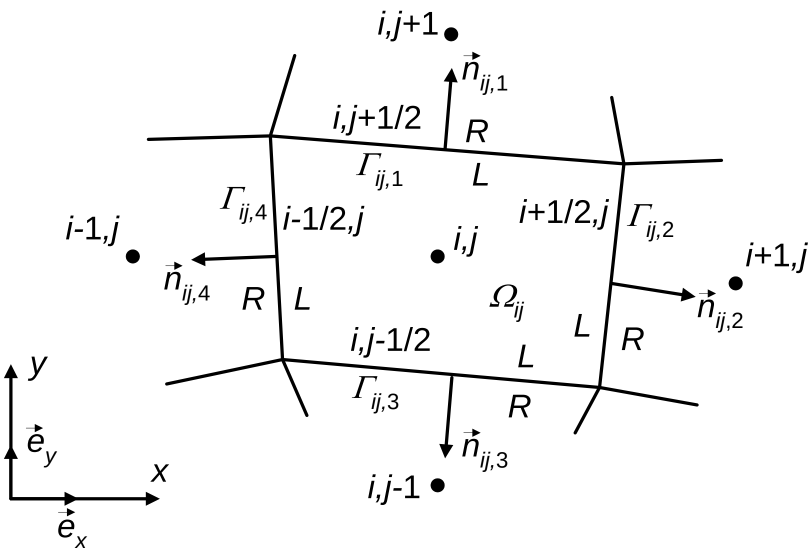
$$H_{vn}(U) = \vec{H}_v(U)\cdot\vec{n} = \begin{pmatrix} 0 \\ \tau_{xx}^{Fk}n_x + \tau_{yx}^{Fk}n_y + \tau_{zx}^{Fk}n_z \\ \tau_{xy}^{Fk}n_x + \tau_{yy}^{Fk}n_y + \tau_{zy}^{Fk}n_z \\ \tau_{xz}^{Fk}n_x + \tau_{yz}^{Fk}n_y + \tau_{zz}^{Fk}n_z \\ \sum_{i=1}^{3}\left[\left(\sum_{j=1}^{3}\tilde{u}_j\tau_{ij}^{Fk} - q_i\right)n_i\right] + \left(\mu + \sigma^*\mu_t\right)\sum_{j=1}^{3}\left[\frac{\partial k}{\partial x_j}n_j\right] \\ \left(\mu + \sigma^*\mu_t\right)\left[\frac{\partial k}{\partial x}n_x + \frac{\partial k}{\partial y}n_y + \frac{\partial k}{\partial z}n_z\right] \\ \left(\mu + \sigma\mu_t\right)\left[\frac{\partial \omega}{\partial x}n_x + \frac{\partial \omega}{\partial y}n_y + \frac{\partial \omega}{\partial z}n_z\right] \end{pmatrix}$$

Flow Modelling by Means of Continuum Mechanics; RANS; Turbulence Modelling - DASFLOW - Discretization

$$\oint_{\Gamma_{ij}} \left[H_{vn}(U)\right] d\Gamma = \sum_{k=1}^{4} \left(\left[H_{vn}\right]_{ij,k} \Gamma_{ij,k} \right)$$

$$H_{vn} = \frac{1}{2}\left[H_{vn}\left(U^{L}\right) + H_{vn}\left(U^{R}\right)\right]$$

$$\iint_{A_{ij}} \left[S(U)\right] dA = \left[S(U)\right]_{ij} A_{ij}$$



$$\frac{\partial}{\partial t} U_{ij} = -\frac{1}{A_{ij}} \left(\sum_{k=1}^{4} \left(\left[H_{n}\right]_{ij,k} \Gamma_{ij,k} \right) - \sum_{k=1}^{4} \left(\left[H_{vn}\right]_{ij,k} \Gamma_{ij,k} \right) \right) + \left[S(U)\right]_{ij} = \Re$$

$$\left.\begin{aligned} U^{0} &= U^{n} \\ U^{k} &= U^{0} + \alpha_k \Delta t \Re\left(U^{k-1}\right) \qquad k = 1,m \\ U^{n+1} &= U^{m} \end{aligned}\right\}$$

- Runge-Kutta módszer

  $m=4$

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence -Theory

RANS Modeling : Averaging

- **Thus, the instantaneous Navier-Stokes momentum equations may be re-write as Reynolds-averaged equations, as follow :**

$$\rho\left(\frac{\partial \overline{u}_i}{\partial t}+\overline{u}_k\frac{\partial \overline{u}_i}{\partial x_k}\right)=-\frac{\partial \overline{p}}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\mu\frac{\partial \overline{u}_i}{\partial x_j}\right)+\frac{\partial R_{ij}}{\partial x_j} \qquad R_{ij}=-\rho\overline{u_i'u_j'}$$

(Reynolds stress tensor)

  – The **Reynolds stresses** are additional unknowns introduced by the averaging procedure, hence they **must be modeled** (related to the averaged flow quantities) in order to close the system of governing equations

$$R_{ij}=-\rho\overline{u_i'u_j'}=\begin{pmatrix} -\rho\overline{u'^2} & -\rho\overline{u'v'} & -\rho\overline{u'w'} \\ -\rho\overline{u'v'} & -\rho\overline{v'^2} & -\rho\overline{v'w'} \\ -\rho\overline{u'w'} & -\rho\overline{v'w'} & -\rho\overline{w'^2} \end{pmatrix}$$

➔ 6 unknows …

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence -Theory

RANS Modeling : The Closure Problem

- The Reynolds Stress tensor $R_{ij} = -\rho \overline{u_i' u_j'}$ must be solved
- The RANS models can be closed in two ways:

Reynolds-Stress Models (RSM)

- R_{ij} is directly solved via transport equations (modeling is still required for many terms in the transport equations)

$$\frac{\partial}{\partial t}\left(\rho \overline{u_i' u_j'}\right) + \frac{\partial}{\partial x_k}\left(\rho \overline{u}_k \overline{u_i' u_j'}\right) = P_{ij} + F_{ij} + D_{ij}^T + \Phi_{ij} - \varepsilon_{ij}$$

- RSM is more advantageous in complex 3D turbulent flows with large streamline curvature and swirl,
- but the model is more complex, computationally intensive, more difficult to converge than eddy viscosity models

Eddy Viscosity Models

- Boussinesq hypothesis
  ➔Reynolds stresses are modeled using an eddy (or turbulent) viscosity, μ_T

$$R_{ij} = -\rho \overline{u_i' u_j'} = \mu_T \left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right) - \frac{2}{3}\mu_T \frac{\partial \overline{u}_k}{\partial x_k}\delta_{ij} - \frac{2}{3}\rho k \delta_{ij}$$

- The hypothesis is reasonable for simple turbulent shear flows: boundary layers, round jets, mixing layers, channel flows, etc.

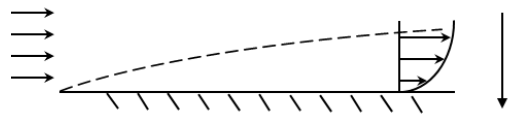
- Note: All turbulence models contain empiricism
  - Equations cannot be derived from fundamental principles
  - Some calibrating to observed solutions and "intelligent guessing" is contained in the models

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence -Theory

Introduction to Turbulence Modelling

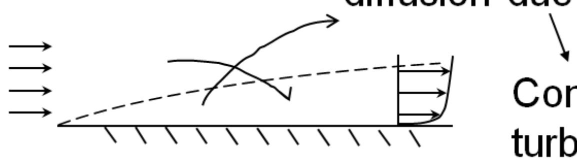
Laminar Flow



Momentum diffusion by viscosity

Turbulent Flow

Additional momentum diffusion due to turbulence



Concept of turbulent (or eddy) viscosity, μ_t

- μ_t is not a fluid property, but depends on level of turbulence in flow
- concept leads to mathematical models to deal with turbulence; each model is an approximation to what is really happening

Forrás: www.tech.plym.ac.uk/sme/dsgn313/CFDNotes06.ppt (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Models

CFX available Turbulence Models

- A large number of turbulence models are available in CFX, some have very specific applications while others can be applied to a wider class of flows with a reasonable degree of confidence

RANS Eddy-viscosity Models:

1) Zero Equation model.
2) Standard k-ε model.
3) RNG k-ε model.
4) Standard k-ω model.
5) Baseline (BSL) zonal k-ω based model.
6) SST zonal k-ω based model.
7) $(k\text{-}\varepsilon)_{1E}$ model.

RANS Reynolds-Stress Models:

1) LRR Reynolds Stress
2) QI Reynolds Stress
3) Speziale, Sarkar and Gatski Reynolds Stress
4) SMC-ω model
5) Baseline (BSL) Reynolds' Stress model

Eddy Simulation Models:

1) Large Eddy Simulation (LES) [transient]
2) Detached Eddy Simulation (DES)* [transient]
3) Scale Adaptive Simulation SST (SAS)* [transient]

* Advanced Turbulence Module is required

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Models

k-omega Model

- A pragmatic compromise for RANS Turbulence modeling:

  ➔ *k*-ω equations based models

- This models have gained popularity mainly because:

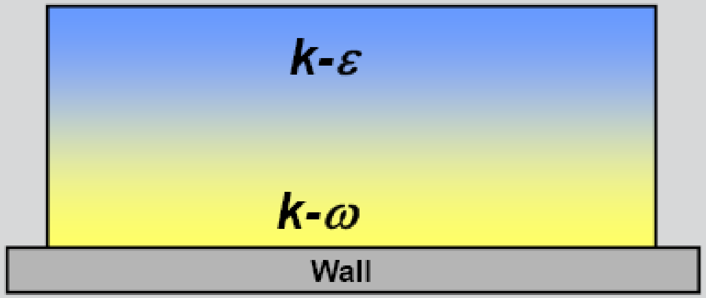
- *k*-ω models perform much better than *k*-ε models for boundary layer flows
  - For separation, transition, low Re effects, impingement, the k-ω models is more accurate than the k-ε models
- Accurate and robust for a wide range of boundary layer flows with pressure gradient
- Several sub-models/options: compressibility effects, transitional flows and shear-flow corrections

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Models

SST Model

- **Shear Stress Transport (SST) Model**
  - The SST model is an hybrid two-equation model that combines the advantages of both k-ε and k-ω models
  - k-ω model performs much better than k-ε models for boundary layer flows
  - Wilcox' original k-ω model is overly sensitive to the freestream value (BC) of ω, while k-ε model is not prone to such problem



  - The k-e and k-w models are blended such that the SST model functions like the k-w close to the wall and the k-e model in the freestream

SST is a good compromise between k-ε and k-ω models

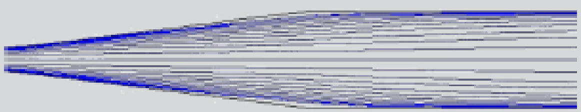
Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Models

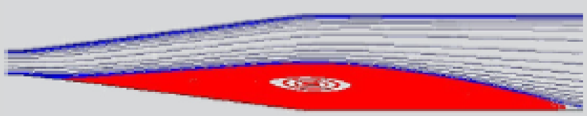
SST Model

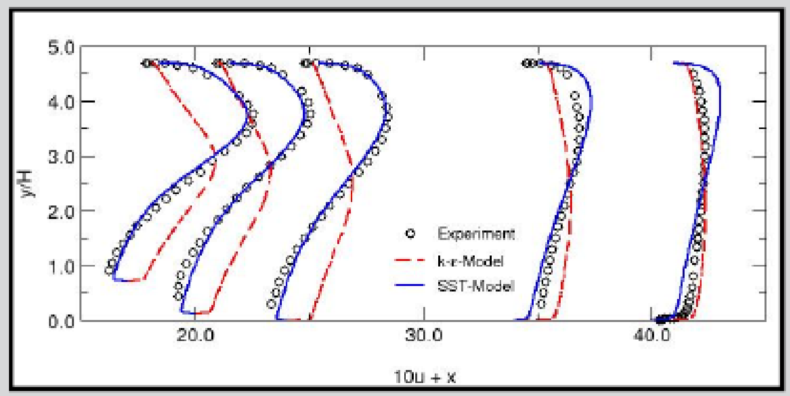
- It accounts for the transport of the turbulent shear stress and gives highly accurate predictions of the onset and the amount of flow separation

Standard k-ε fails to predict separation



SST result and experiment





Experiment Gersten et al.

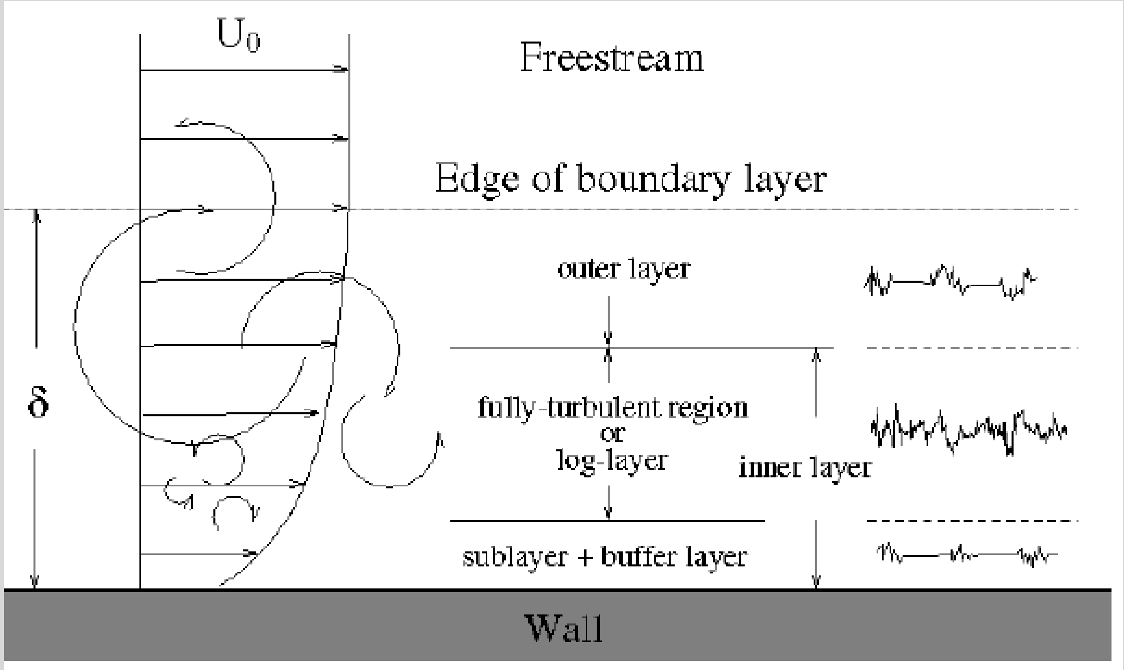
SST is a good compromise

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Handling Flow at Solid Walls

Turbulence Near the Wall

- The Structure of Near-Wall Flows



Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

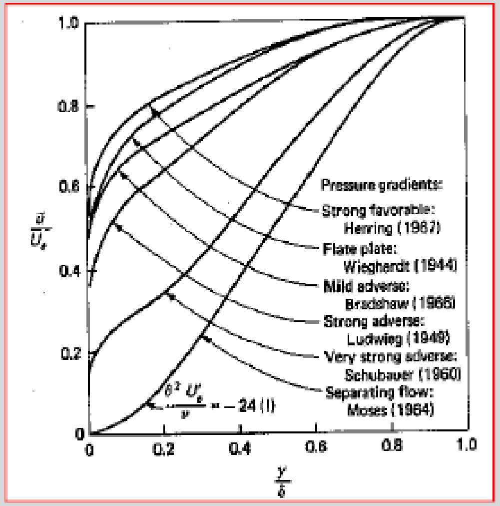
Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Handling Flow at Solid Walls

Turbulence Near the Wall

- Walls are main source of vorticity and turbulence
- The velocity profile near the wall is important:
  - Pressure Drop
  - Separation
  - Shear Effects
  - Recirculation

Accurate near-wall modeling is important for most engineering applications

• Turbulence models are generally suited to model the flow outside the boundary layer but need special treatments near the walls



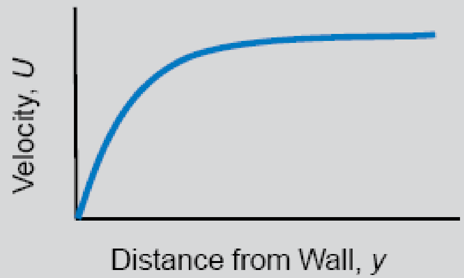
The above graph shows non-dimensional velocity versus non-dimensional distance from the wall. Different flows show different boundary layer profiles

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Handling Flow at Solid Walls

Turbulence near a Wall

- **Near to a wall, the velocity changes rapidly**



- **If we plot the same graph again, where**
  - Log scale axes are used
  - The velocity is made dimensionless, from U/Ut ($u_\tau = \sqrt{\frac{\tau_{wall}}{\rho}}$)
  - The wall distance vector is made dimensionless $y^+ = \frac{y u_\tau}{\nu}$
- **Then we arrive at the graph on the next page. The shape of this is generally the same for all flows:**

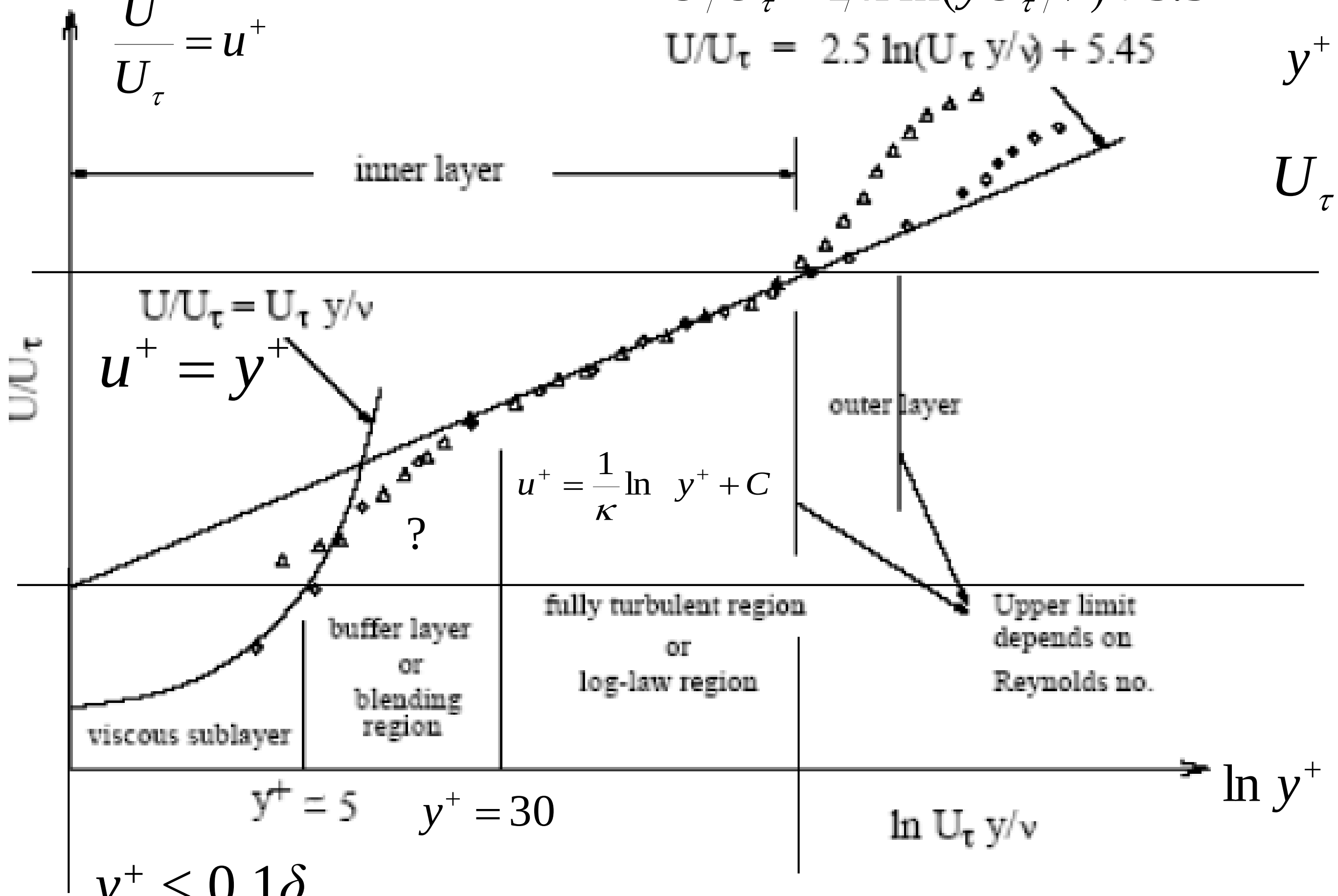
Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Handling Flow at Solid Walls

Law of the wall by Prandtl

$$U/U_\tau = 1/\kappa \ln\left(yU_\tau/\nu\right) + 5.5$$

$$\frac{U}{U_\tau} = u^+$$



$$u^+ = y^+$$

$$u^+ = \frac{1}{\kappa}\ln\ y^+ + C$$

$$y^+ = 30$$

$$y^+ < 0.1\delta$$

$$y^+ = \frac{\rho U_\tau y}{\mu}$$

$$U_\tau = \sqrt{\tau_w/\rho}$$

$$\tau_w = \mu\frac{dU}{dy}$$

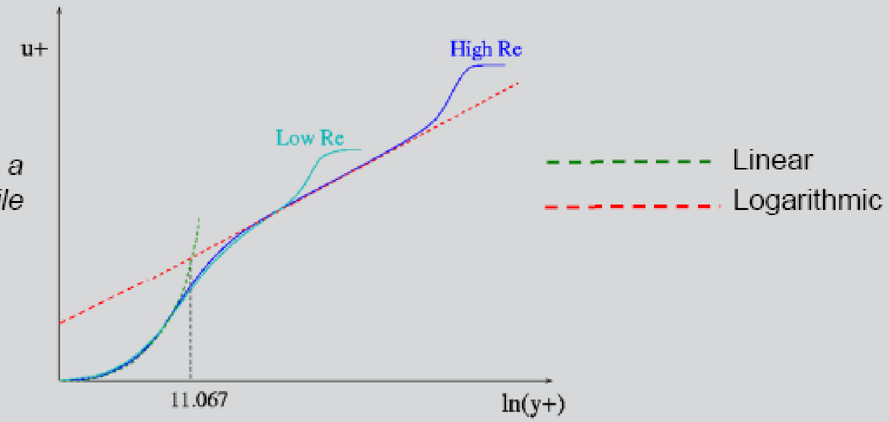
$$\ln y^+ = \ln\frac{U_\tau y}{\nu}$$

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Handling Flow at Solid Walls

Turbulence Near the Wall

- By scaling the variables near the wall the velocity profile data takes on a predictable form (transitioning from linear to logarithmic behavior)

Scaling the non-dimensional velocity and non-dimensional distance from the wall results in a predictable boundary layer profile for a wide range of flows



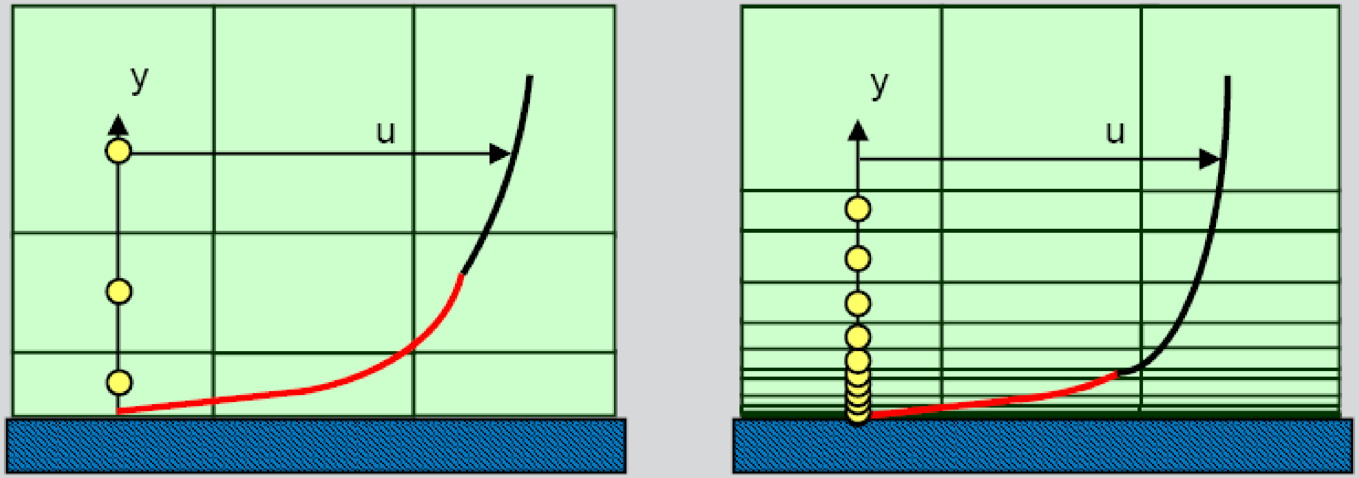
- Since near wall conditions are often predictable, functions can be used to determine the near wall profiles rather than using a fine mesh to actually resolve the profile
  - These functions are called wall functions

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Handling Flow at Solid Walls

Turbulence Near the Wall

- Fewer nodes are needed normal to the wall when Logarithmic-based wall functions are used (compared to more detailed low-Re wall modeling)



Logarithmic-based Wall functions used to resolve boundary layer

Near-wall resolving approach used to resolve boundary layer

Boundary layer

First node wall distance is reflected by y+ value

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Placement of The First Grid Point

- For Logarithmic-based wall functions, each wall-adjacent cell centroid should be located within the log-law layer : $y_p^+ \approx 30 - 300$
- For resolved wall treatment, each wall-adjacent cell centroid should be located within the viscous sublayer : $y_p^+ \approx 1$
- How to estimate the size of wall-adjacent cells before creating the grid:

$$y_p^+ = \frac{y_p u_\tau}{\nu} \quad \Rightarrow \quad y_p = \frac{y_p^+ \nu}{u_\tau} \qquad u_\tau = \sqrt{\frac{\tau_w}{\rho}} = U_e \sqrt{\frac{\overline{C}_f}{2}}$$

Flat Plate: $\frac{\overline{C}_f}{2} \approx \frac{0.037}{\mathrm{Re}_L^{1/5}}$

Duct: $\frac{\overline{C}_f}{2} \approx \frac{0.039}{\mathrm{Re}_{D_h}^{1/4}}$

y^+ ~10-15 values should be avoided!

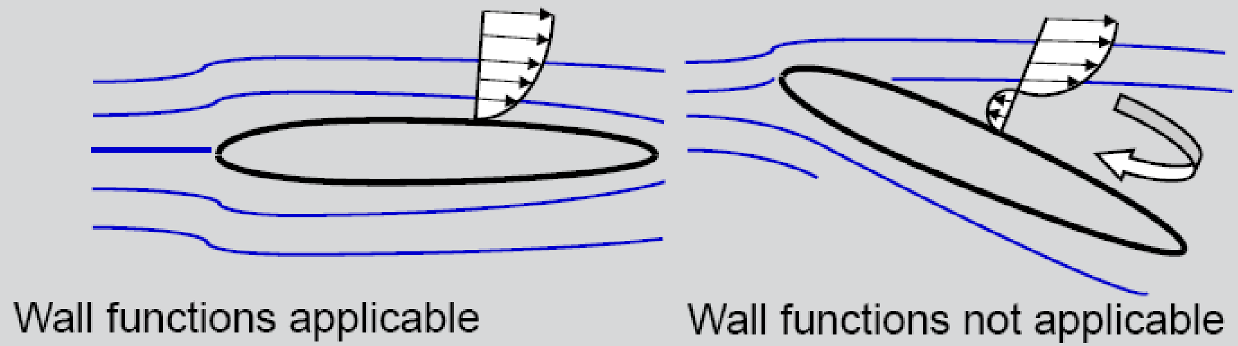
→ y^+= 11.067 is the exact transition point between the linear and logarithmic behavior of the boundary layer

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Handling Flow at Solid Walls

Limitations of Wall Function

- In some situations, such as boundary layer separation, logarithmic-based wall functions do not correctly predict the boundary layer profile



- In these cases logarithmic-based wall functions should not be used
- Instead, directly resolving the boundary layer can provide accurate results
- Not all turbulence models allow the wall functions to be turned off

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Handling Flow at Solid Walls

y+ for the SST and k-omega Models

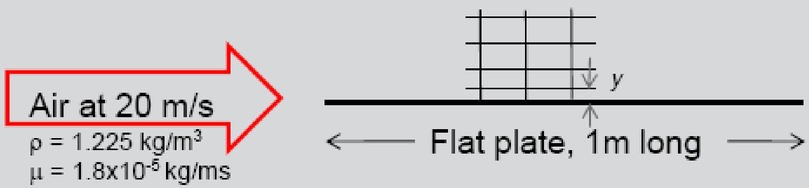
- **When using the SST or *k*-ω models y^+ should be < 300 so that the logarithmic-based wall function approach is valid**
  - This will not take advantage of the low-Reynolds formulation, which is necessary for accurate separation prediction
  - However, the model can still be used on these coarser near-wall mesh and produce valid results, within the limitations of the log wall functions
- **To take full advantage of the low-Reynolds formulation, y^+ should be < 2**

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Handling Flow at Solid Walls

Example in predicting near-wall cell size

- During the pre-processing stage, you will need to know a suitable size for the first layer of grid cells (inflation layer) so that Y^+ is in the desired range.
- The actual flow-field will not be known until you have computed the solution (and indeed it is sometimes unavoidable to have to go back and remesh your model on account of the computed Y^+ values).
- To reduce the risk of needing to remesh, you may want to try and predict the cell size by performing a hand calculation at the start. For example:



The question is what height (y) should the first row of grid cells be. We will use SWF, and are aiming for $Y^+ \approx 50$

- For a flat plate, Reynolds number ($\mathrm{Re}_l = \dfrac{\rho V L}{\mu}$) gives Re_l = 1.4x10^6
- (Recall from earlier slide, flow over a surface is turbulent when Re_L > 5x10^5)

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Handling Flow at Solid Walls

Example in predicting near-wall cell size [2]

- A literature search suggests a formula for the skin friction on a plate[1] thus

$$C_f = 0.058\,\mathrm{Re}_l^{-0.2}$$
$$C_f = 0.0034$$

- Use this value to predict the wall shear stress τ_w

$$\tau_w = \frac{1}{2} C_f \rho U_\infty^2 \qquad \tau_w = 0.83\ \mathrm{kg/ms^2}$$

- From τ_w compute the velocity U_τ

$$U_\tau = \sqrt{\frac{\tau_w}{\rho}} \qquad U_\tau = 0.82\ \mathrm{m/s}$$

- Rearranging the equation shown previously for y^+ gives a formula for the first cell height, *y*, in terms of $U\tau$

$$y = 9x10^{-4}\,m$$

- We know we are aiming for y^+ of 50, hence our first cell height *y* should be approximately 1 mm.

[1] An equivalent formula for internal flows, based on the pipe-diameter Reynolds number is

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Example in predicting near-wall cell size [3]

For Conjugate Heat Transfer Simulations one would need a y+ value of 1. Let's estimate the first grid node for y+= 1:

V= 20 m/s , ρ = 1.225 kg/m3 , μ = 1.8x10-5 kg/ms

$$\mathrm{Re}_l = \frac{\rho V L}{\mu}$$ → $\mathrm{Re}_l = 1.4\mathrm{x}10^6$

→ C_f =0.0034

→ t_w = 0.83 kg/ms^2

$$C_f = 0.058\ \mathrm{Re}_l^{-0.2}$$ → U_τ = 0.82 m/s

$$U_\tau = \sqrt{\frac{\tau_w}{\rho}}$$ → y = 0.02 mm

$$\upsilon = \frac{\mu}{\rho} = 1.469 \times 10^{-5}$$

$$y = \frac{y^+ \upsilon}{U_\tau} = 1.8 \times 10^{-5} m$$

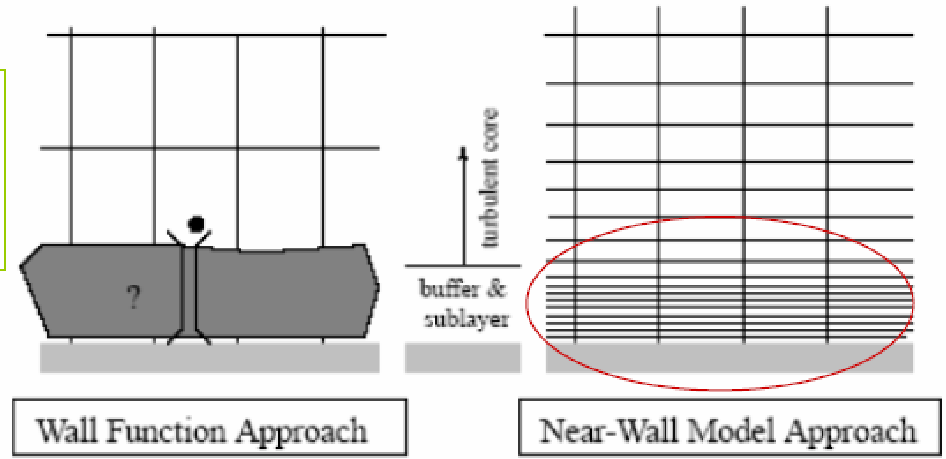
aiming for y+ of 1:
our first cell height y
should be ≈ 0.02 mm

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Handling Flow at Solid Walls

Wall Boundary Conditions

-Standard k- ε
-RNG k- ε
-Realizable k- ε
-RSM



-Spalart-Allmaras
-Standard ω - ε
-SST k- ω

- The viscosity-affected region is not resolved, instead is bridged by the wall function.
- High-Re turbulence models can be used.

- The near-wall region is resolved all the way down to the wall.
- The turbulence models ought to be valid throughout the near-wall region.

Forrás: Introduction to ANSYS CFX

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Handling Flow at Solid Walls

Placement of The First Grid Point

- Wall function ~ $y^+ \fallingdotseq$ 30-300
- Near-wall function ~ $y^+ \fallingdotseq 1$

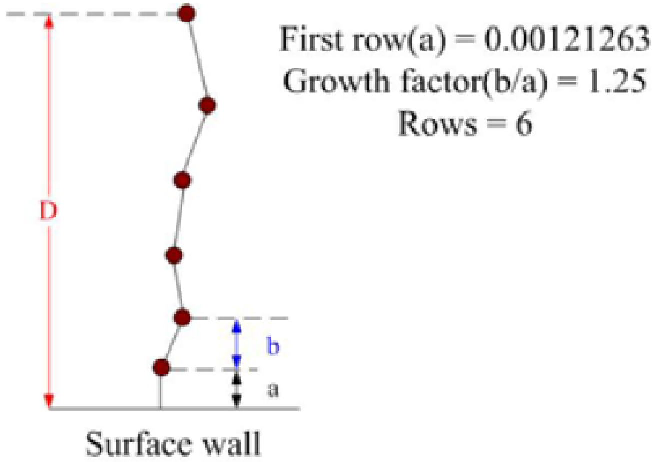
Non-dimensional distance from a wall in a turbulent boundary layer is given by y+

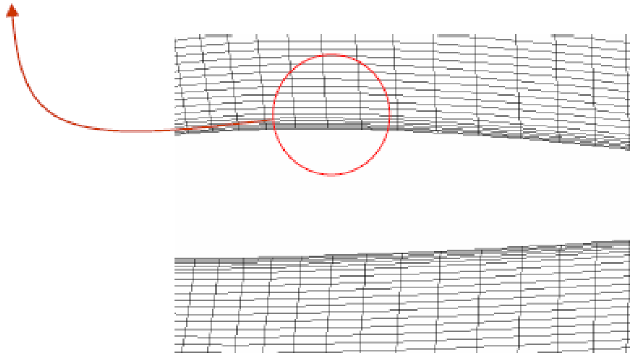
$$y^+ = \frac{\rho u_t y}{\mu}$$

where u_t is the friction velocity, define as $(\tau_w/\rho)^{0.5}$.

The shear stress on the surface:

$$\frac{\tau_w}{\frac{1}{2}\rho U^2} = \frac{0.73}{\sqrt{R_N}}$$





Forrás: Introduction to ANSYS CFX

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Handling Flow at Solid Walls

ANSYS CFX

Formulation

The estimates will be based on correlations for a flat plate with a Reynolds number of:

$$Re_L = \frac{\rho U_\infty L}{\mu} \qquad \text{Equation 14.}$$

with characteristic velocity U_∞ and length of the plate L.

The correlation for the wall shear stress coefficient, c_f is given by:

$$c_f = 0.025 Re_x^{-1/7} \qquad \text{Equation 15.}$$

where x is the distance along the plate from the leading edge.

The definition of Δy^+ for this estimate is:

$$\Delta y^+ = \frac{\Delta y u_\tau}{\nu} \qquad \text{Equation 16.}$$

with Δy being the mesh spacing between the wall and the first node away from the wall.

Using the definition

$$c_f = 2\frac{\rho u_\tau^2}{\rho U_\infty^2} = 2\left(\frac{u_\tau}{U_\infty}\right)^2 \qquad \text{Equation 17.}$$

Forrás: Introduction to ANSYS CFX

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Handling Flow at Solid Walls

ANSYS CFX

u_τ can be eliminated in Equation 16 to yield:

$$\Delta y = \Delta y^+ \sqrt{\frac{2}{c_f}} \frac{\nu}{U_\infty} \qquad \text{Equation 18.}$$

c_f can be eliminated using Equation 15 to yield:

$$\Delta y = L\,\Delta y^+ \sqrt{80}\, Re_x^{1/14} \frac{1}{Re_L} \qquad \text{Equation 19.}$$

Further simplification can be made by assuming that:

$$Re_x = C\, Re_L$$

where C is some fraction.

Assuming that $C^{1/14} \approx 1$, then, except for very small Re_x, the result is:

$$\Delta y = L\,\Delta y^+ \sqrt{80}\, Re_L^{-13/14} \qquad \text{Equation 20.}$$

This equation allows us to set the target Δy^+ value at a given x location and obtain the mesh spacing, Δy for nodes in the boundary layer.

Forrás: Introduction to ANSYS CFX

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Handling Flow at Solid Walls

ANSYS CFX

Minimum Number of Nodes

Goal

A good mesh should have a minimum number of mesh points inside the boundary layer in order for the turbulence model to work properly. As a general guideline, a boundary layer should be resolved with at least:

$$N_{normal} = \begin{cases} 10 \text{ for wall function} \\ 15 \text{ for low-Re model} \end{cases} \qquad \text{Equation 21.}$$

where N_{normal} is the number of nodes in the boundary layer in the direction normal to the wall.

Formulation

The boundary layer thickness δ can then be computed from the correlation:

$$Re_\delta = 0.14 Re_x^{6/7} \qquad \text{Equation 22.}$$

to be:

$$\delta = 0.14\, L\, Re_x^{6/7} \frac{1}{Re_L} \qquad \text{Equation 23.}$$

The boundary layer for a blunt body does not start with zero thickness at the stagnation point for Re_x. It is, therefore, safe to assume that Re_δ is some fraction of Re_L, say 25%. With this assumption, you get:

$$\delta = 0.035\, L\, Re_L^{-1/7} \qquad \text{Equation 24.}$$

You would, therefore, select a point, say the fifteenth off the surface (for a low-Re model, or 10th for a wall function model) and check to make sure that:

$$n(15) - n(1) \le \delta \qquad \text{Equation 25.}$$

Forrás: Introduction to ANSYS CFX

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Handling Flow at Solid Walls

ANSYS CFX

It is important to note the following points:

- To fully resolve the boundary layer, you should put at least 10 nodes into the boundary layer.
- Do not use Standard Wall Functions unless required for backwards compatibility.
- The upper limit for y^+ is a function of the device Reynolds number. For example, a large ship may have a Reynolds number of 10^9 and y^+ can safely go to values much greater than 1000. For lower Reynolds numbers (for example, a small pump), the entire boundary layer might only extend to around $y^+ = 300$. In this case, a fine near wall spacing is required to ensure a sufficient number of nodes in the boundary layer.

If the results deviate greatly from these ranges, the mesh at the designated Wall boundaries will require modification, unless wall shear stress and heat transfer are not important in the simulation.

Forrás: ANSYS, Inc., *ANSYS CFX-Solver Theory Guide, Release 14.5*, ANSYS, Inc. Southpointe, 275 Technology Derive Canonsburg, PA 15317, ansysinfo@ansys.com, http://www.ansys.com, USA, 2012

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Inlet Boundary Condition

Inlet Turbulence Conditions

When turbulent flow enters a domain at inlets or outlets (backflow), boundary conditions for k, ε, ω and/or $\overline{u'_i u'_j}$ must be specified, depending on which turbulence model has been selected

Several options exist for the specification of turbulence quantities at inlets:

- Explicitly input k, ε, ω, or $\overline{u'_i u'_j}$
- Turbulence intensity and length scale
- Turbulence intensity and turbulent viscosity ratio

  Turbulent Intensity : $I = \frac{u'}{U} \approx \frac{1}{U}\sqrt{\frac{2k}{3}}$

  Turbulent viscosity ratio : μ_t/μ

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Inlet Boundary Condition

CFX Inlet Turbulence Conditions

- **Default Intensity and Autocompute Length Scale**
  - The default turbulence intensity of 0.037 (3.7%) is used together with a computed length scale to approximate inlet values of *k* and ε. The length scale is calculated to take into account varying levels of turbulence.
  - In general, the autocomputed length scale is not suitable for external flows
- **Intensity and Autocompute Length Scale**
  - This option allows you to specify a value of turbulence intensity but the length scale is still automatically computed. The allowable range of turbulence intensities is restricted to 0.1%-10.0% to correspond to very low and very high levels of turbulence accordingly.
  - In general, the autocomputed length scale is not suitable for external flows
- **Intensity and Length Scale**
  - You can specify the turbulence intensity and length scale directly, from which values of *k* and ε are calculated

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Inlet Boundary Condition

CFX Inlet Turbulence Conditions

- **Low (Intensity = 1%)**
  - This defines a 1% intensity and a viscosity ratio equal to 1
- **Medium (Intensity = 5%)**
  - This defines a 5% intensity and a viscosity ratio equal to 10
  - This is the recommended option if you do not have any information about the inlet turbulence
- **High (Intensity = 10%)**
  - This defines a 10% intensity and a viscosity ratio equal to 100
- **Specified Intensity and Eddy Viscosity Ratio**
  - This defines a 10% intensity and a viscosity ratio equal to 100
  - Use this feature if you wish to enter your own values for intensity and viscosity ratio
- **k and Epsilon**
  - Specify the values of *k* and ε directly
- **Zero Gradient**
  - Use this setting for fully developed turbulence conditions

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Inlet Boundary Condition

Inlet Turbulence Conditions

- If you have absolutely no idea of the turbulence levels in your simulation, you could use following values of turbulence intensities and length scales:
- Usual turbulence intensities range from 1% to 5%
- The default turbulence intensity value of 0.037 (that is, 3.7%) is sufficient for nominal turbulence through a circular inlet, and is a good estimate in the absence of experimental data

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Summary

RANS Turbulence Model

Model	Description
Standard k–ε	The baseline two-transport-equation model solving for k and ε. This is the default k–ε model. Coefficients are empirically derived; valid for fully turbulent flows only. Options to account for viscous heating, buoyancy, and compressibility are shared with other k–ε models.
RNG k–ε Re-Normalisation Group	A variant of the standard k–ε model. Equations and coefficients are analytically derived. Significant changes in the ε equation improves the ability to model highly strained flows. Additional options aid in predicting swirling and low Reynolds number flows.
Standard k–ω	A two-transport-equation model solving for k and ω, the specific dissipation rate (ε / k) based on Wilcox (1998). This is the default k–ω model. Demonstrates superior performance for wall-bounded and low Reynolds number flows. Shows potential for predicting transition. Options account for transitional, free shear, and compressible flows.
SST k–ω	A variant of the standard k–ω model. Combines the original Wilcox model for use near walls and the standard k–ε model away from walls using a blending function. Also limits turbulent viscosity to guarantee that $\tau_T \sim k$. The transition and shearing options are borrowed from standard k–ω. No option to include compressibility.
RSM	Reynolds stresses are solved directly using transport equations, avoiding isotropic viscosity assumption of other models. Use for highly swirling flows. Quadratic pressure-strain option improves performance for many basic shear flows.

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Summary

RANS Turbulence Model usage

Model	Behavior and Usage
Standard k–ε	Robust. Widely used despite the known limitations of the model. Performs poorly for complex flows involving severe pressure gradient, separation, strong streamline curvature. Suitable for initial iterations, initial screening of alternative designs, and parametric studies.
RNG k–ε	Suitable for complex shear flows involving rapid strain, moderate swirl, vortices, and locally transitional flows (e.g. boundary layer separation, massive separation, and vortex shedding behind bluff bodies, stall in wide-angle diffusers, room ventilation).
Standard k–ω	Superior performance for wall-bounded boundary layer, free shear, and low Reynolds number flows. Suitable for complex boundary layer flows under adverse pressure gradient and separation (external aerodynamics and turbomachinery). Can be used for transitional flows (though tends to predict early transition). Separation is typically predicted to be excessive and early.
SST k–ω	Offers similar benefits as standard k–ω. Dependency on wall distance makes this less suitable for free shear flows.
RSM	Physically the most sound RANS model. Avoids isotropic eddy viscosity assumption. More CPU time and memory required. Tougher to converge due to close coupling of equations. Suitable for complex 3D flows with strong streamline curvature, strong swirl/rotation (e.g. curved duct, rotating flow passages, swirl combustors with very large inlet swirl, cyclones).

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Flow Modelling by Means of Continuum Mechanics; RANS; Modelling of Turbulence - Summary

Summary – Turbulence Modeling Guidelines

- **Successful turbulence modeling requires engineering judgment of:**
  - Flow physics
  - Computer resources available
  - Project requirements
    - Accuracy
    - Turnaround time
  - Near-wall treatments
- **Modeling procedure**
  - Calculate characteristic *Re* and determine whether the flow is turbulent
  - Estimate y^+ before generating the mesh
  - The SST model is good choice for most flows
  - Use the Reynolds Stress Model or the SST model with Curvature Correction (see documentation) for highly swirling, 3-D, rotating flows

Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)

Numerical Modelling of Flow

Discretization of the Governing Equations

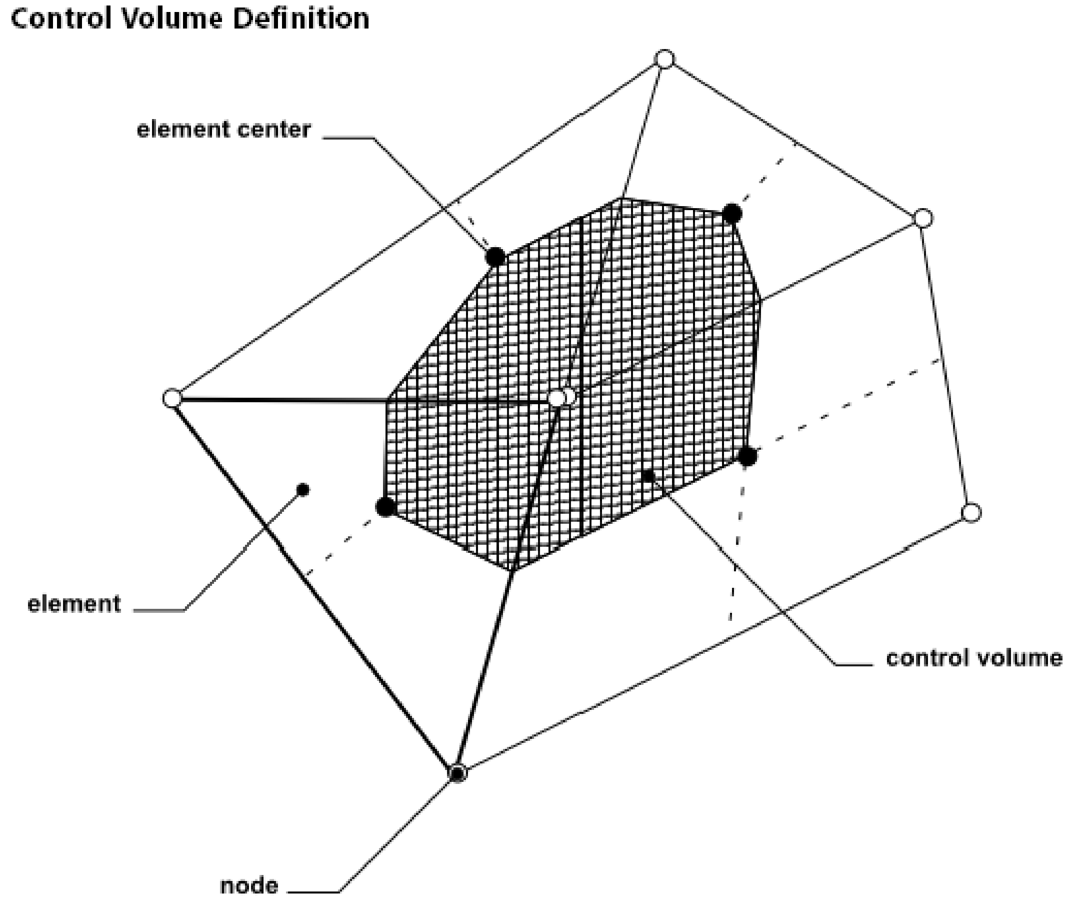
Flow Modelling by Means of Continuum Mechanics; RANS; Discretization - CFX

FINITE DIFFERENCE, 1. Introduced by Euler in the 18th century.
2. Governing equations in differential form→ domain with grid→ replacing the partial derivatives by approximations in terms of node values of the functions→ one algebraic equation per grid node→ linear algebraic equation system. 3. Applied to structured grids.

FINITE VOLUME, 1. Governing equations in integral form→ solution domain is subdivided into a finite number of contiguous control volumes→ conservation equation applied to each CV.
2. Computational node locates at the centroid of each CV.
3. Applied to any type of grids, especially complex geometries
4. Compared to FD, FV with methods higher than 2nd order will be difficult, especially for 3D.

FINITE ELEMENT,
1. Similar to FV
2. Equations are multiplied by a weight function before integrated over the entire domain.

Flow Modelling by Means of Continuum Mechanics; RANS; Discretization - CFX



Forrás: ANSYS CFX-Solver Theory Guide, ANSYS Inc., Southpointe, 275 Technology Drive, Canonsburg, PA 15317, October 2012

Flow Modelling by Means of Continuum Mechanics; RANS; Discretization - CFX

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial \boldsymbol{x}_j}\left(\rho\, \boldsymbol{U}_j\right) = 0$$

$$\frac{\partial}{\partial t}\left(\rho\, \boldsymbol{U}_i\right) + \frac{\partial}{\partial \boldsymbol{x}_j}\left(\rho\, \boldsymbol{U}_j\, \boldsymbol{U}_i\right) = -\frac{\partial P}{\partial \boldsymbol{x}_i} + \frac{\partial}{\partial \boldsymbol{x}_j}\left(\mu_{eff}\left(\frac{\partial \boldsymbol{U}_i}{\partial \boldsymbol{x}_j} + \frac{\partial \boldsymbol{U}_j}{\partial \boldsymbol{x}_i}\right)\right)$$

$$\frac{\partial}{\partial t}\left(\rho\, \varphi\right) + \frac{\partial}{\partial \boldsymbol{x}_j}\left(\rho\, \boldsymbol{U}_j\, \varphi\right) = \frac{\partial}{\partial \boldsymbol{x}_j}\left(\Gamma_{eff}\left(\frac{\partial \varphi}{\partial \boldsymbol{x}_j}\right)\right) + S_\varphi$$

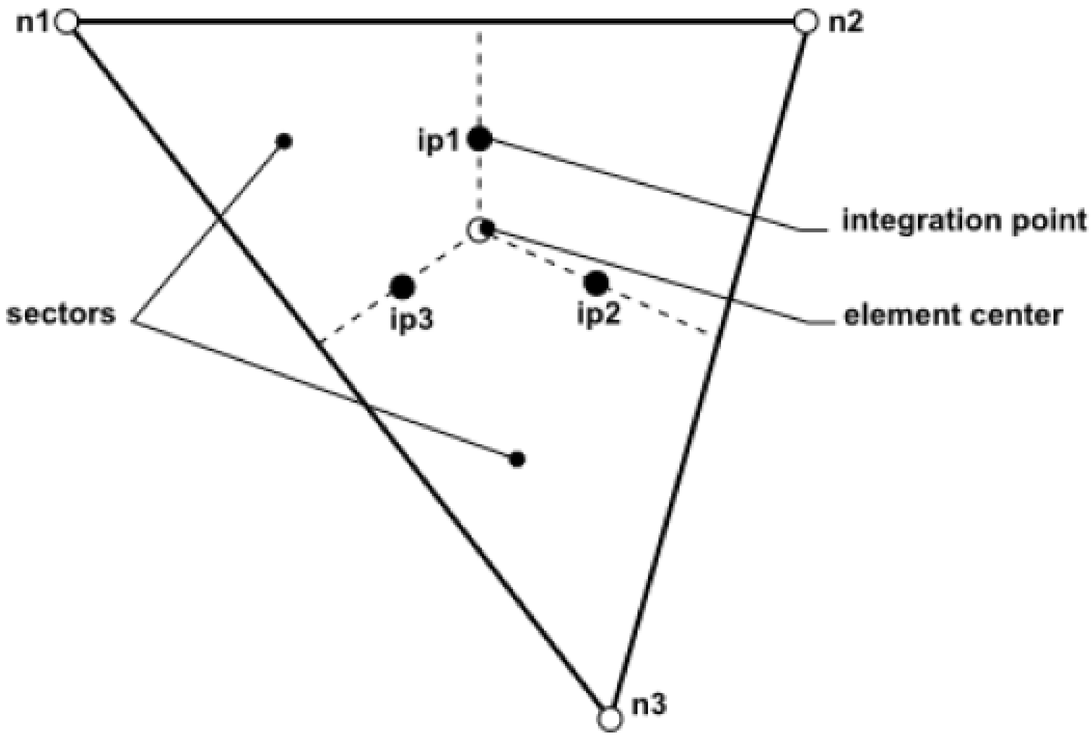
$$\frac{d}{dt}\int_V \rho\, dV + \int_s \rho\, \boldsymbol{U}_j\, dn_j = 0$$

$$\frac{d}{dt}\int_V \rho\, \boldsymbol{U}_i\, dV + \int_s \rho\, \boldsymbol{U}_j\, \boldsymbol{U}_i\, dn_j = -\int_s P\, dn_j + \int_s \mu_{eff}\left(\frac{\partial \boldsymbol{U}_i}{\partial \boldsymbol{x}_j} + \frac{\partial \boldsymbol{U}_j}{\partial \boldsymbol{x}_i}\right) dn_j + \int_V S_{\boldsymbol{U}_i}\, dV$$

$$\frac{d}{dt}\int_V \rho\, \varphi\, dV + \int_s \rho\, \boldsymbol{U}_j\, \varphi\, dn_j = \int_s \Gamma_{eff}\left(\frac{\partial \varphi}{\partial \boldsymbol{x}_j}\right) dn_j + \int_V S_\varphi\, dV$$

Forrás: ANSYS CFX-Solver Theory Guide, ANSYS Inc., Southpointe, 275 Technology Drive, Canonsburg, PA 15317, October 2012

Flow Modelling by Means of Continuum Mechanics; RANS; Discretization - CFX



Forrás: ANSYS CFX-Solver Theory Guide, ANSYS Inc., Southpointe, 275 Technology Drive, Canonsburg, PA 15317, October 2012

Flow Modelling by Means of Continuum Mechanics; RANS; Discretization - CFX

$$V\left(\frac{\rho-\rho^{o}}{\Delta t}\right)+\sum_{ip}\dot{m}_{ip}=0$$

$$V\left(\frac{\rho\,\boldsymbol{U}_i-\rho^{o}\,\boldsymbol{U}_i^{o}}{\Delta t}\right)+\sum_{ip}\dot{m}_{ip}\,(\boldsymbol{U}_i)_{ip}=\sum_{ip}(P\;\Delta n_i)_{ip}+$$

$$\sum_{ip}\left(\mu_{eff}\left(\frac{\partial\,\boldsymbol{U}_i}{\partial\,\boldsymbol{x}_j}+\frac{\partial\,\boldsymbol{U}_j}{\partial\,\boldsymbol{x}_i}\right)\Delta n_j\right)_{ip}+\overline{S_{\boldsymbol{U}_i}}\;V$$

$$V\left(\frac{\rho\,\varphi-\rho^{o}\,\varphi^{o}}{\Delta t}\right)+\sum_{ip}\dot{m}_{ip}\,\varphi_{ip}=\sum_{ip}\left(\Gamma_{eff}\,\frac{\partial\,\varphi}{\partial\,\boldsymbol{x}_j}\;\Delta n_j\right)_{ip}+\overline{S_{\varphi}}\;V$$

° : parameters of the next iteration step,   : parameters of the actual iteration step, First or Second order Implicit Backward Euler method has been used for solving the system of algebraic equations.

Forrás: ANSYS CFX-Solver Theory Guide, ANSYS Inc., Southpointe, 275 Technology Drive, Canonsburg, PA 15317, October 2012

Flow Modelling by Means of Continuum Mechanics; RANS; Solution of System of Algebraic Equations - CFX

ANSYS CFX uses a Multigrid (MG) accelerated Incomplete Lower Upper (ILU) factorization technique for solving the discrete system of linearized equations. It is an iterative solver whereby the exact solution of the equations is approached during the course of several iterations.

The linearized system of discrete equations described above can be written in the general matrix form:

$$[A]\,[\varphi] = [b]$$

where $[A]$ is the coefficient matrix, $[\varphi]$ the solution vector and $[b]$ the right hand side.

Forrás: ANSYS CFX-Solver Theory Guide, ANSYS Inc., Southpointe, 275 Technology Drive, Canonsburg, PA 15317, October 2012

Flow Modelling by Means of Continuum Mechanics; RANS; Solution of System of Algebraic Equations - CFX

Pressure-Velocity Coupling: ANSYS CFX uses a co-located (non-staggered) grid layout such that the control volumes are identical for all transport equations. As discussed by Patankar, however, naïve co-located methods lead to a decoupled (checkerboard) pressure field. Rhie and Chow [2] proposed an alternative discretization for the mass flows to avoid the decoupling, and this discretization was modified by Majumdar to remove the dependence of the steady-state solution on the time step. Similar strategy is adopted in ANSYS CFX.

Solver: ANSYS CFX uses a coupled solver, which solves the hydrodynamic equations (for *u*, *v*, *w*, *p*) as a single system by means of a fully implicit discretization of the equations at any given time step. For steady state problems, the time-step behaves like an 'acceleration parameter', to guide the approximate solutions in a physically based manner to a steady-state solution. This reduces the number of iterations required for convergence to a steady state, or to calculate the solution for each time step in a time-dependent analysis. ANSYS CFX uses first or second order (better for transient due to the higher accuracy) Backward Euler scheme for time discretization (see slide 85).

Forrás: ANSYS CFX-Solver Theory Guide, ANSYS Inc., Southpointe, 275 Technology Drive, Canonsburg, PA 15317, October 2012

Flow Modelling by Means of Continuum Mechanics; RANS; Numerical Methods and Their Characteristics

Mathematical properties of the numerical discretization

System of partial differential equations, L(Ť) $\xrightarrow{\text{Discretization}}$ System of algebraic equations

System of partial differential equations, L(Ť) $\xleftarrow{\text{Consistency}}$ System of algebraic equations

System of algebraic equations $\xrightarrow{\text{Stability}}$ Approximate solution, T

Exact solution, Ť $\xleftarrow{\text{Convergence}}$ Approximate solution, T

if $\Delta x, \Delta t \rightarrow 0$

- Initial and boundary conditions

The Main Steps and Rules of Completing a CFD Task

The Main Steps of a CFD Task

Pre-processing

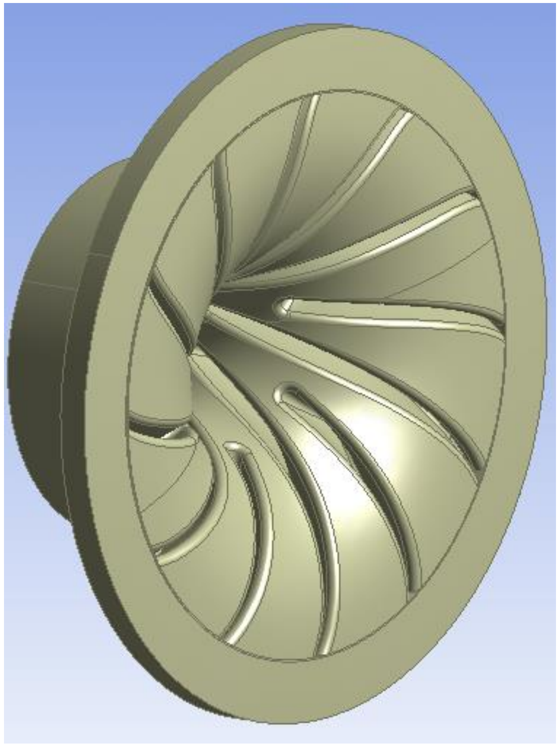
- Goal, definition and review of the task which has to be solved and mapped that into the modelling space, making time and cost plan,
- Creating geometrical (flow) model,
- Making of the numerical mesh in the flow domain,
- Definition of the material properties,
- Setting up related physical models and their parameters,
- Imposing boundary conditions and assigning them to the geometry,
- Specify initial conditions,
- Setting solver parameters,
- Start calculation and evaluate convergences,

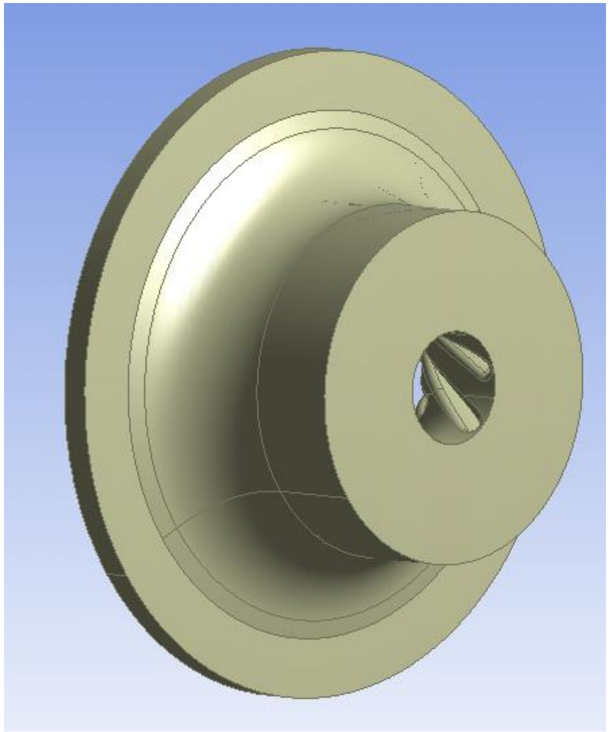
Post-processing

- View, analyze, and evaluate the results,
- Verification, plausibility check and validation,
- Mesh and parameter sensitivity analyses,
- Documentation preparation with especial care for the suggestions for improvements in case of relevancies.

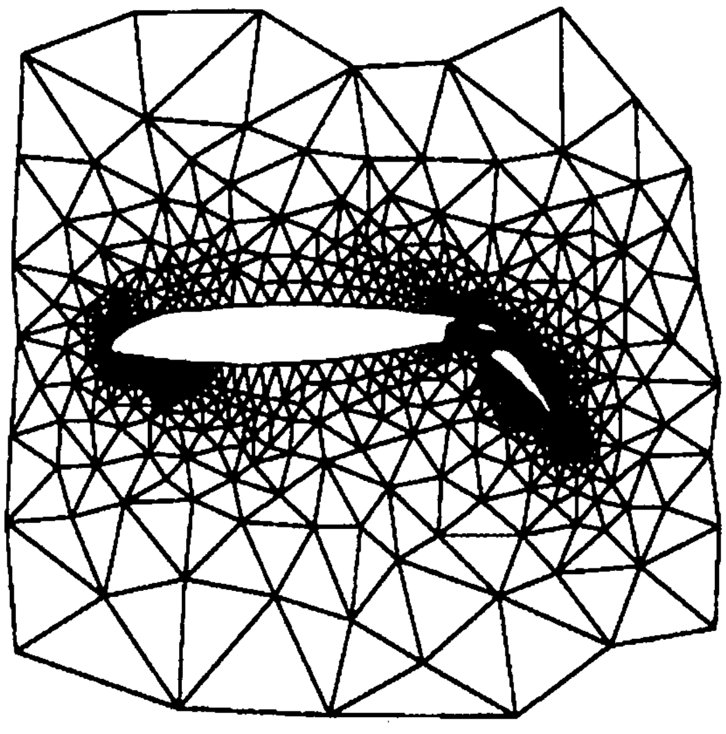
The Main Steps of a CFD Task

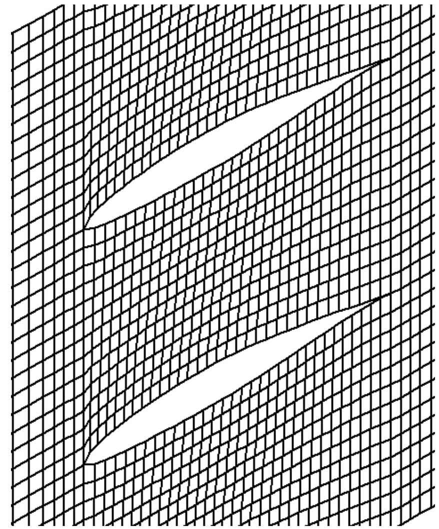
Geometry – Flow field

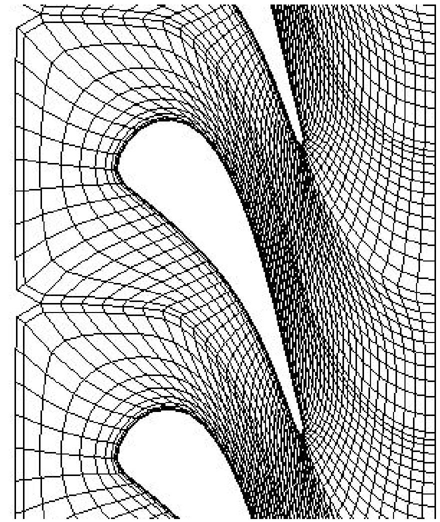


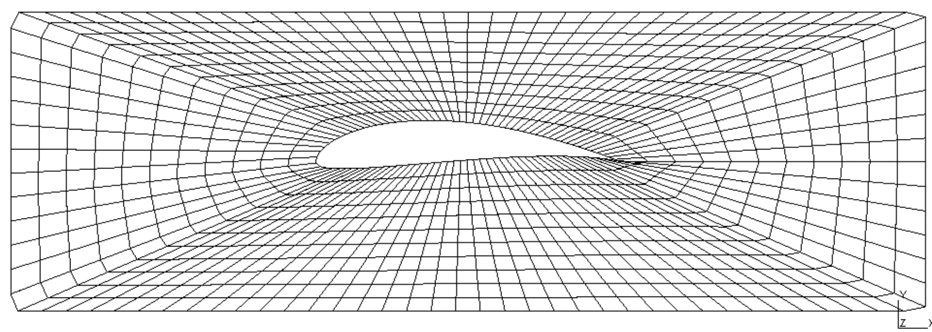


The Main Steps of a CFD Task – Discretization of the Flow Field

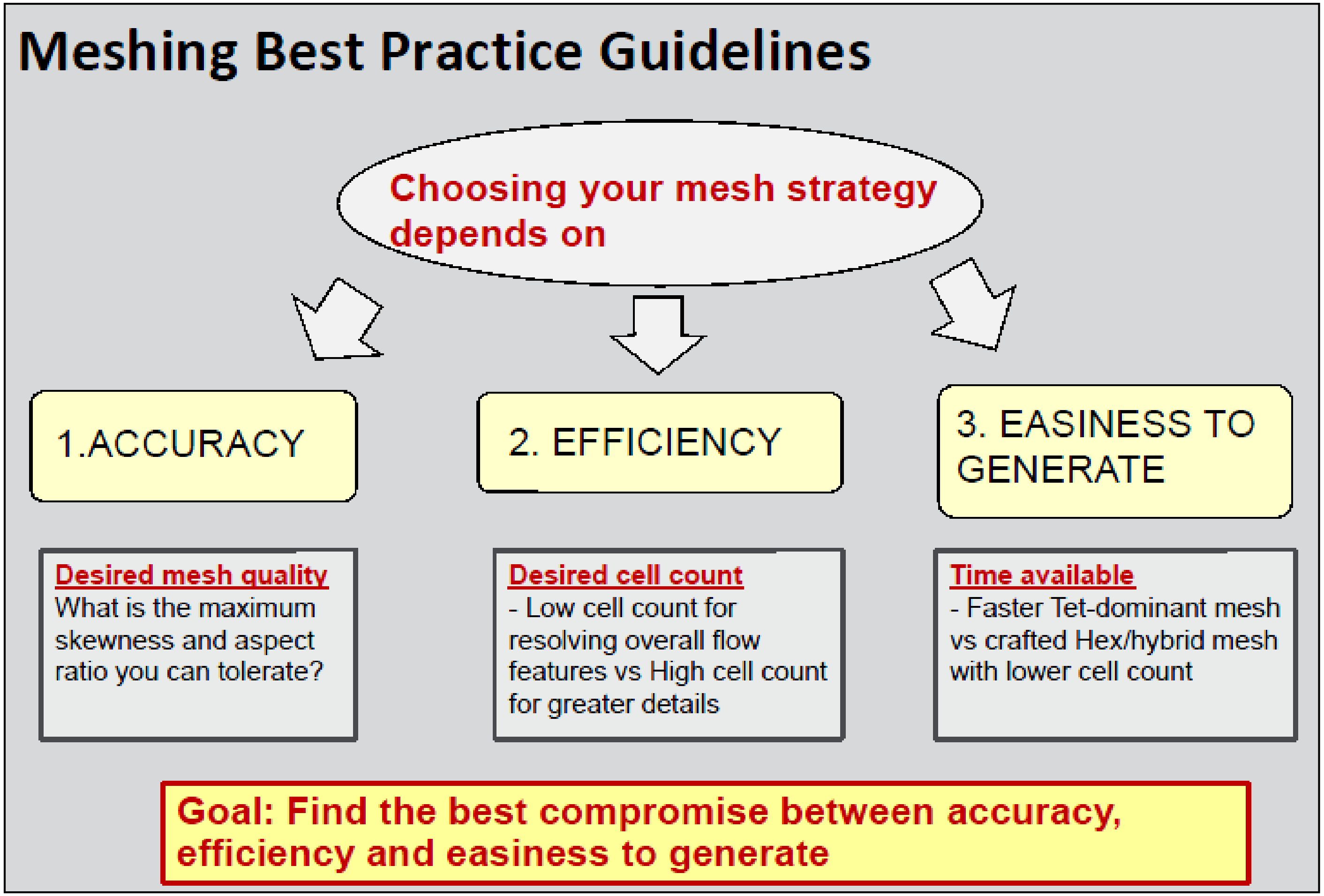








The Main Steps of a CFD Task – Discretization of the Flow Field



Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)

The Main Steps of a CFD Task – Discretization of the Flow Field

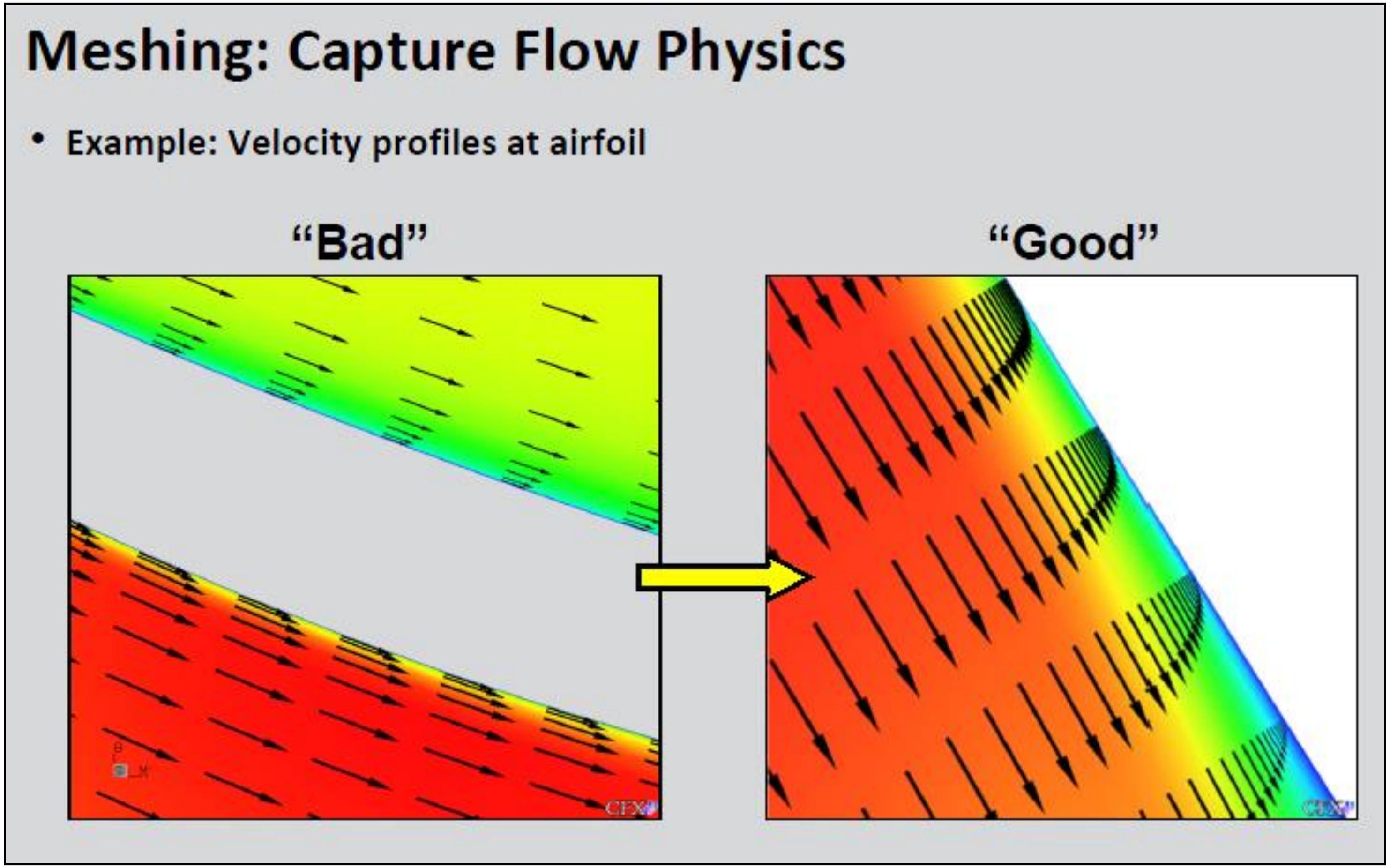
Meshing: Capture Flow Physics

- **Grid must be able to capture important physics:**
  - Boundary layers
  - Heat transfer
  - Wakes, shock
  - Flow gradients

- **Boundary layers:**
  - Velocity and temperature
  - 10-15 elements
  - Expansion ratios:
    - ≤ 1.2 ... 1.3
  - y+ ≈ 1 for heat transfer and transition modeling

Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)

The Main Steps of a CFD Task – Discretization of the Flow Field

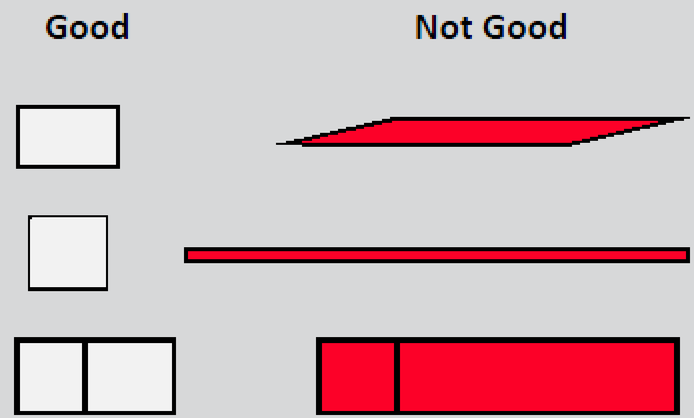


Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)

The Main Steps of a CFD Task – Discretization of the Flow Field

Mesh Quality

- A good mesh depends on :
  - Cell not too distorted
  - Cell not too stretched
  - Smooth Cells transition



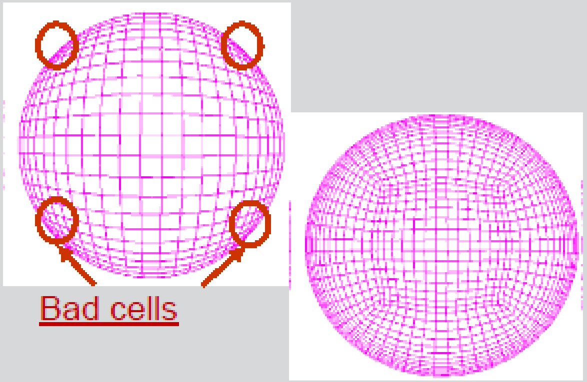
Részletesen lásd: Mesh Metrix, Mesh_metrix_in_ANSYS_WB_13_v11.ppt

Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)

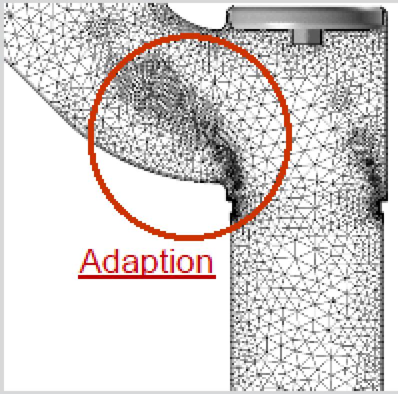
The Main Steps of a CFD Task – Discretization of the Flow Field

Mesh Quality

- **Grid generation:**
  - Scalable grids
  - Skewness < 0.9 (accuracy, convergence)
  - Aspect ratios < 100
  - Expansion ratios < 1.5 …2
  - Capture physics based on experience (shear layers, shocks)
  - Angle between grid face & flow vector



- **Grid refinement:**
  - Manual, based on error estimate
  - Automatic adaptive based on 'error sensor'

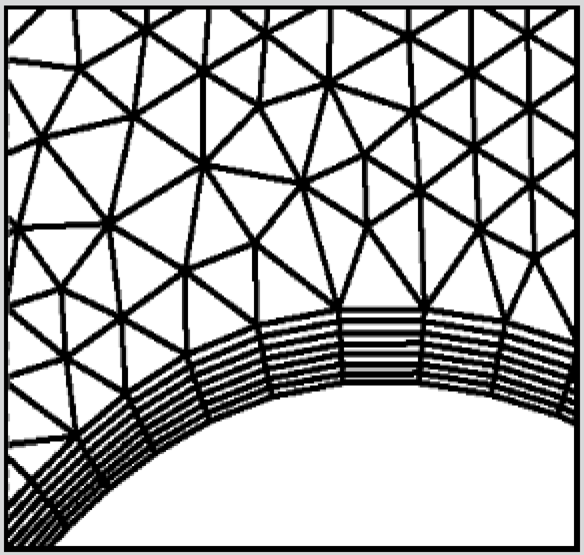


Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)

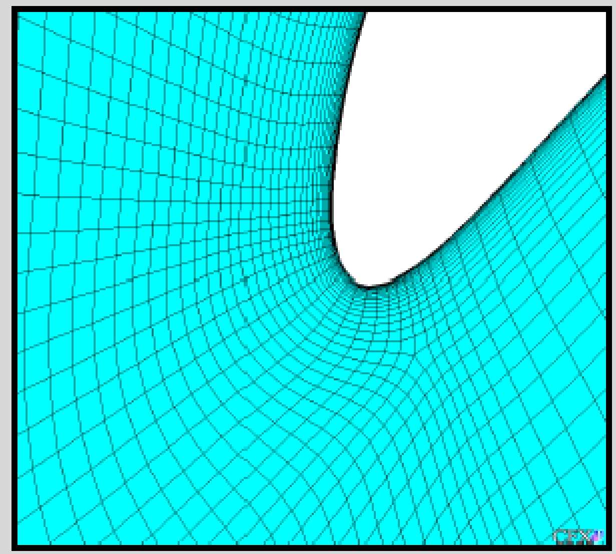
The Main Steps of a CFD Task – Discretization of the Flow Field

Mesh Quality

- Avoid sudden change in mesh density



Not good



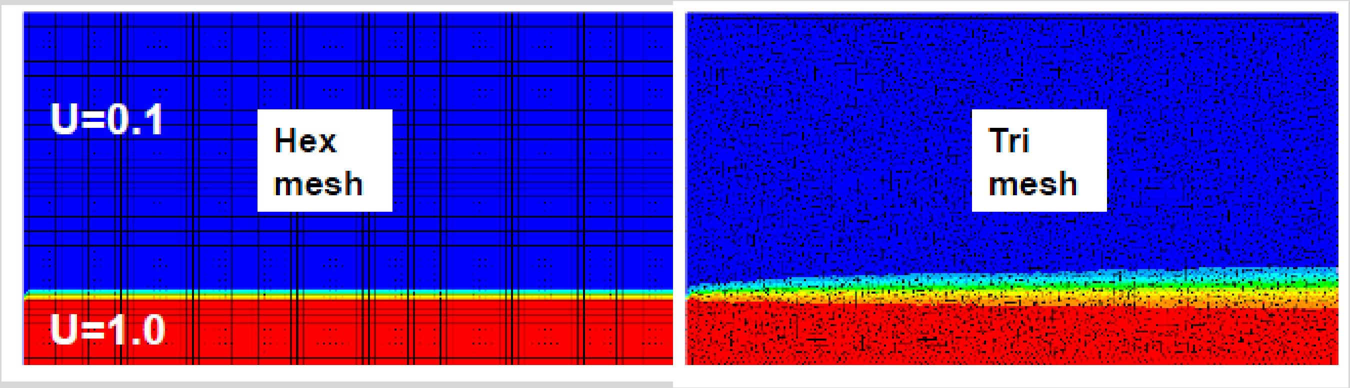
Good

Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)

The Main Steps of a CFD Task – Discretization of the Flow Field

Hex vs Tet Mesh : Accuracy comparison

- **Direction of the flow well known**
  ⇨ Quad/Hex aligned with the flow are more accurate than Tri with the same interval size



Contours of axial velocity magnitude for an inviscid co-flow jet

Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)

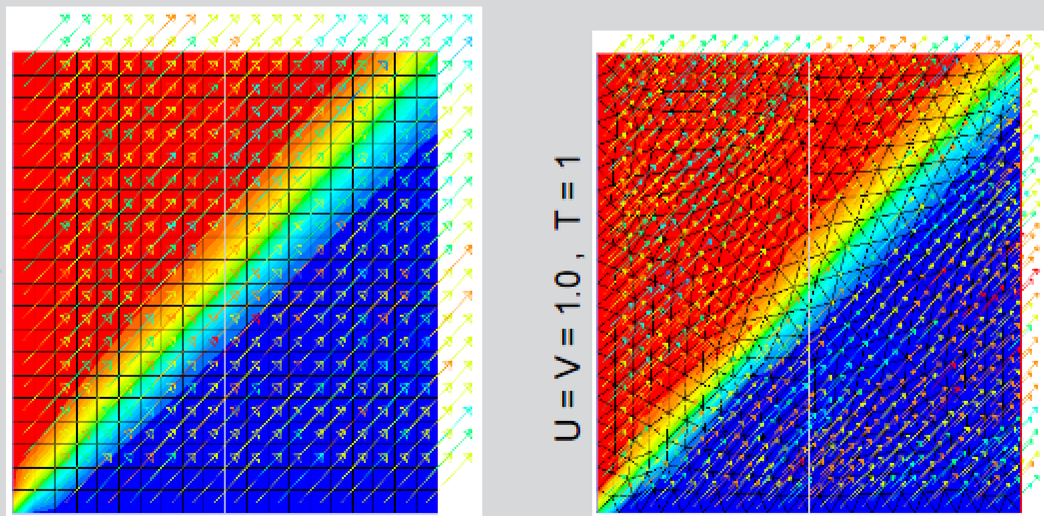
The Main Steps of a CFD Task – Discretization of the Flow Field

Hex vs Tet Mesh : Accuracy comparison

- For complex flows without dominant flow direction, Quad and Hex meshes lose their advantage
  ⇨ Quad & Tri equivalent

U = V = 1.0 , T = 1

U = V = 1.0 , T = 1



U = V = 1.0 , T = 0

U = V = 1.0 ,T = 0

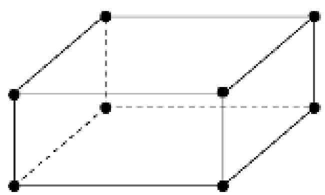
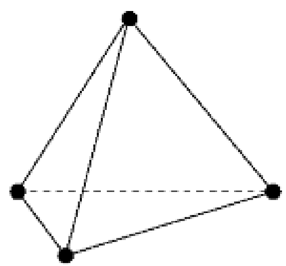
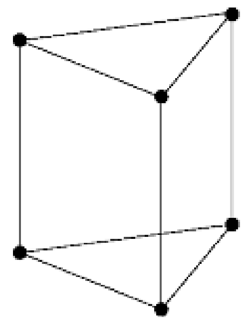
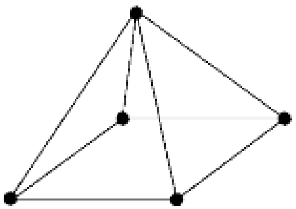
Contours of temperature for inviscid flow

Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)

The Main Steps of a CFD Task – Discretization of the Flow Field

Element Types

- Common 3-D element types:

Hexahedron (hex)	Tetrahedron (tet)	Prism	Pyramid
			

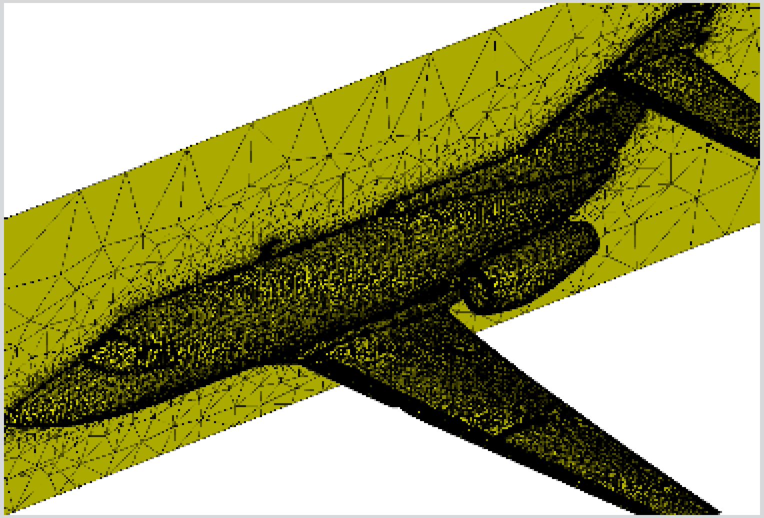
- General polyhedra, …
- Difference between control volumes & elements

Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)

The Main Steps of a CFD Task – Discretization of the Flow Field

Elements: Tet

- **Pro:**
  - High degree of automation for grid generation
- **Con:**
  - Memory & calculation time per node ≈ 1.5 × hex
  - Poor shear layer element
  - No streamline orientation
  - Quantity must (and can) make up for quality

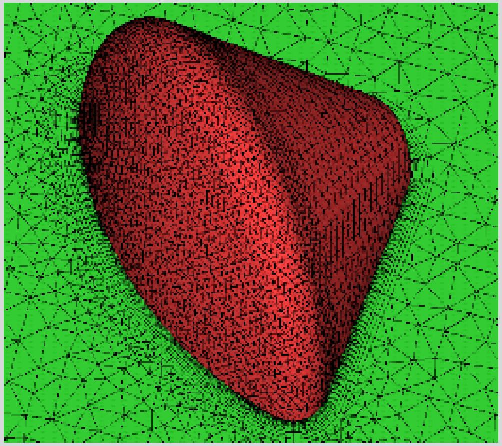


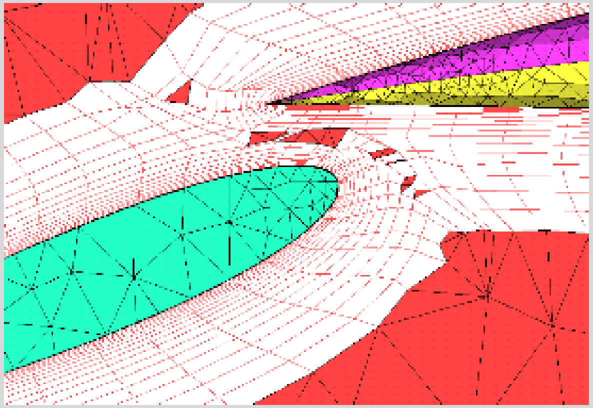
Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)

The Main Steps of a CFD Task – Discretization of the Flow Field

Elements: Prism

- **Pro:**
  - Better shear layer resolution than tet
  - High degree of automation
  - Tet/prism combination
- **Con:**
  - Less efficient than hex
  - Topological difficulties (corners, …) → poor grid quality (angles, …)
  - Manual repair



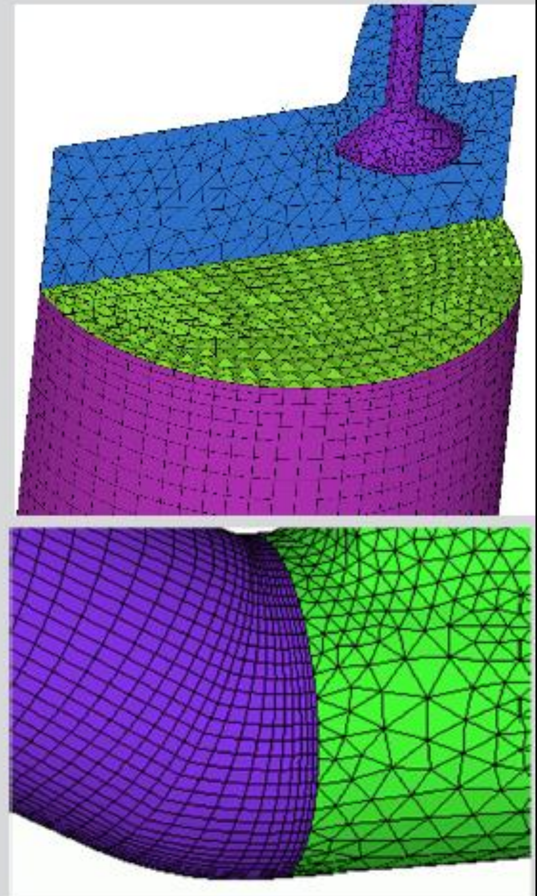


Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)

The Main Steps of a CFD Task – Discretization of the Flow Field

Elements: Pyramid

- **Use in hybrid grids**
- **Transition element between hex and tet**
- **Polyhedral grids**
  - ANSYS Fluent:
    - Generate base types
    - Convert
  - ANSYS CFX builds polyhedrals around vertices



Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)

The Main Steps of a CFD Task – Discretization of the Flow Field

Recommendations

- **1st Option → Hex grid**
  - Best accuracy and numerical efficiency
  - Time and effort manageable?
- **2nd Option → Tet/hex/pyramid grid**
  - Hex near walls & shear layers
  - Developing technology …
- **3rd Option → Tet/prism grid**
  - High degree of automation
  - Quality (prism/tet transition, …)
- **4th Option → Tet grid**
  - Shear layer resolution?

Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)

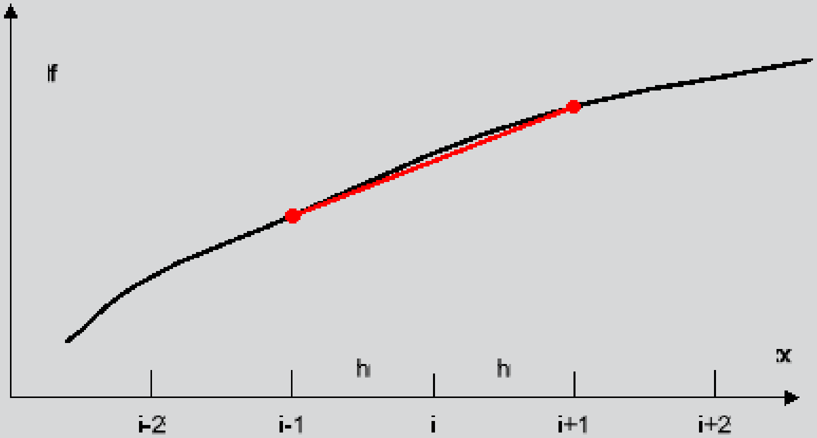
The Main Steps of a CFD Task – Discretization of the Flow Field

Grid Optimization

- Truncation errors → source of discretisation errors
- Minimize truncation errors → minimize discretization errors
- Truncation error → Difference between 'analog' and 'discrete' representation

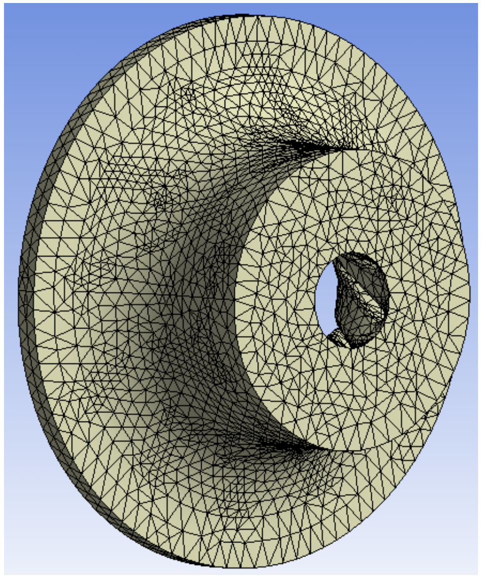
$$\left(\frac{\partial f}{\partial x}\right)_i = \frac{f_{i+1} - f_{i-1}}{2h} + \tau_i$$

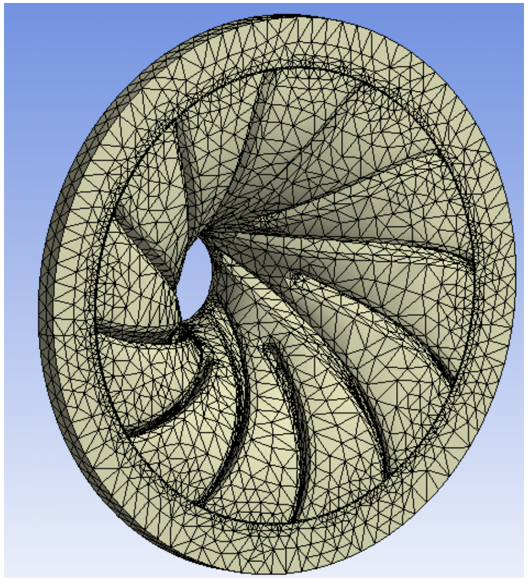
$$\tau_i = \frac{h^2}{6}\left(\frac{\partial^3 f}{\partial x^3}\right)_i + \ldots$$



Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)

The Main Steps of a CFD Task – Discretization of the Flow Field

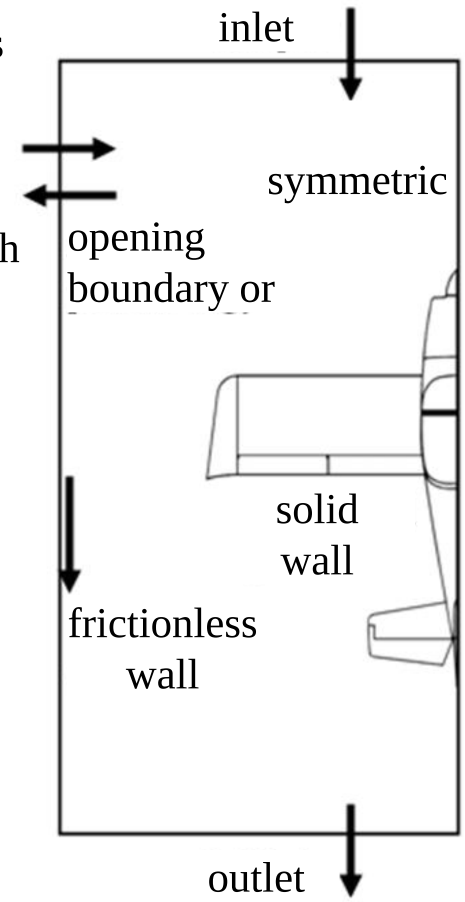


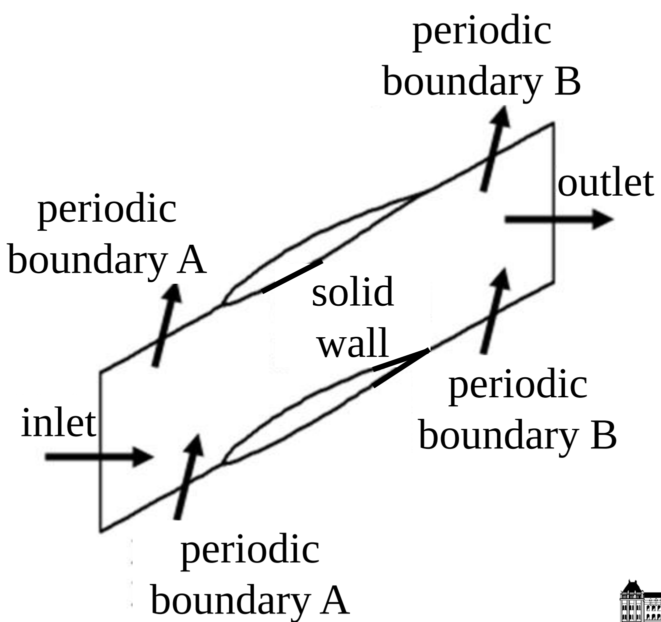


Rotational flow domain of a centrifugal compressor. The quality and the resolution of the mesh at the high gradient regions should be improved until it has influence for the results. Minimum 20 cells are necessary in case of the smallest gap at compressible flow.

The Main Steps of a CFD Task – Material Properties and Boundary Conditions

Typical boundary conditions in case of CFD simulations: inlet, outlet, solid wall, symmetry, opening (far field) and periodic. The boundaries should be placed on such a far distance from the object under investigation that the disturbances should not be propagated till the boundaries (e.g.: the streamlines should not cross the opening boundaries, the flow should not enter at the outlet section) together with the minimal number of cells with the mesh independent results.

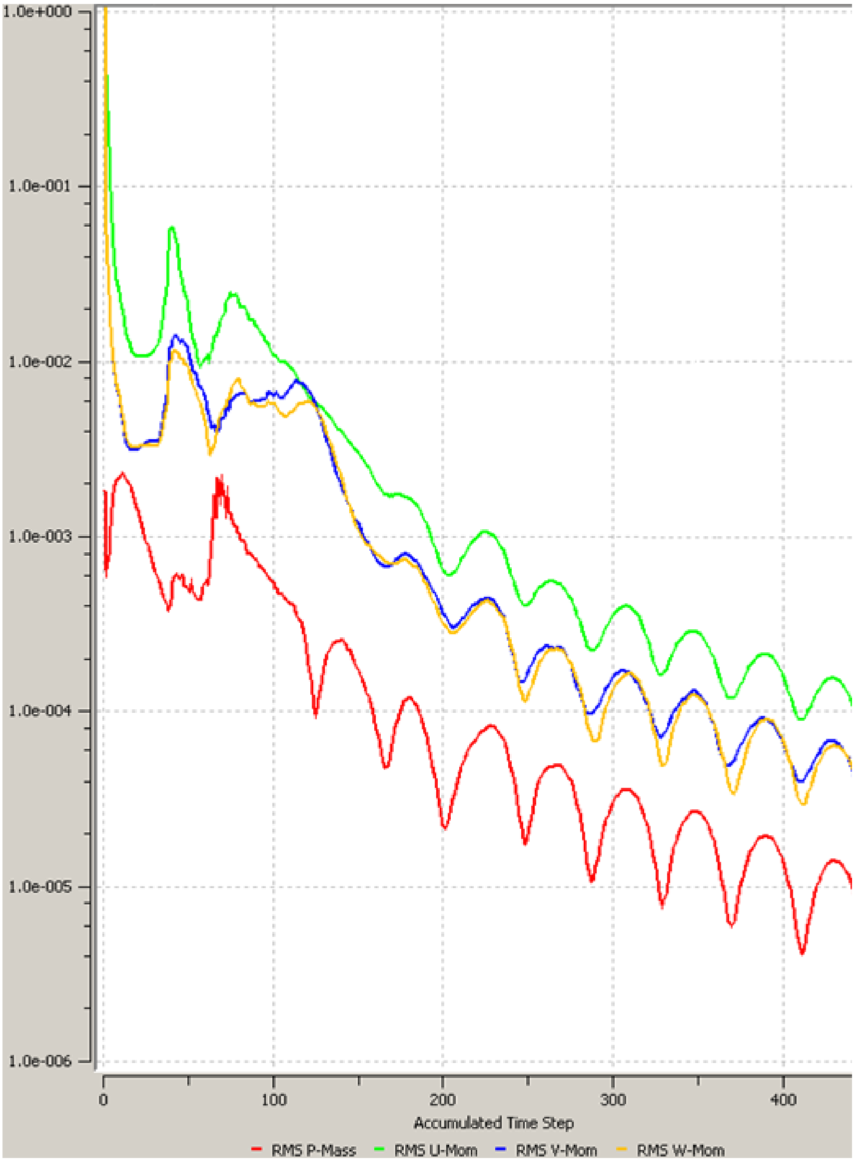




The Main Steps of a CFD Task – Solution and Convergence

Convergence

$$\left\| \frac{\Delta\rho}{\rho} \right\| = log_{10} \sqrt{\frac{1}{N_p} \sum_{i=1}^{N_p} \left(\frac{\Delta\rho_i}{\rho_i} \right)^2}$$



The Main Steps of a CFD Task – Solution and Convergence

Residuals Theory

- The continuous governing equations are discretized into a set of linear equations that can be solved. The set of linear equations can be written in the form:

$$[A]\,[\Phi] = [b]$$

  where [A] is the coefficient matrix and [Φ] is the solution variable

- If the equation were solved exactly we would have:

$$[A]\,[\Phi] - [b] = [0]$$

- The residual vector [R] is the error in the numerical solution:

$$[A]\,[\Phi] - [b] = [R]$$

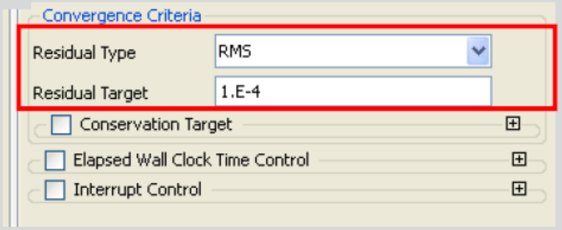
- Since each control volume has a residual we usually look at the RMS average or the maximum normalized residual

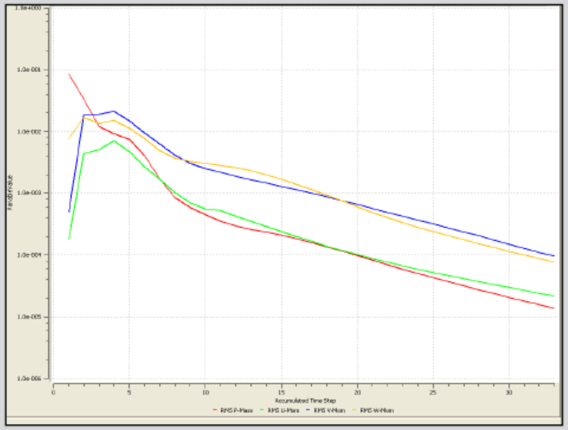
Forrás: Introduction to ANSYS CFX, Lecture 05 - Solver Settings and Output File - CFX-Intro_14.0_L05_SolverSettings_OutFile (2013.09.01.)

The Main Steps of a CFD Task – Solution and Convergence

Residuals

- **Residual Type**
  - MAX: Convergence based on maximum residual anywhere
  - RMS: Convergence based on average residual from all control volumes
  - Root Mean Square = $\sqrt{\dfrac{\sum_i R_i^2}{\mathrm{n}}}$
- **Residual Target**
  - For reasonable convergence MAX residuals should be 1.0E-3, RMS should be at least 1.0E-4
  - The targets dependent on the accuracy needed
    - Lower values may be needed for greater accuracy





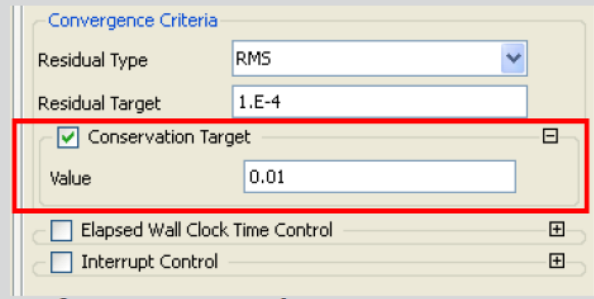
Forrás: Introduction to ANSYS CFX, Lecture 05 - Solver Settings and Output File - CFX-Intro_14.0_L05_SolverSettings_OutFile (2013.09.01.)

The Main Steps of a CFD Task – Solution and Convergence

Conservation Target

- **The Conservation Target sets a target for the global imbalances**

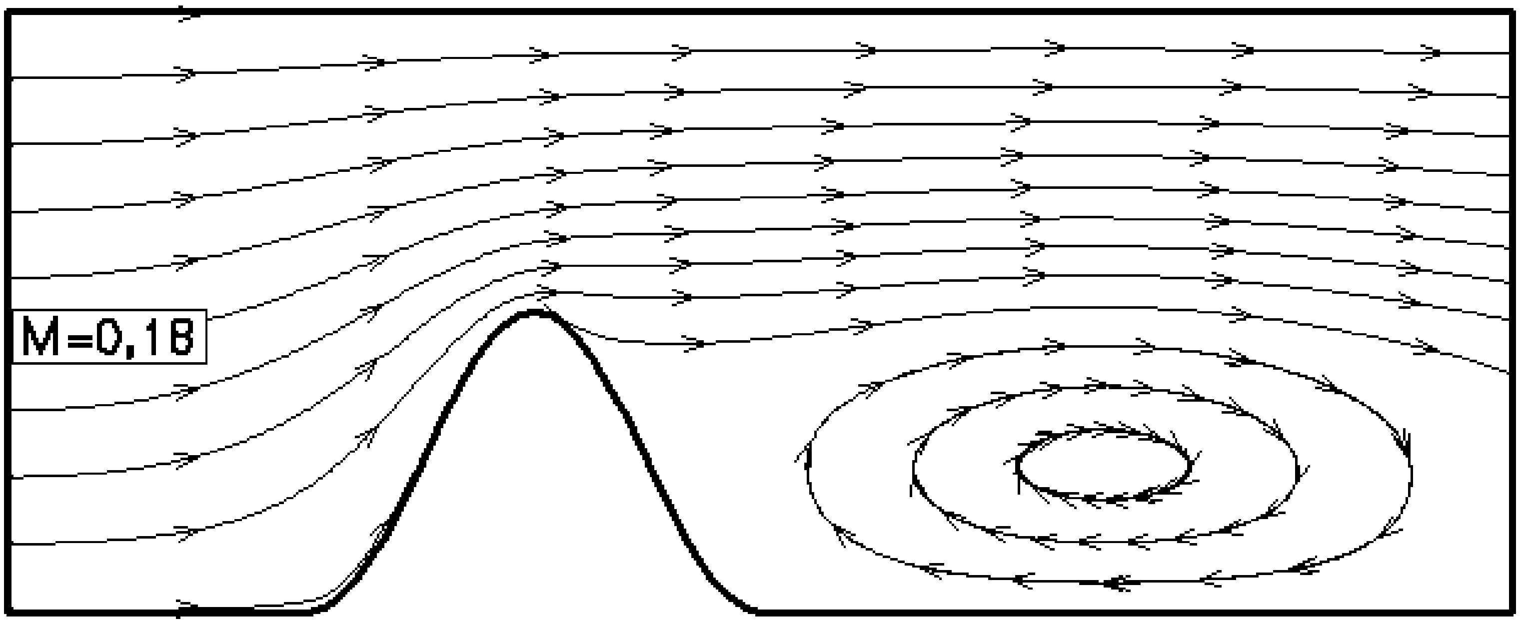
$$\% \text{ Imbalance} = \frac{\text{Flux In} - \text{Flux Out}}{\text{Maximum Flux}}$$



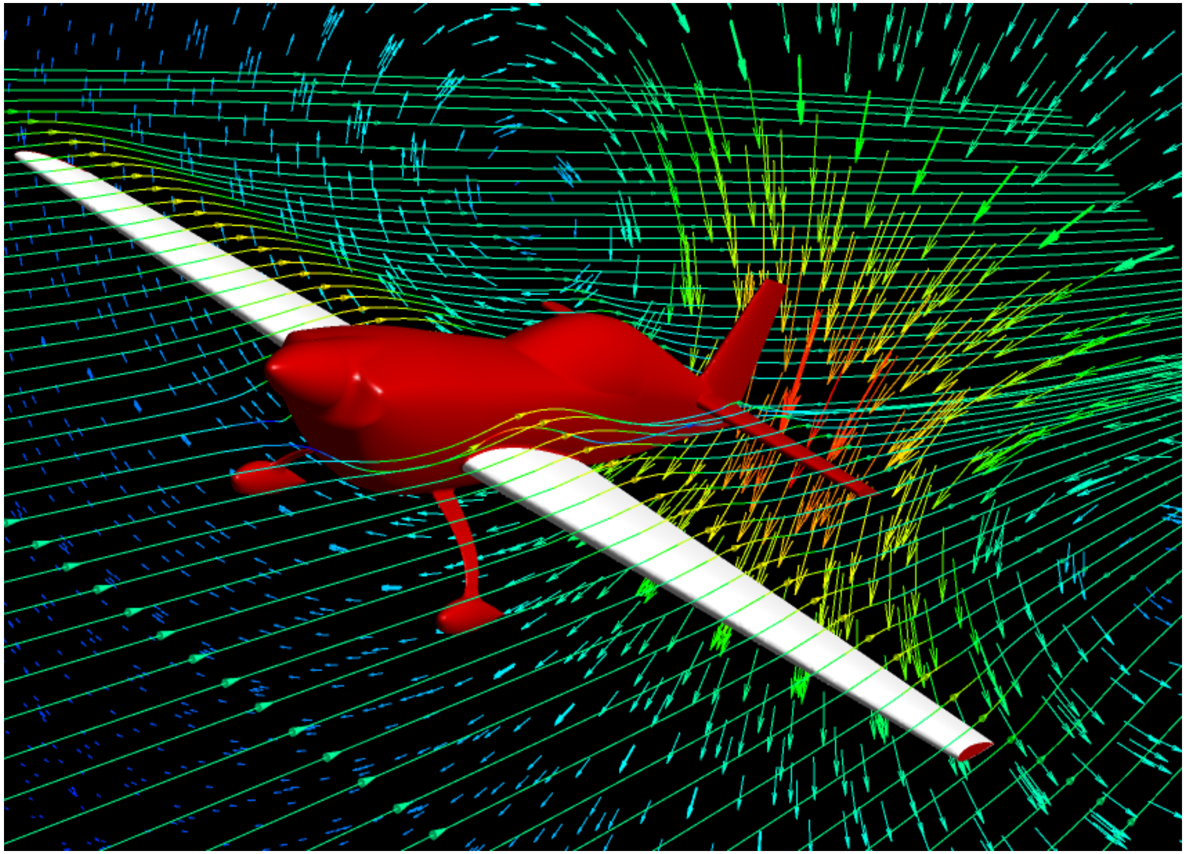
- **The imbalances measure the overall conservation of a quantity (mass, momentum, energy) in the entire flow domain**
- **Clearly in a converged solution Flux In should equal Flux Out**
- **It's good practice to set a *Conservation Target* and/or monitor the imbalances during the run**
- **When set, the Solver must meet both the *Residual* and *Conservation Target* before stopping (assuming *Max. Iterations* is not reached)**
- **Set a target of 0.01 (1%) or less**
  - Flux In – Flux Out < 1%

Forrás: Introduction to ANSYS CFX, Lecture 05 - Solver Settings and Output File - CFX-Intro_14.0_L05_SolverSettings_OutFile (2013.09.01.)

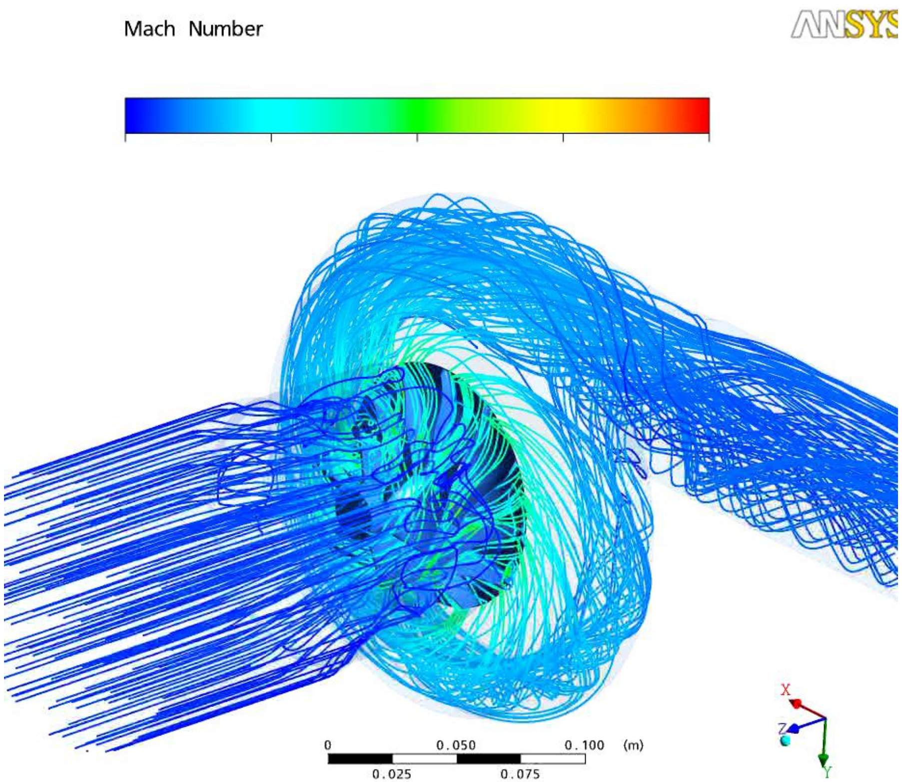
The Main Steps of a CFD Task – Visualization of the Results

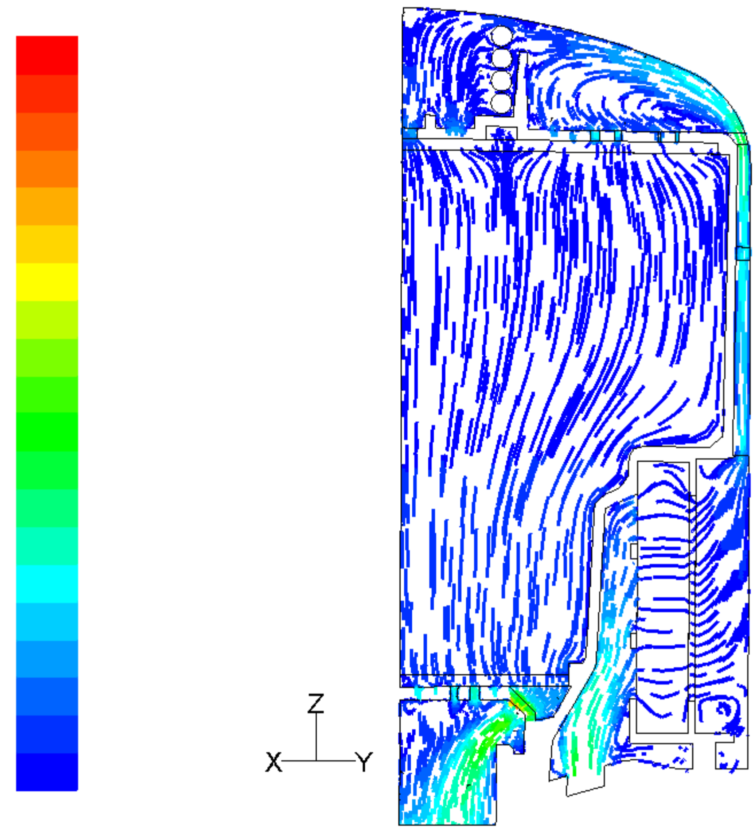


The Main Steps of a CFD Task – Visualization of the Results

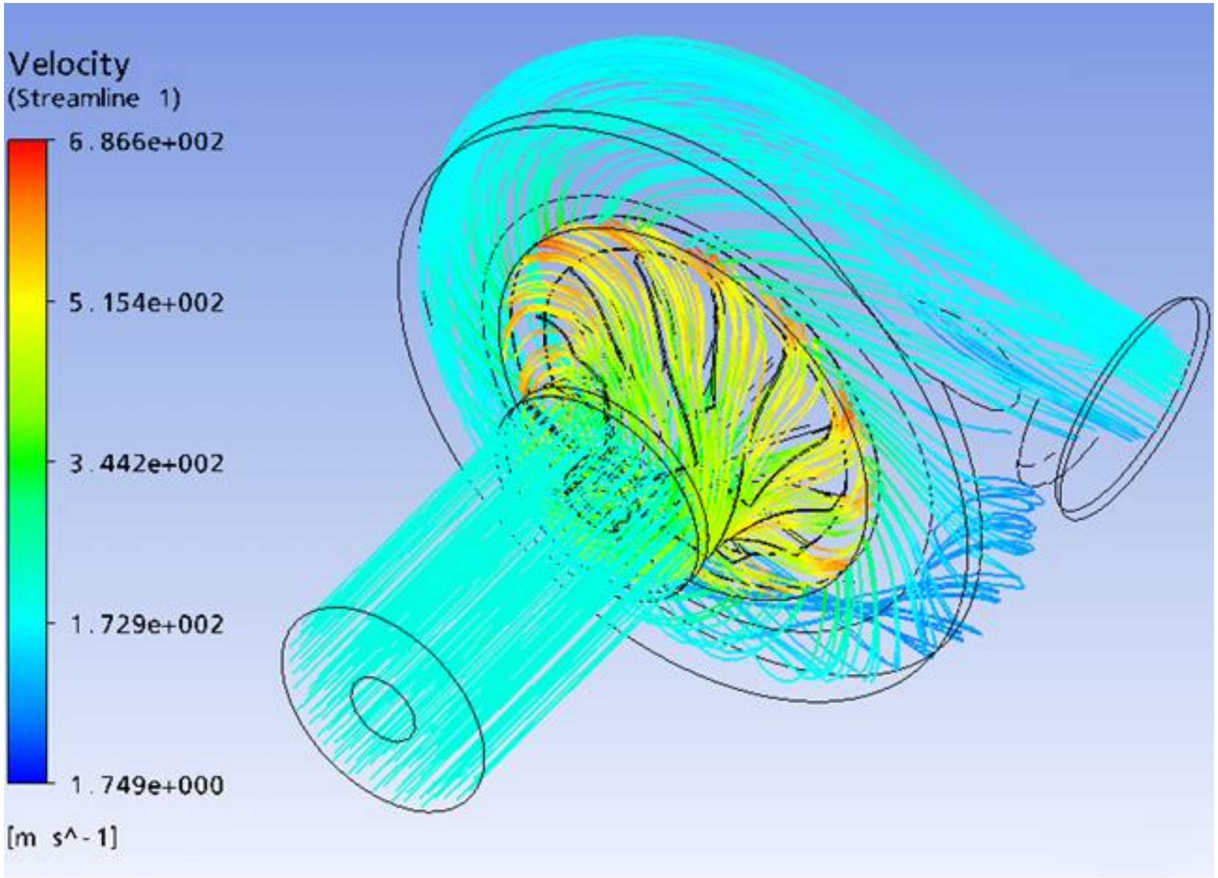


The Main Steps of a CFD Task – Visualization of the Results

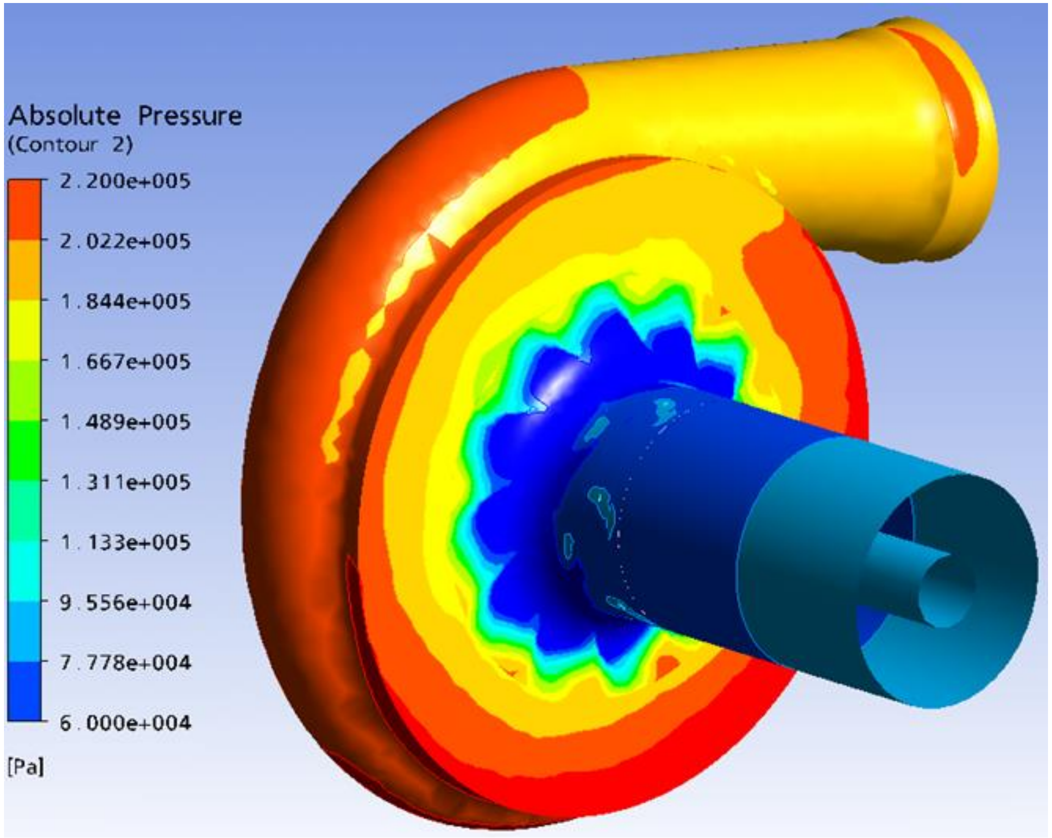




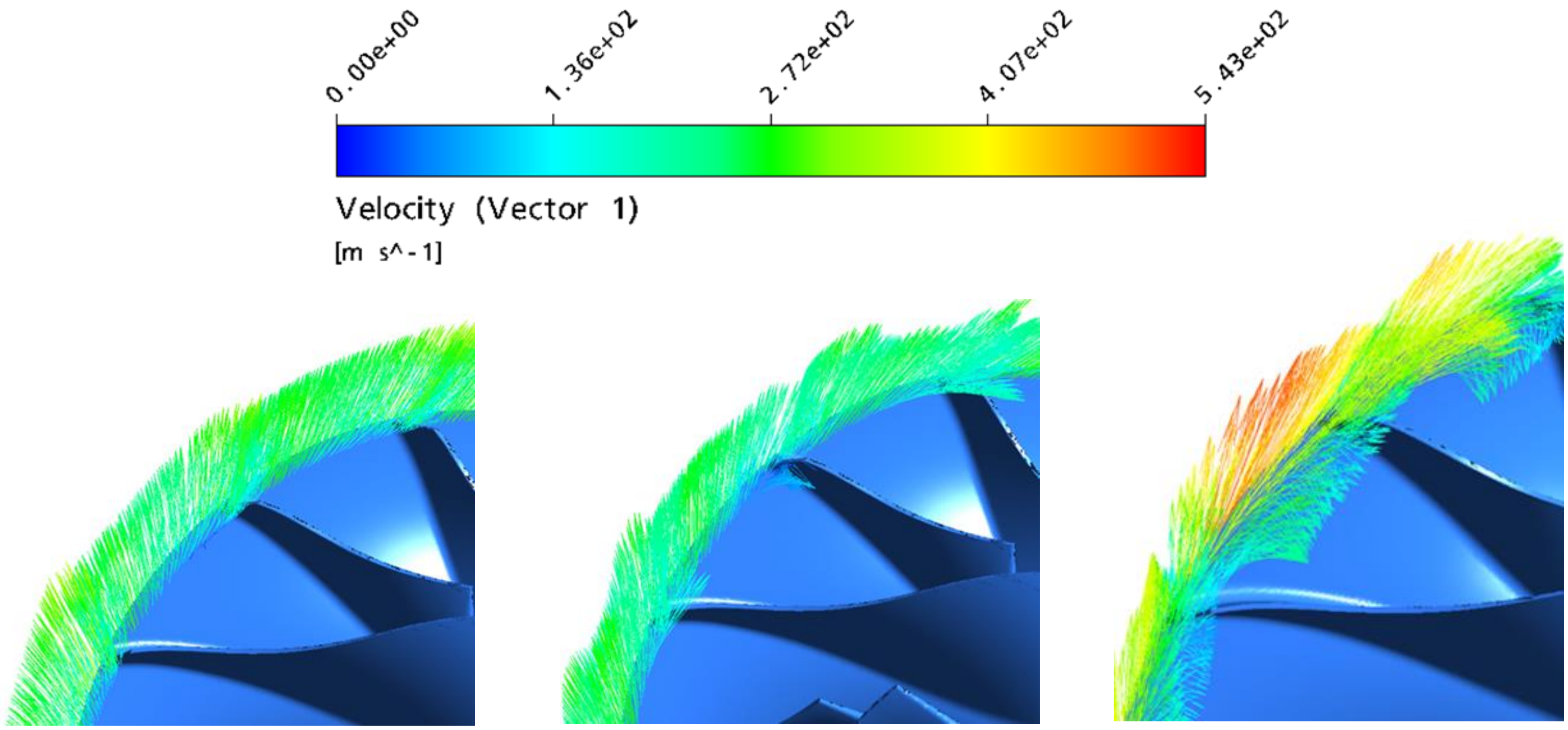
The Main Steps of a CFD Task – Visualization of the Results



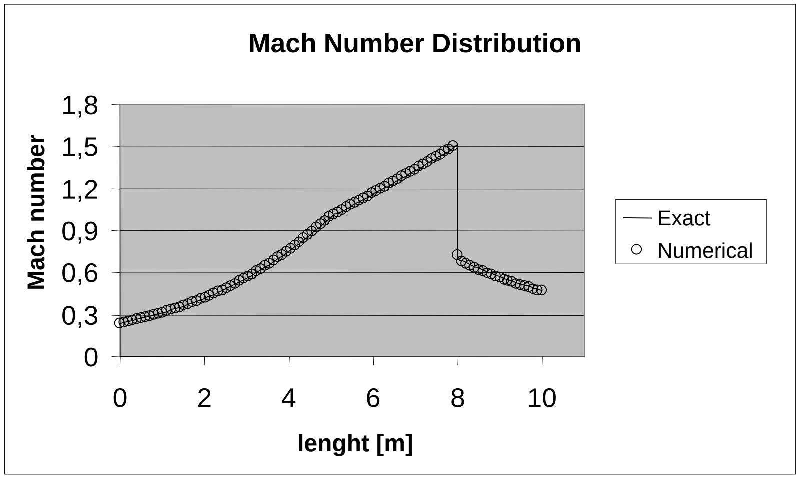
The Main Steps of a CFD Task – Visualization of the Results



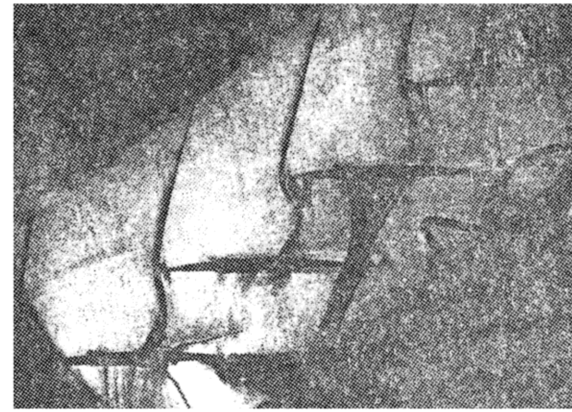
The Main Steps of a CFD Task – Visualization of the Results

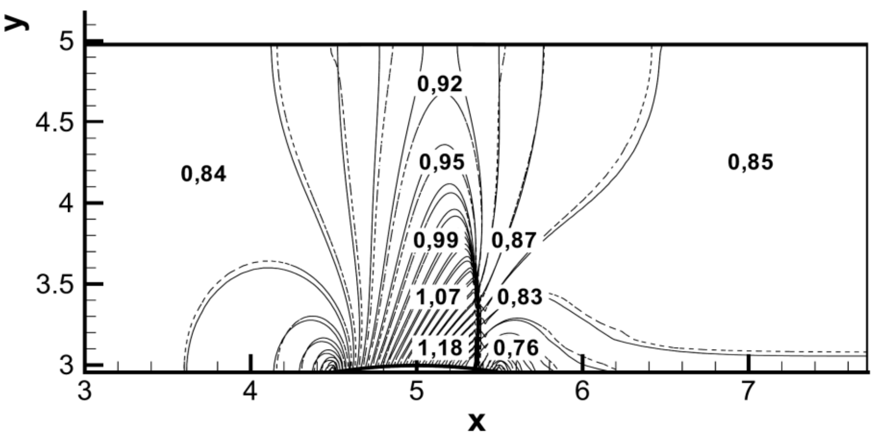


Visualization - Validation – Numerical Solutions of the Euler Equations

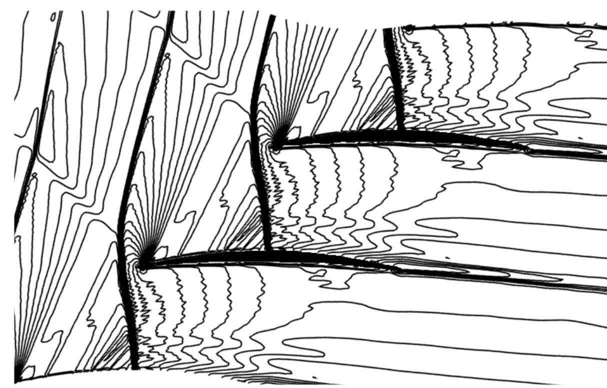


$M_1=1.1$



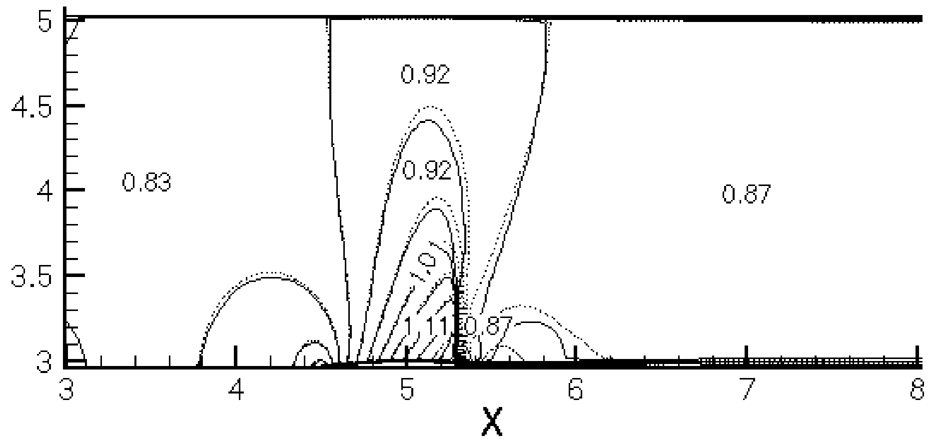


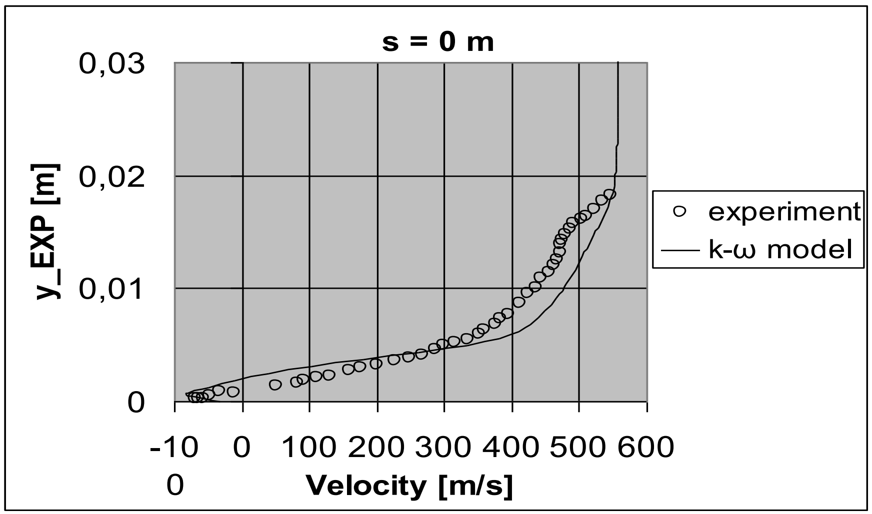
$M_1=1.1$

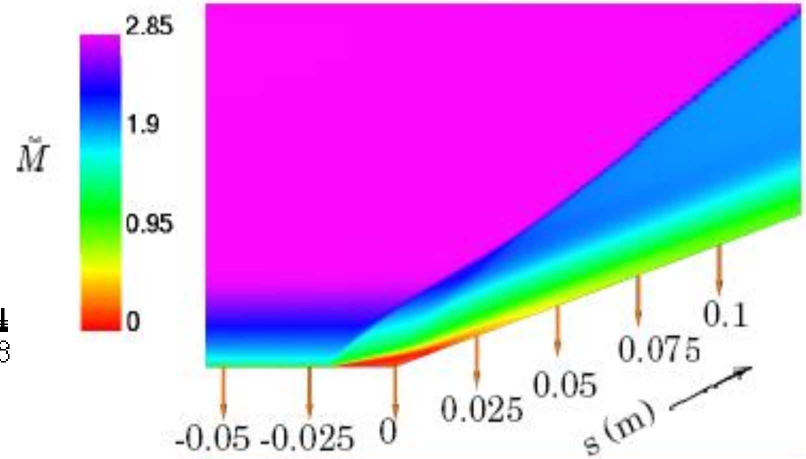


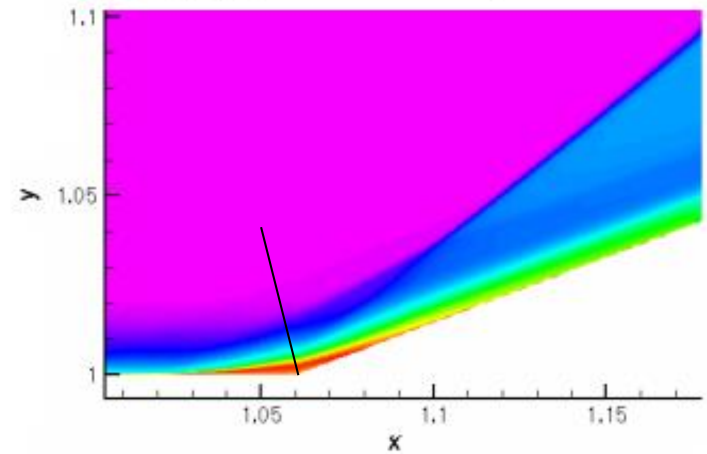
Source: Starken, H., 1971, “Untersuchung der Strömung in ebenen Überschallverzöger-ungsgittern,” DLR-Forschungsbericht 71-99.

Visualization - Validation – Numerical Solution of the Navier-Stokes Equations





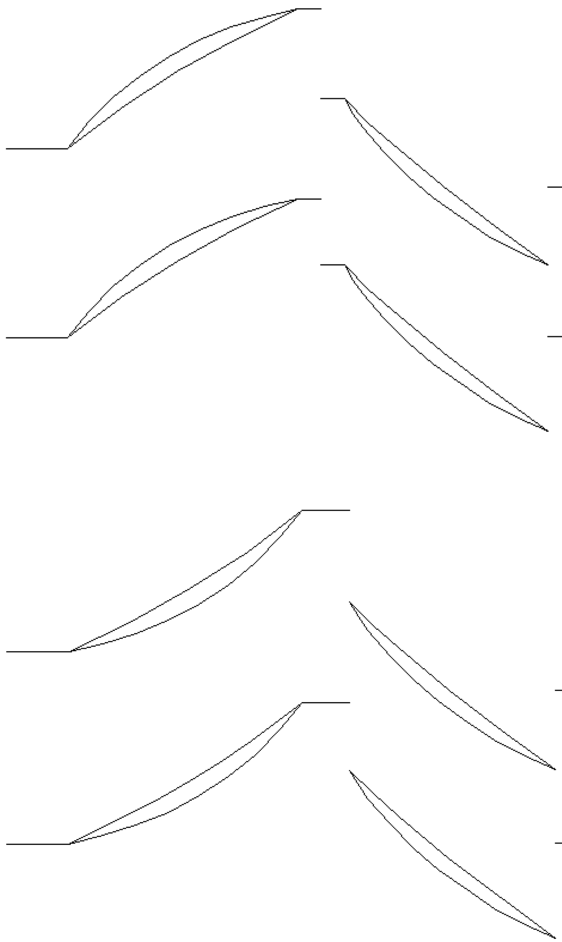


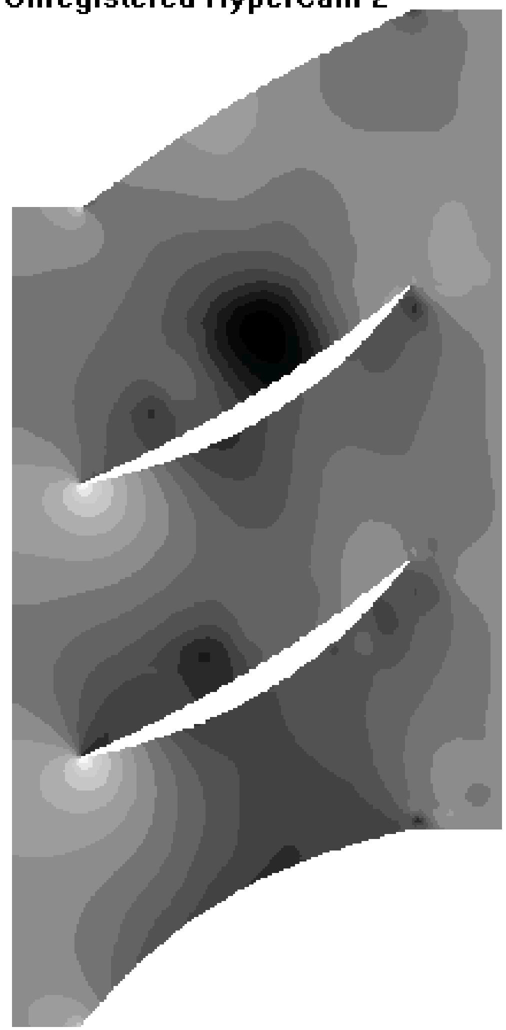


A mérés forrásanyaga: Gerolymos, G. A.; Sauret, E. & Vallet, I. (2003). *Oblique-Shock-Wave/Boundary-Layer Interaction using Near-Wall Reynolds-Stress Models*, Université Pierre-et-Marie-Curie, AIAA 2003-3466, 33rd Fluid Dynamics Conference, 23-26 June 2003 Orlando, Florida, USA

Visualization - DASFLOW Program – Numerical Solutions of the Navier-Stokes Equations in Cascade

Mach szám eloszlás

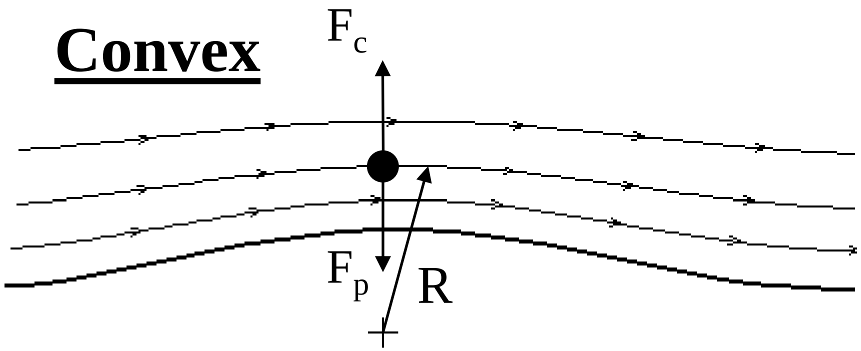




Appendices I.
The Effect of Concave and Convex Curvature

The Effect of Concave and Convex Curvature

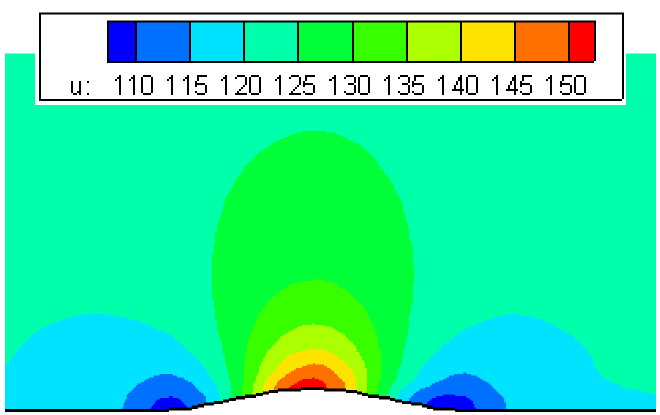
Convex



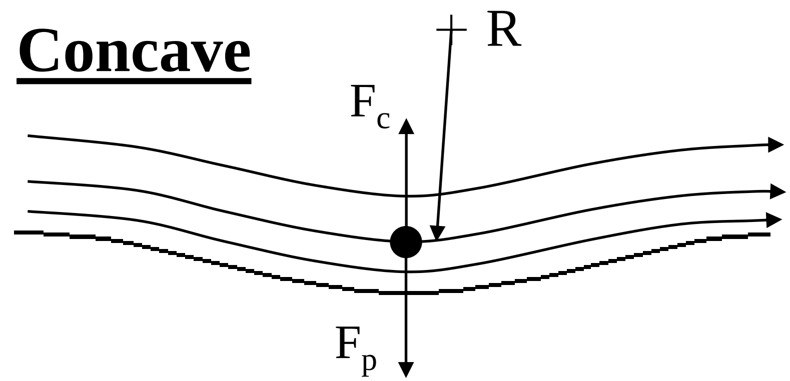
$$\text{Ha } R\downarrow \Rightarrow \; mR\omega^2,\; F_c\downarrow \Rightarrow F_p\downarrow$$

$$\text{Ha } F_p\downarrow \Rightarrow \; p\downarrow \Rightarrow v\uparrow$$

$$\frac{p}{\rho}+\frac{1}{2}v^2 \approx \acute{a}lland\acute{o}$$



Concave

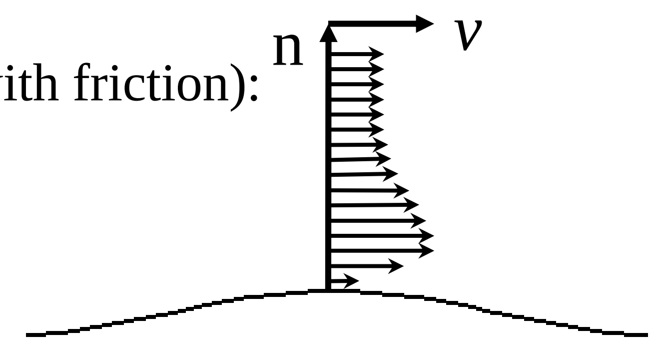


$$\text{Ha } R\uparrow \Rightarrow \; mR\omega^2,\; F_c\uparrow \Rightarrow F_p\uparrow$$

$$\text{Ha } F_p\uparrow \Rightarrow \; p\uparrow \Rightarrow v\downarrow$$

$$\frac{p}{\rho}+\frac{1}{2}v^2 \approx \acute{a}lland\acute{o}$$

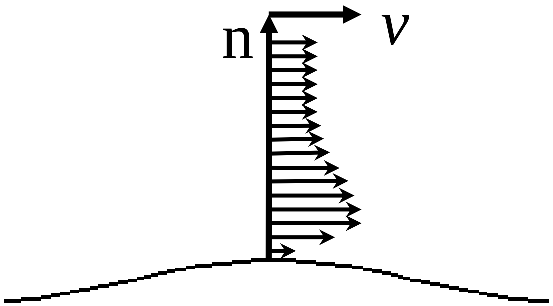
Real flow (with friction):

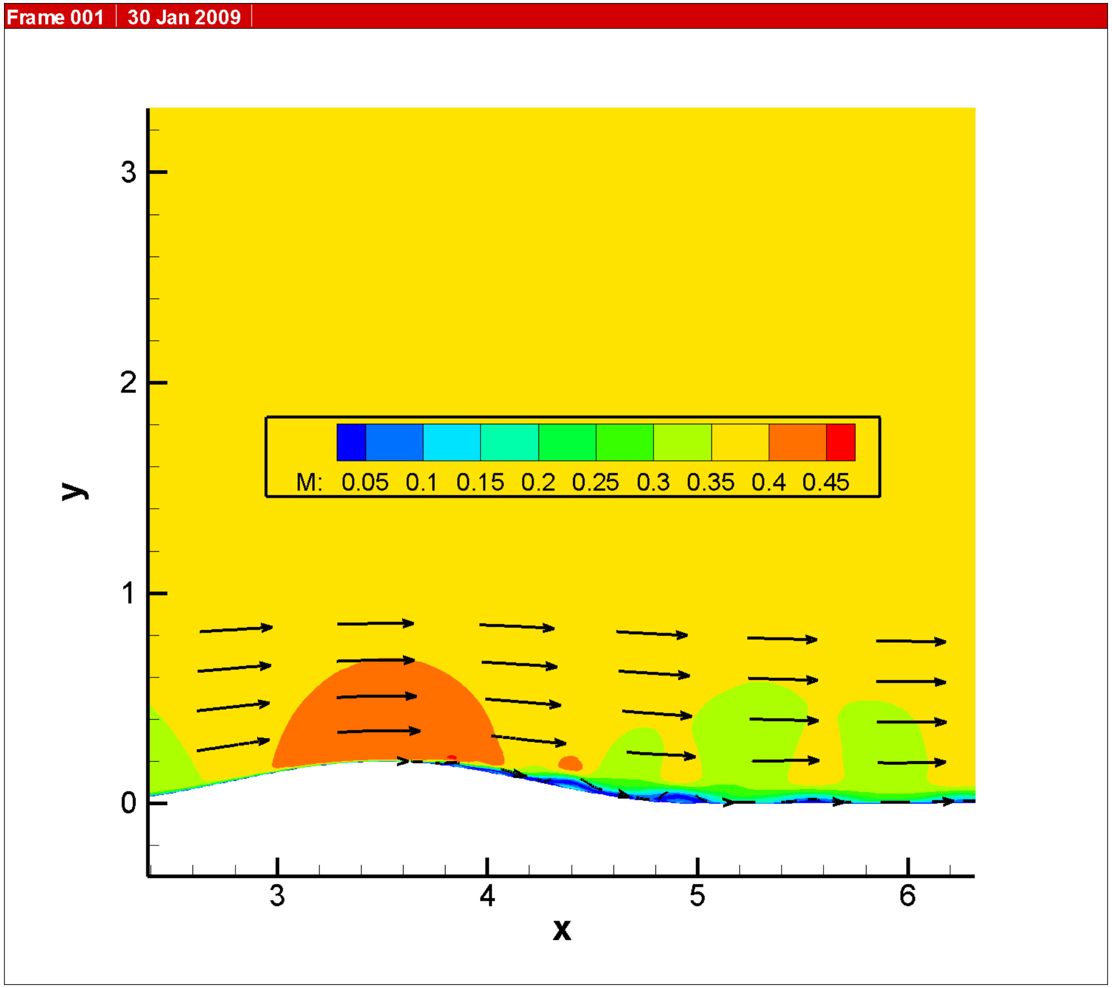


The Effect of Concave and Convex Curvature

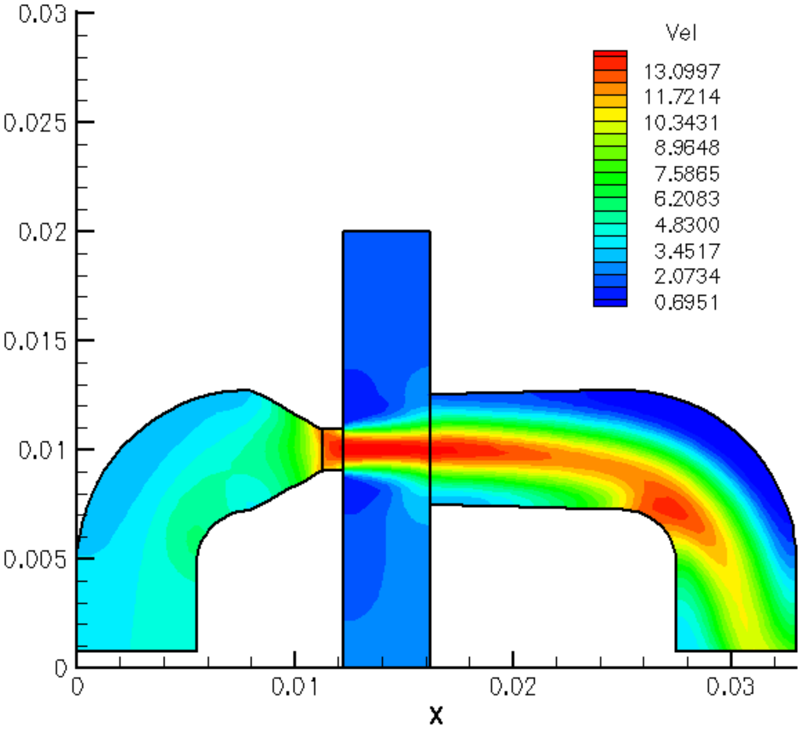
Convex

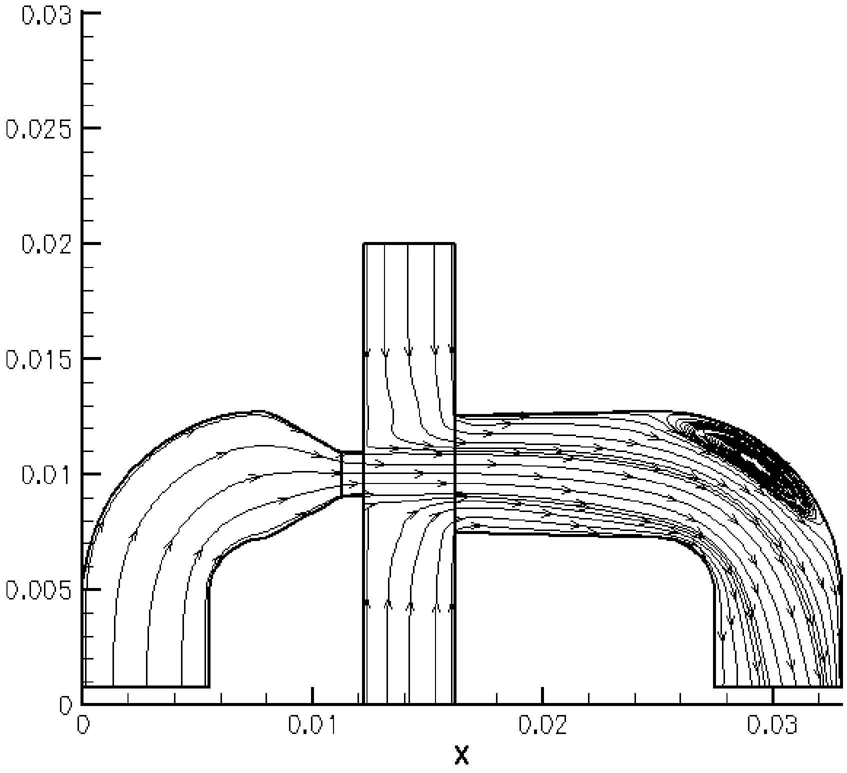
Real flow (with friction):



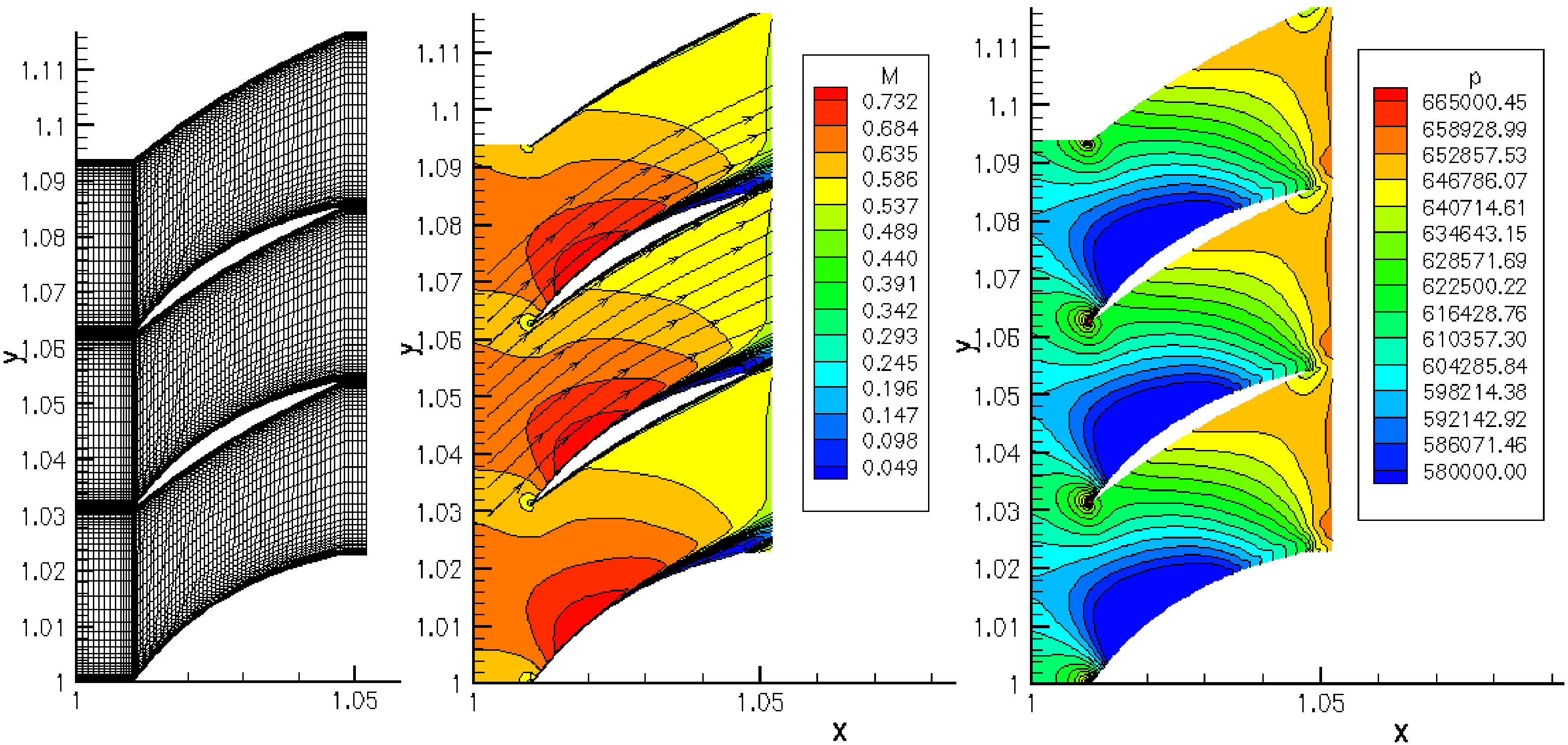


The Effect of Concave and Convex Curvature

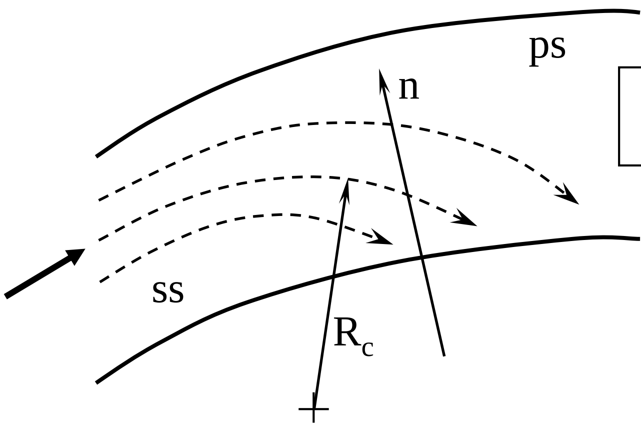


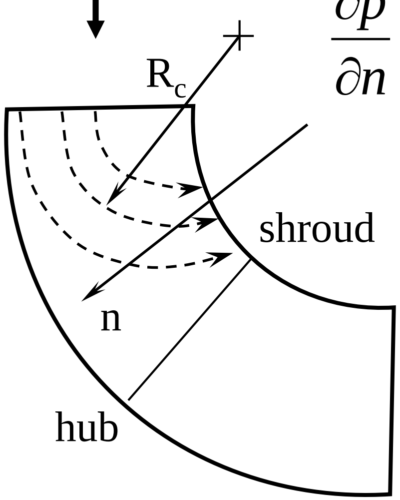


The Effect of Concave and Convex Curvature



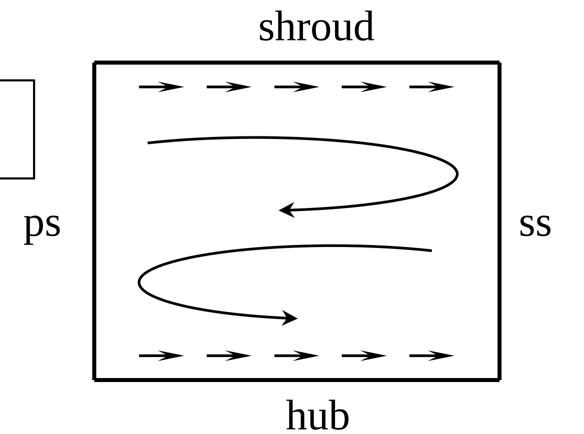
Secondary Flow in Cascades

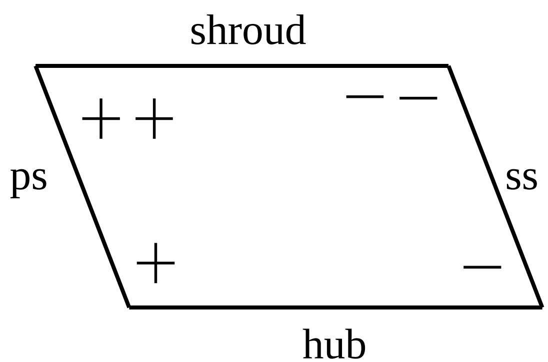




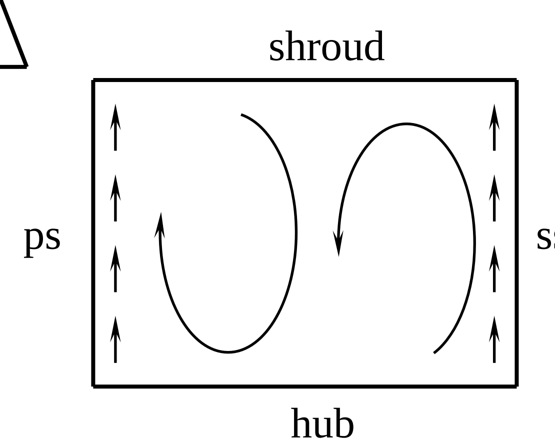
$$\frac{\partial p}{\partial n} = \frac{v^2}{R_c}$$

Passage Vortex

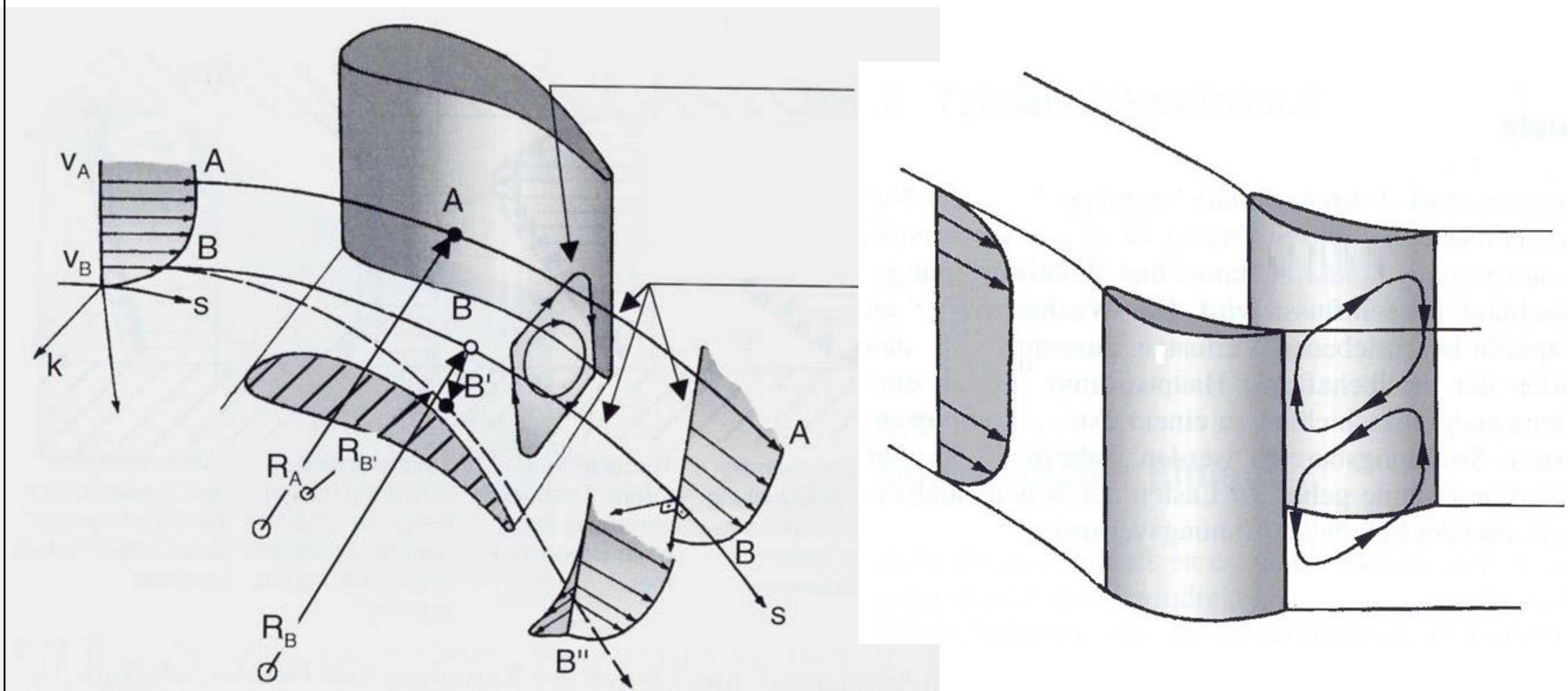




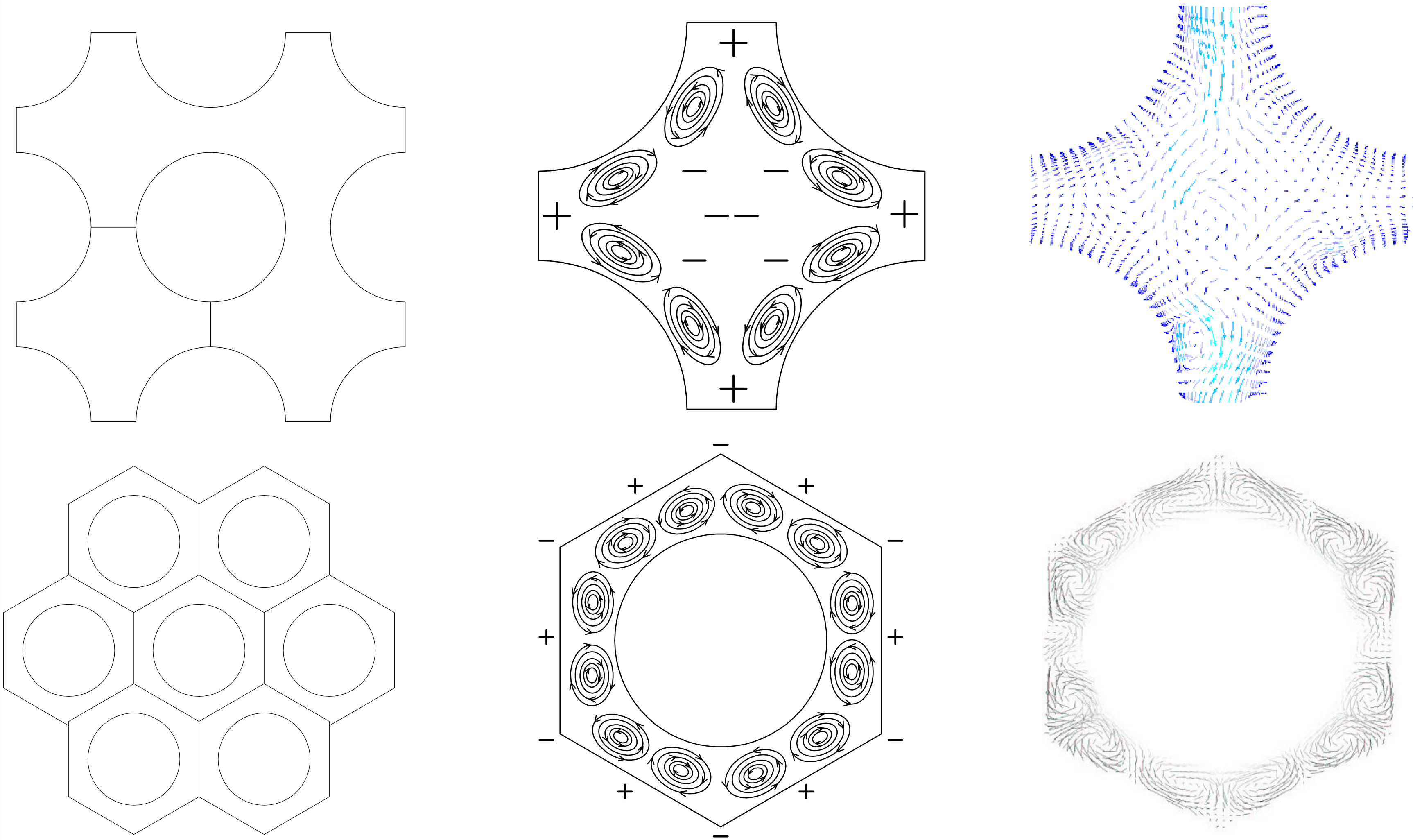
Blade Vortex



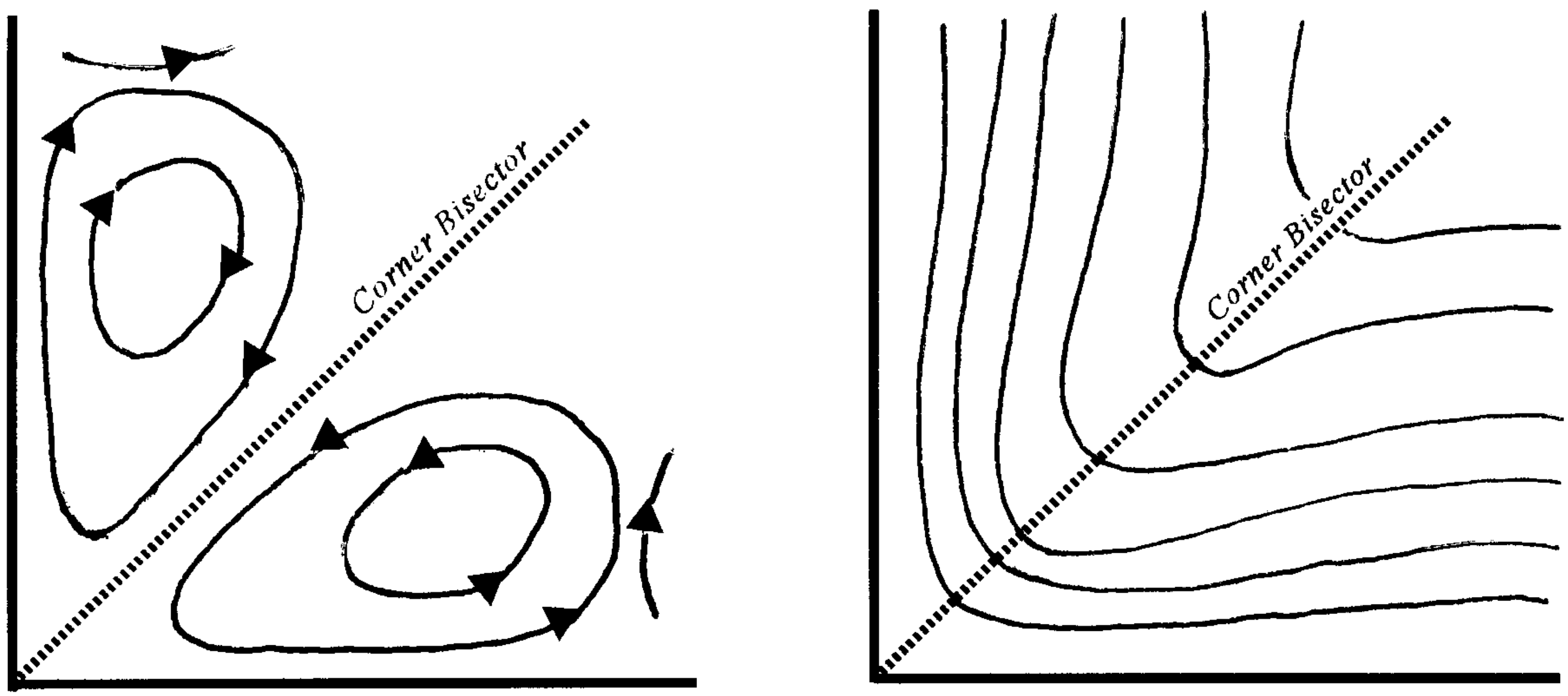
Secondary Flow in Cascades
(Passage Vortex)



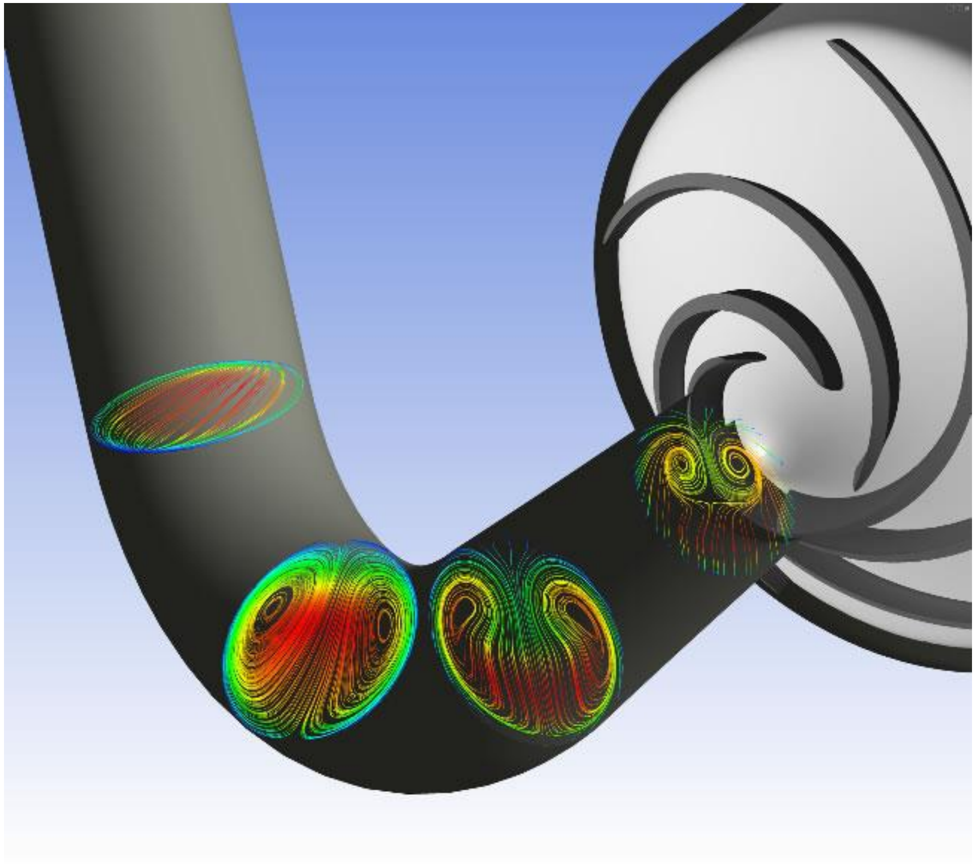
Secondary Flow Patterns in Rod Bundles in nuclear reactor



Secondary Flow Pattern in a Rectangle Cross Flow Channel

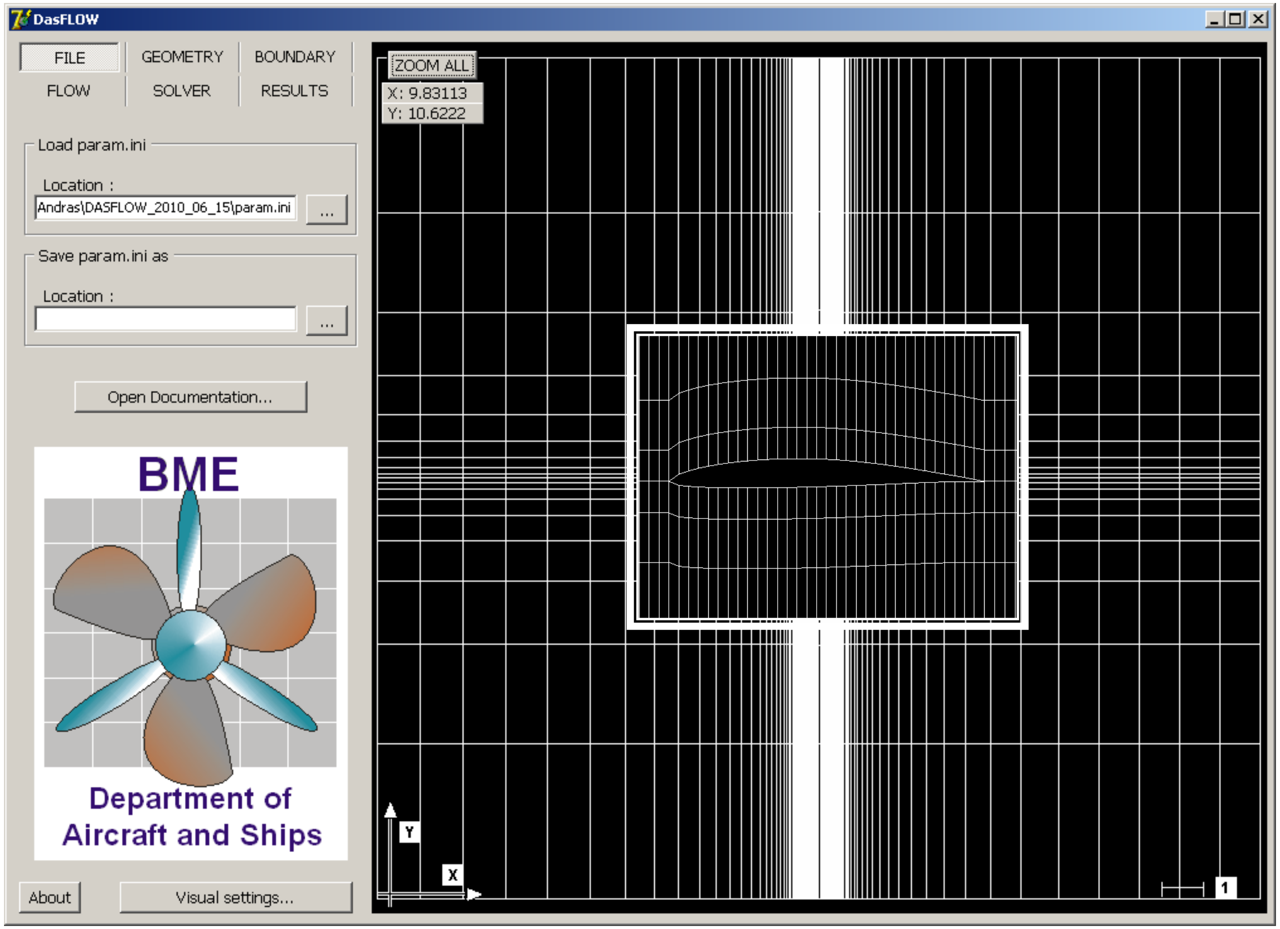


Secondary Flow Pattern in a Knee Pipe



Appendices II. DASFLOW Software and Its Industrial Applications

DASFLOW – Tanszéki fejlesztésű CFD és inverz tervezésre alkalmas program

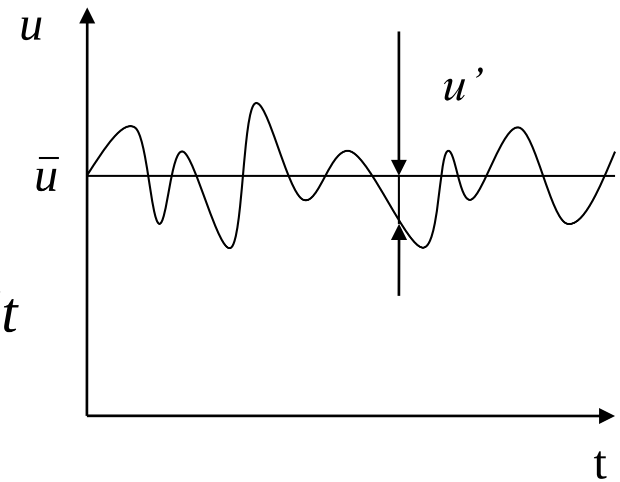


Áramlásmodellezés – Kontinuum-mechanika alapján
RANS egyenletek – Reynolds és Favre átlagolás - DASFLOW

$$u = \bar{u} + u' \quad v = \bar{v} + v'$$

$$\rho = \bar{\rho} + \rho'$$

$$w = \bar{w} + w' \quad p = \bar{p} + p'$$



$$\underbrace{\bar{u} = \frac{1}{\Delta t}\int_{t_0}^{t_0+\Delta t} u\,dt}_{\text{Reynolds Átlagolás}} \qquad \Longrightarrow \qquad \underbrace{\tilde{u} = \frac{1}{\bar{\rho}}\frac{1}{\Delta t}\int_{t_0}^{t_0+\Delta t} (\rho u)\,dt}_{\text{Favre Átlagolás}}$$

Reynolds Átlagolás

Favre Átlagolás

Nagy sebességű, összenyomható áramlás esetén.

$$u = \tilde{u} + u'' \quad v = \tilde{v} + v'' \quad w = \tilde{w} + w'' \quad p = \bar{p} + p' \quad \rho = \bar{\rho} + \rho'$$

$$h = \tilde{h} + h'' \quad e = \tilde{e} + e'' \quad T = \tilde{T} + T'' \quad q_j = \bar{q}_j + q'_j \quad \bar{q}_j = q_{L\,j}$$

Áramlásmodellezés – Kontinuum-mechanika alapján
RANS egyenletek – Reynolds és Favre átlagolás - DASFLOW

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial \bar{\rho}\tilde{u}}{\partial x} + \frac{\partial \bar{\rho}\tilde{v}}{\partial y} + \frac{\partial \bar{\rho}\tilde{w}}{\partial z} = 0$$

$$\frac{\partial \bar{\rho}\tilde{u}}{\partial t} + \frac{\partial \bar{\rho}\tilde{u}\tilde{u}}{\partial x} + \frac{\partial \bar{\rho}\tilde{v}\tilde{u}}{\partial y} + \frac{\partial \bar{\rho}\tilde{w}\tilde{u}}{\partial z} = -\frac{\partial \bar{p}}{\partial x} + \left[\frac{\partial \tau_{xx}^{F}}{\partial x} + \frac{\partial \tau_{xy}^{F}}{\partial y} + \frac{\partial \tau_{xz}^{F}}{\partial z} \right]$$

$$\frac{\partial \bar{\rho}\tilde{v}}{\partial t} + \frac{\partial \bar{\rho}\tilde{u}\tilde{v}}{\partial x} + \frac{\partial \bar{\rho}\tilde{v}\tilde{v}}{\partial y} + \frac{\partial \bar{\rho}\tilde{w}\tilde{v}}{\partial z} = -\frac{\partial \bar{p}}{\partial y} + \left[\frac{\partial \tau_{yx}^{F}}{\partial x} + \frac{\partial \tau_{yy}^{F}}{\partial y} + \frac{\partial \tau_{yz}^{F}}{\partial z} \right]$$

$$\frac{\partial \bar{\rho}\tilde{w}}{\partial t} + \frac{\partial \bar{\rho}\tilde{u}\tilde{w}}{\partial x} + \frac{\partial \bar{\rho}\tilde{v}\tilde{w}}{\partial y} + \frac{\partial \bar{\rho}\tilde{w}\tilde{w}}{\partial z} = -\frac{\partial \bar{p}}{\partial z} + \left[\frac{\partial \tau_{zx}^{F}}{\partial x} + \frac{\partial \tau_{zy}^{F}}{\partial y} + \frac{\partial \tau_{zz}^{F}}{\partial z} \right]$$

Áramlásmodellezés – Kontinuum-mechanika alapján
RANS egyenletek – Reynolds és Favre átlagolás - DASFLOW

$$\tau_{xx}^{F} = 2\mu\frac{\partial\tilde{u}}{\partial x} - \frac{2}{3}\mu\nabla^{T}\tilde{\vec{V}} - \overline{\rho u''u''} \qquad \tau_{yy}^{F} = 2\mu\frac{\partial\tilde{v}}{\partial y} - \frac{2}{3}\mu\nabla^{T}\tilde{\vec{V}} - \overline{\rho v''v''}$$

$$\tau_{zz}^{F} = 2\mu\frac{\partial\tilde{w}}{\partial z} - \frac{2}{3}\mu\nabla^{T}\tilde{\vec{V}} - \overline{\rho w''w''} \qquad \tau_{xy}^{F} = \tau_{yx}^{F} = \mu\left(\frac{\partial\tilde{u}}{\partial y} + \frac{\partial\tilde{v}}{\partial x}\right) - \overline{\rho u''v''}$$

$$\tau_{xz}^{F} = \tau_{zx}^{F} = \mu\left(\frac{\partial\tilde{u}}{\partial z} + \frac{\partial\tilde{w}}{\partial x}\right) - \overline{\rho u''w''} \qquad \tau_{yz}^{F} = \tau_{zy}^{F} = \mu\left(\frac{\partial\tilde{v}}{\partial z} + \frac{\partial\tilde{w}}{\partial y}\right) - \overline{\rho v''w''}$$

$$\frac{\partial}{\partial t}\left[\overline{\rho}\left(\tilde{e} + \frac{1}{2}\sum_{i=1}^{3}\tilde{u}_i\tilde{u}_i\right) + \frac{1}{2}\sum_{i=1}^{3}\overline{\rho u_i''u_i''}\right] + \sum_{j=1}^{3}\frac{\partial}{\partial x_j}\left[\overline{\rho}\tilde{u}_j\left(\tilde{h} + \frac{1}{2}\sum_{i=1}^{3}\tilde{u}_i\tilde{u}_i\right) + \tilde{u}_j\,\frac{1}{2}\sum_{i=1}^{3}\overline{\rho u_i''u_i''}\right] =$$

$$= \sum_{j=1}^{3}\frac{\partial}{\partial x_j}\left[-\overline{q}_j - \overline{\rho u_j''h''} + \sum_{i=1}^{3}\overline{\tau_{ji}u_i''} - \frac{1}{2}\sum_{i=1}^{3}\overline{\rho u_j''u_i''u_i''}\right] + \sum_{j=1}^{3}\frac{\partial}{\partial x_j}\left[\sum_{i=1}^{3}\left[\tilde{u}_i\left(\overline{\tau}_{ji} - \overline{\rho u_i''u_j''}\right)\right]\right]$$

Áramlásmodellezés – Kontinuum-mechanika alapján
RANS egyenletek – Reynolds és Favre átlagolás - DASFLOW

$$\bar{\rho}\frac{\partial k}{\partial t}+\bar{\rho}\tilde{u}_j\frac{\partial k}{\partial x_j}=-\overline{\rho u_i'' u_j''}\frac{\partial \tilde{u}_i}{\partial x_j}-\beta^*\bar{\rho}k\omega+\frac{\partial}{\partial x_j}\left[\left(\mu+\sigma^*\mu_t\right)\frac{\partial k}{\partial x_j}\right]$$

$$\bar{\rho}\frac{\partial \omega}{\partial t}+\bar{\rho}\tilde{u}_j\frac{\partial \omega}{\partial x_j}=-\overline{\rho u_i'' u_j''}\alpha\frac{\omega}{k}\frac{\partial \tilde{u}_i}{\partial x_j}-\beta\bar{\rho}\omega^2+\frac{\partial}{\partial x_j}\left[\left(\mu+\sigma\mu_t\right)\frac{\partial \omega}{\partial x_j}\right]$$

$$\mu_t=\bar{\rho}\frac{k}{\omega}\quad \alpha=\frac{13}{25}\quad \sigma^*=\frac{1}{2}\quad \sigma=\frac{1}{2}\quad \beta^*=\beta_0^* f_{\beta^*}\left[1+\xi^* F\left(M_t\right)\right]\quad \beta_0=\frac{9}{125}\quad \beta_0^*=\frac{9}{100}$$

$$\beta=\beta_0 f_\beta-\beta_0^* f_{\beta^*}\xi^* F\left(M_t\right)\quad f_{\beta^*}=\begin{cases}1 & if \quad \chi_k\le 0\\ \dfrac{1+680\chi_k^2}{1+400\chi_k^2} & if \quad \chi_k>0\end{cases}\quad \chi_k=\frac{1}{\omega^3}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j}$$

$$\chi_\omega=\left|\frac{\Omega_{ij}\Omega_{jk}S_{ki}}{\left(\beta_0^*\omega\right)^3}\right|\quad f_\beta=\frac{1+70\chi_\omega}{1+80\chi_\omega}\quad \xi^*=\frac{3}{2}\quad \chi_\omega=0\quad M_{t0}=\frac{1}{4}\quad \varepsilon=\beta^*\omega k\quad l=\frac{k^{1/2}}{\omega}$$

$$F\left(M_t\right)=\left[M_t^2-M_{t0}^2\right]\mathrm{H}\left(M_t-M_{t0}\right)$$

$$\mathrm{H}(x)=\begin{cases}0 & if \quad x\le 0\\ 1 & if \quad x>0\end{cases}\quad S_{ij}=\frac{1}{2}\left(\frac{\partial \tilde{u}_i}{\partial x_j}+\frac{\partial \tilde{u}_j}{\partial x_i}\right)$$

$$M_t^2=\frac{2k}{a^2}\quad \Omega_{ij}=\frac{1}{2}\left(\frac{\partial \tilde{u}_i}{\partial x_j}-\frac{\partial \tilde{u}_j}{\partial x_i}\right)$$

Áramlásmodellezés – Kontinuum-mechanika alapján RANS egyenletek – Reynolds és Favre átlagolás – Konzervatív összevont forma - DASFLOW

$$\frac{\partial}{\partial t}\iint_A U dA + \oint_\Gamma [H_n(U)] d\Gamma = \oint_\Gamma [H_{vn}(U)] d\Gamma + \iint_A [S(U)] dA$$

$$U = \begin{pmatrix} \bar\rho \\ \bar\rho\tilde{u} \\ \bar\rho\tilde{v} \\ \bar\rho\tilde{E} \\ \bar\rho k \\ \bar\rho\omega \end{pmatrix} \qquad H_n(U) = \begin{pmatrix} \bar\rho V_n \\ \bar\rho\tilde{u} V_n + p^* n_x \\ \bar\rho\tilde{v} V_n + p^* n_y \\ \left(\bar\rho\tilde{E} + p^*\right) V_n \\ \bar\rho V_n k \\ \bar\rho V_n \omega \end{pmatrix} \qquad S(U) = \begin{pmatrix} 0 \\ 0 \\ 0 \\ 0 \\ S_k \\ S_\omega \end{pmatrix}$$

Áramlásmodellezés – Kontinuum-mechanika alapján
RANS egyenletek – Konzervatív forma – RARS - DASFLOW

$$H_n(U) = \tilde{H}_n\left(U^L, U^R\right) = \frac{1}{2}\left\{H_n\left(U^L\right) + H_n\left(U^R\right) - \left|\hat{D}_n\left(U^L, U^R\right)\right|\left(U^R - U^L\right)\right\}$$

$$\left|\hat{D}_n\right| \Delta U = \sum_{i=1}^{4} \left|\hat{\lambda}_n^i\right| \hat{r}_n^i \Delta W_n^i$$

$$\hat{\Lambda} = \begin{bmatrix} \hat{\vec{V}}\vec{n} \\ \hat{\vec{V}}\vec{n} \\ \hat{\vec{V}}\vec{n} + \hat{c} \\ \hat{\vec{V}}\vec{n} - \hat{c} \end{bmatrix} \qquad \Delta W_n = \begin{bmatrix} \Delta\rho - \dfrac{\Delta p}{\hat{c}^2} \\ \vec{s}\,\Delta\vec{V} \\ \vec{n}\Delta\vec{V} + \dfrac{\Delta p}{\hat{\rho}\hat{c}} \\ -\vec{n}\Delta\vec{V} + \dfrac{\Delta p}{\hat{\rho}\hat{c}} \end{bmatrix}$$

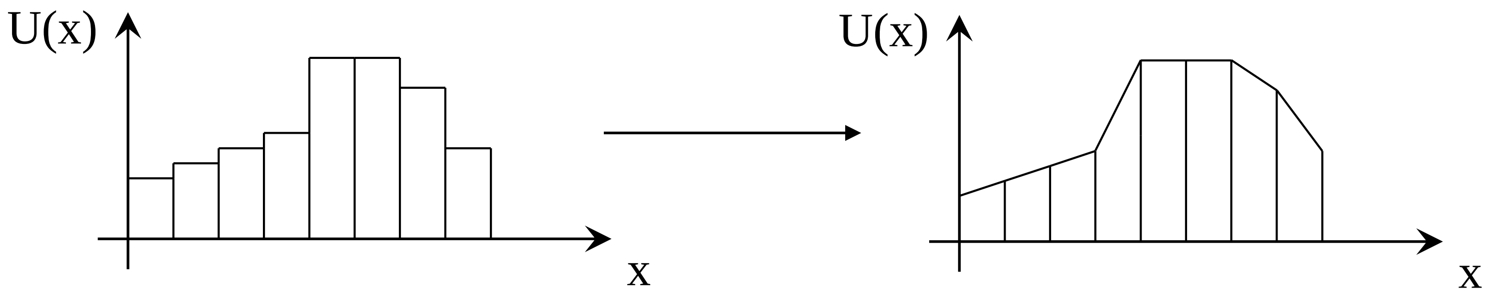
$$\hat{r}_n^1 = \left[1, \hat{u}, \hat{v}, 0.5\left(\hat{u}^2 + \hat{v}^2\right)\right]^T$$

$$\hat{r}_n^2 = \left[0, \hat{\rho} n_y, -\hat{\rho} n_x, \rho(\hat{u} n_y - \hat{v} n_x)\right]^T$$

$$\hat{r}_n^3 = \left[\frac{\hat{\rho}}{2\hat{c}}, \frac{\hat{\rho}}{2\hat{c}}\left(\hat{u} + \hat{c} n_x\right), \frac{\hat{\rho}}{2\hat{c}}\left(\hat{v} + \hat{c} n_y\right), \frac{\hat{\rho}}{2\hat{c}}\left(\alpha + \frac{\hat{c}^2}{\beta} + \hat{c}\hat{V}_n\right)\right]^T$$

$$\hat{r}_n^4 = \left[\frac{\hat{\rho}}{2\hat{c}}, \frac{\hat{\rho}}{2\hat{c}}\left(\hat{u} - \hat{c} n_x\right), \frac{\hat{\rho}}{2\hat{c}}\left(\hat{v} - \hat{c} n_y\right), \frac{\hat{\rho}}{2\hat{c}}\left(\alpha + \frac{\hat{c}^2}{\beta} - \hat{c}\hat{V}_n\right)\right]^T$$

Áramlásmodellezés – Kontinuum-mechanika alapján RANS egyenletek – Konzervatív forma – MUSCL - DASFLOW



$$U(x) = U_i + \frac{\partial U}{\partial x}\Big|_{x_i}(x - x_i) + \frac{1}{2}\frac{\partial^2 U}{\partial x^2}\Big|_{x_i}(x - x_i)^2 + O(\Delta x^3)$$

$$\overline{U}^{L}_{i+1/2} = U_i + \frac{1}{4}\left[(1+\kappa)\Delta_{i+1/2} + (1-\kappa)\Delta_{i-1/2}\right]$$

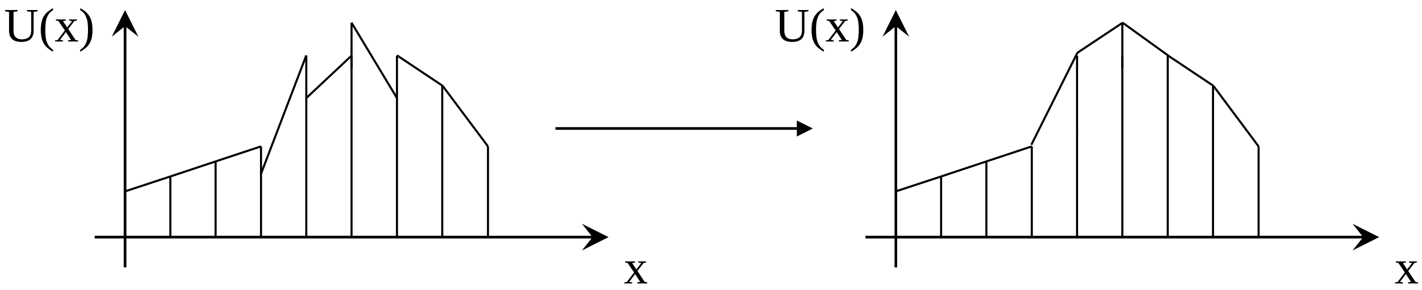
$$\kappa = 1/3$$

$$\overline{U}^{R}_{i-1/2} = U_i - \frac{1}{4}\left[(1-\kappa)\Delta_{i+1/2} + (1+\kappa)\Delta_{i-1/2}\right]$$

$$\kappa = [-1,1]$$

$$\Delta_{i+1/2} = U_{i+1} - U_i$$

Áramlásmodellezés – Kontinuum-mechanika alapján
RANS egyenletek – Konzervatív forma – MUSCL (limiterek) - DASFLOW



$$\overline{U}^{L}_{i+1/2} = U_i + \frac{r^i}{4}\left[\left(1+\kappa r^i\right)\Delta_{i+1/2} + \left(1-\kappa r^i\right)\Delta_{i-1/2}\right]$$

$$\overline{U}^{R}_{i-1/2} = U_i - \frac{r^i}{4}\left[\left(1-\kappa r^i\right)\Delta_{i+1/2} + \left(1+\kappa r^i\right)\Delta_{i-1/2}\right]$$

$$r^i = \frac{2\Delta_{i+1/2}\Delta_{i-1/2} + 2\varepsilon^2}{\Delta^2_{i+1/2} + \Delta^2_{i-1/2} + 2\varepsilon^2}$$

$$\kappa = 1/3$$

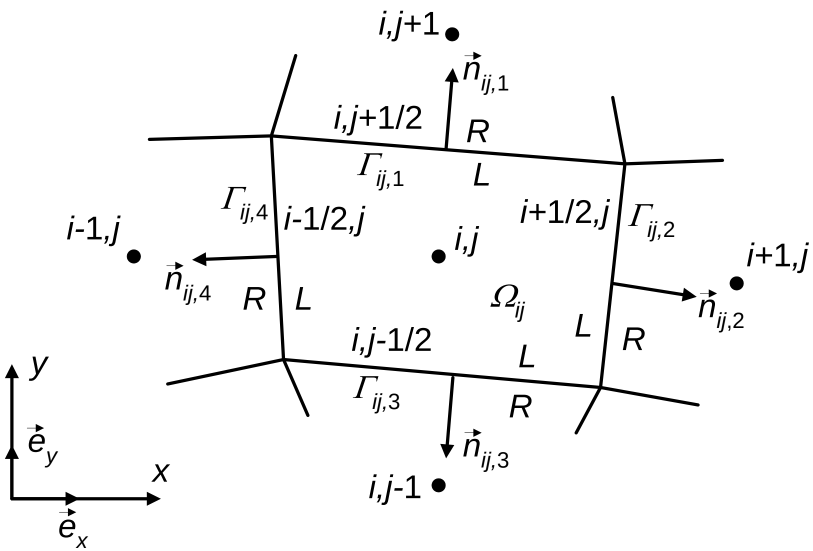
$$\varepsilon \approx 10^{-7}$$

Áramlásmodellezés – Kontinuum-mechanika alapján
RANS egyenletek – Diszkretizáció - DASFLOW

$$\oint_{\Gamma_{ij}} \left[H_{vn}(U)\right] d\Gamma = \sum_{k=1}^{4} \left(\left[H_{vn}\right]_{ij,k} \Gamma_{ij,k} \right)$$

$$H_{vn} = \frac{1}{2}\left[H_{vn}\left(U^L\right) + H_{vn}\left(U^R\right)\right]$$

$$\iint_{A_{ij}} \left[S(U)\right] dA = \left[S(U)\right]_{ij} A_{ij}$$



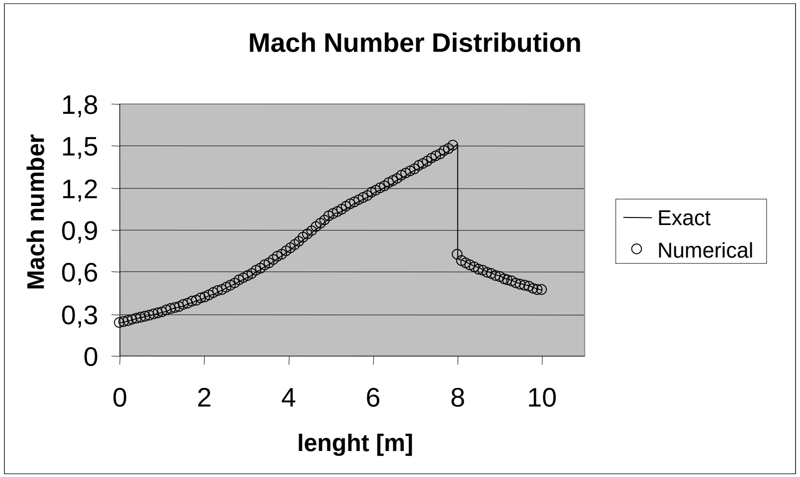
$$\frac{\partial}{\partial t} U_{ij} = -\frac{1}{A_{ij}} \left(\sum_{k=1}^{4} \left(\left[H_n\right]_{ij,k} \Gamma_{ij,k} \right) - \sum_{k=1}^{4} \left(\left[H_{vn}\right]_{ij,k} \Gamma_{ij,k} \right) \right) + \left[S(U)\right]_{ij} = \Re$$

$$\left.\begin{aligned} U^{0} &= U^{n} \\ U^{k} &= U^{0} + \alpha_k \Delta t \Re\left(U^{k-1}\right) \qquad k = 1,m \\ U^{n+1} &= U^{m} \end{aligned}\right\}$$

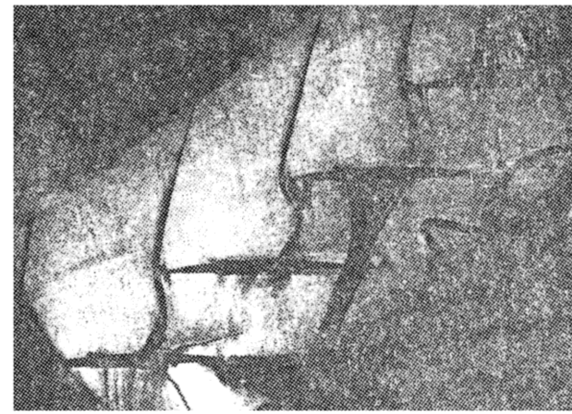
- Runge-Kutta módszer

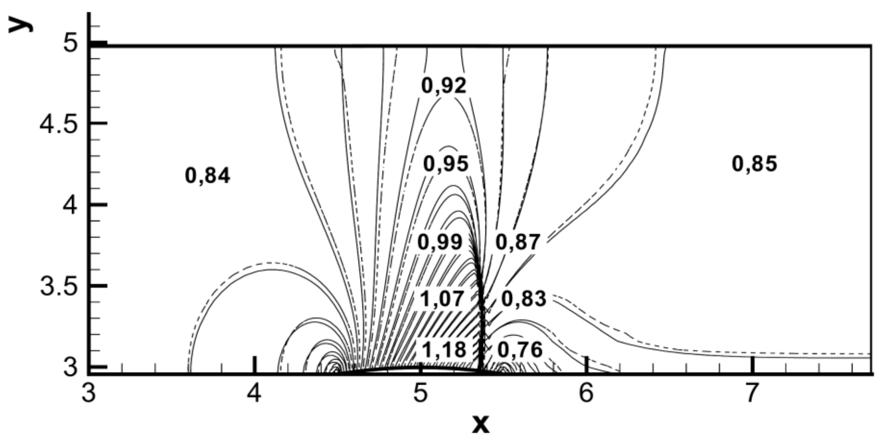
  m=4

DASFLOW Program – Egyenletek – Validáció (súrlódásmentes)

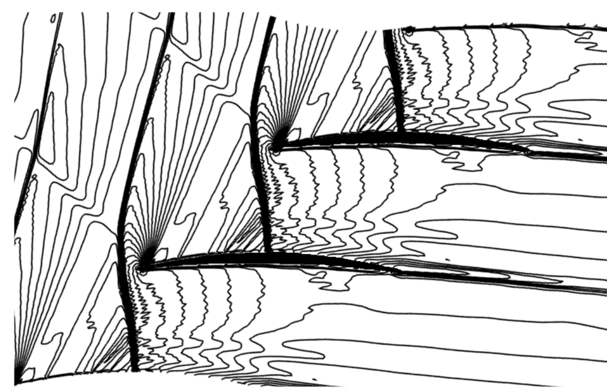


M_1=1.1

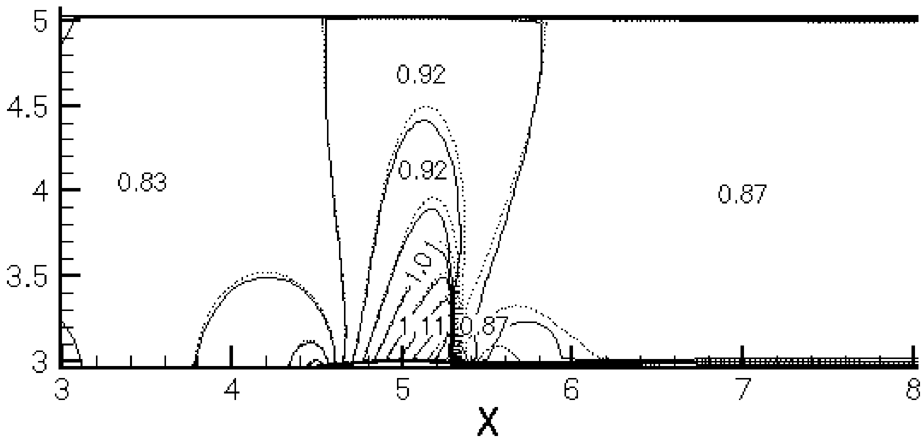


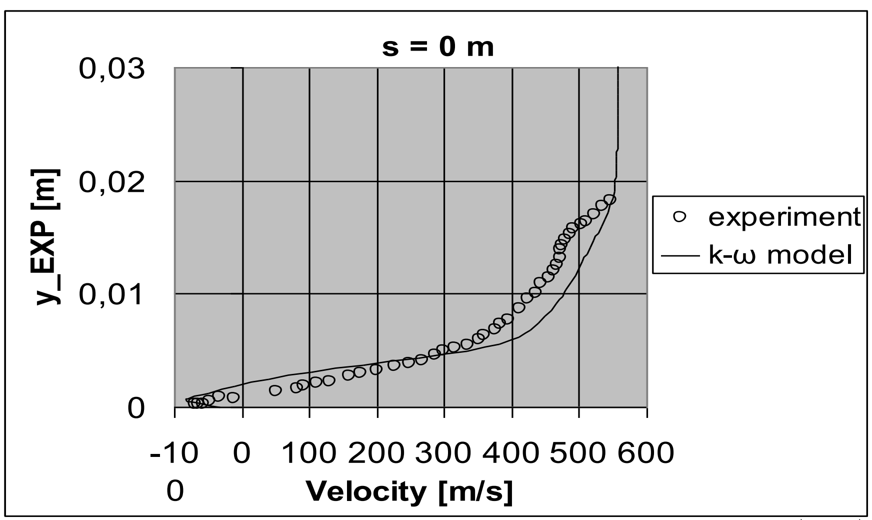


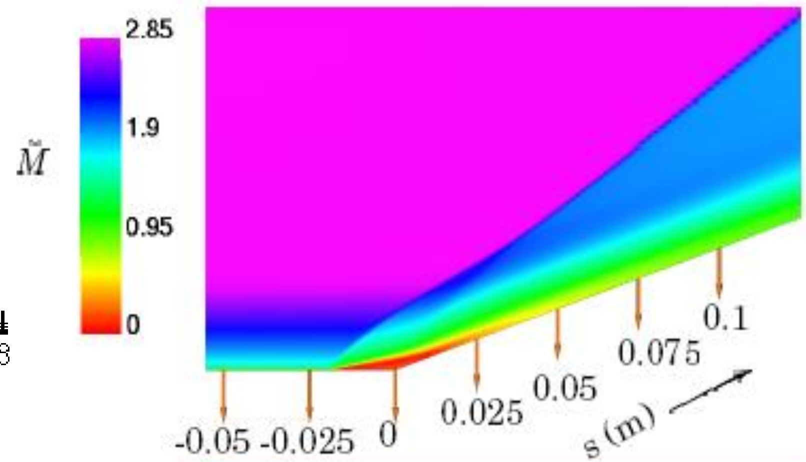
M_1=1.1

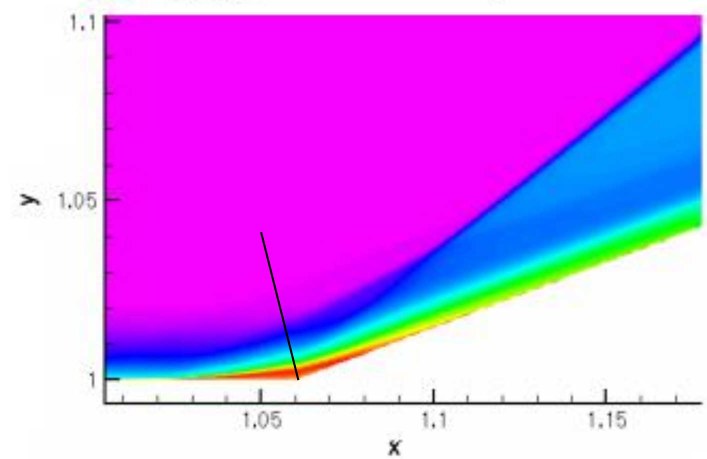


DASFLOW Program – Egyenletek – Validáció (súrlódásos)



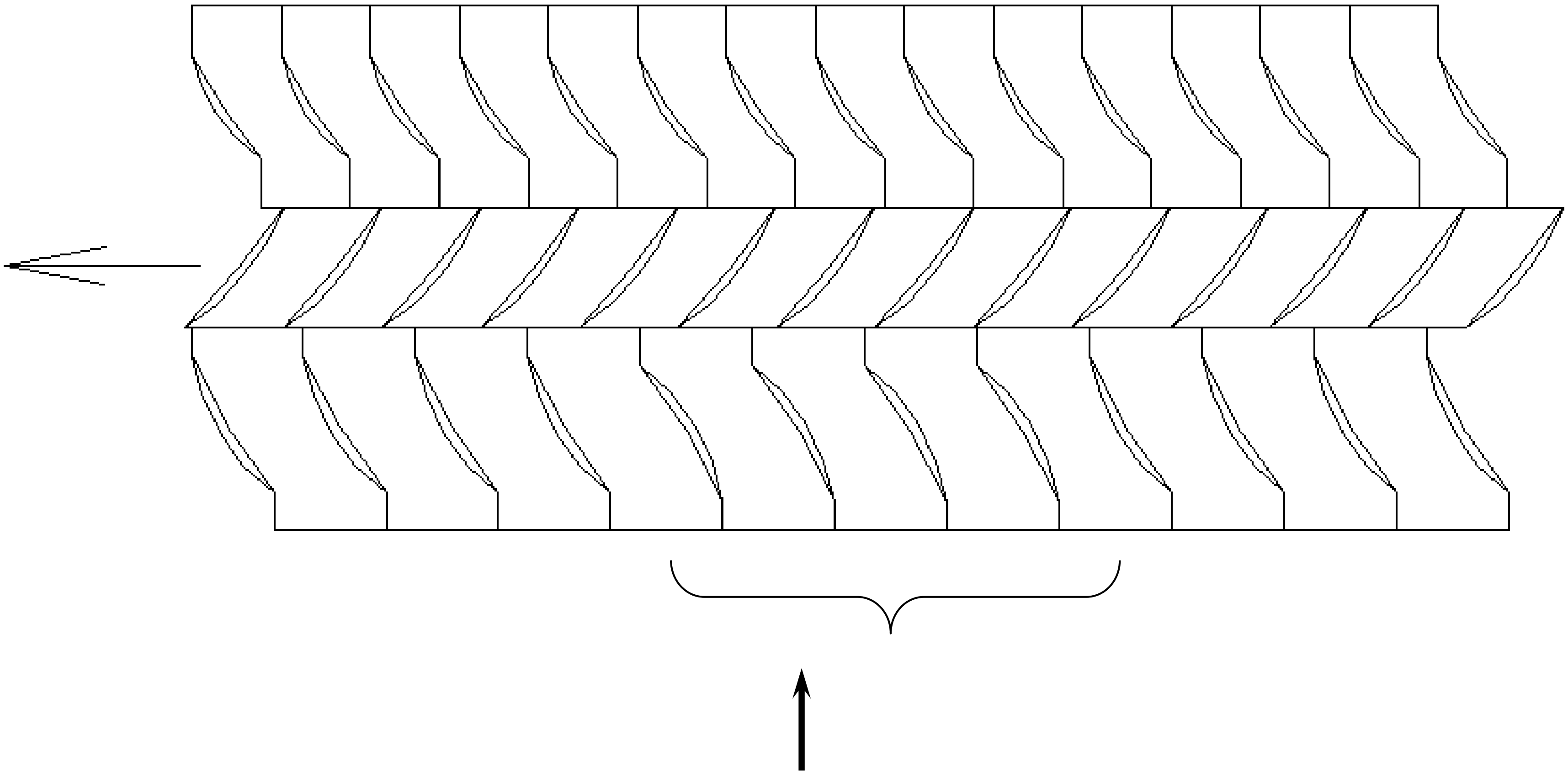






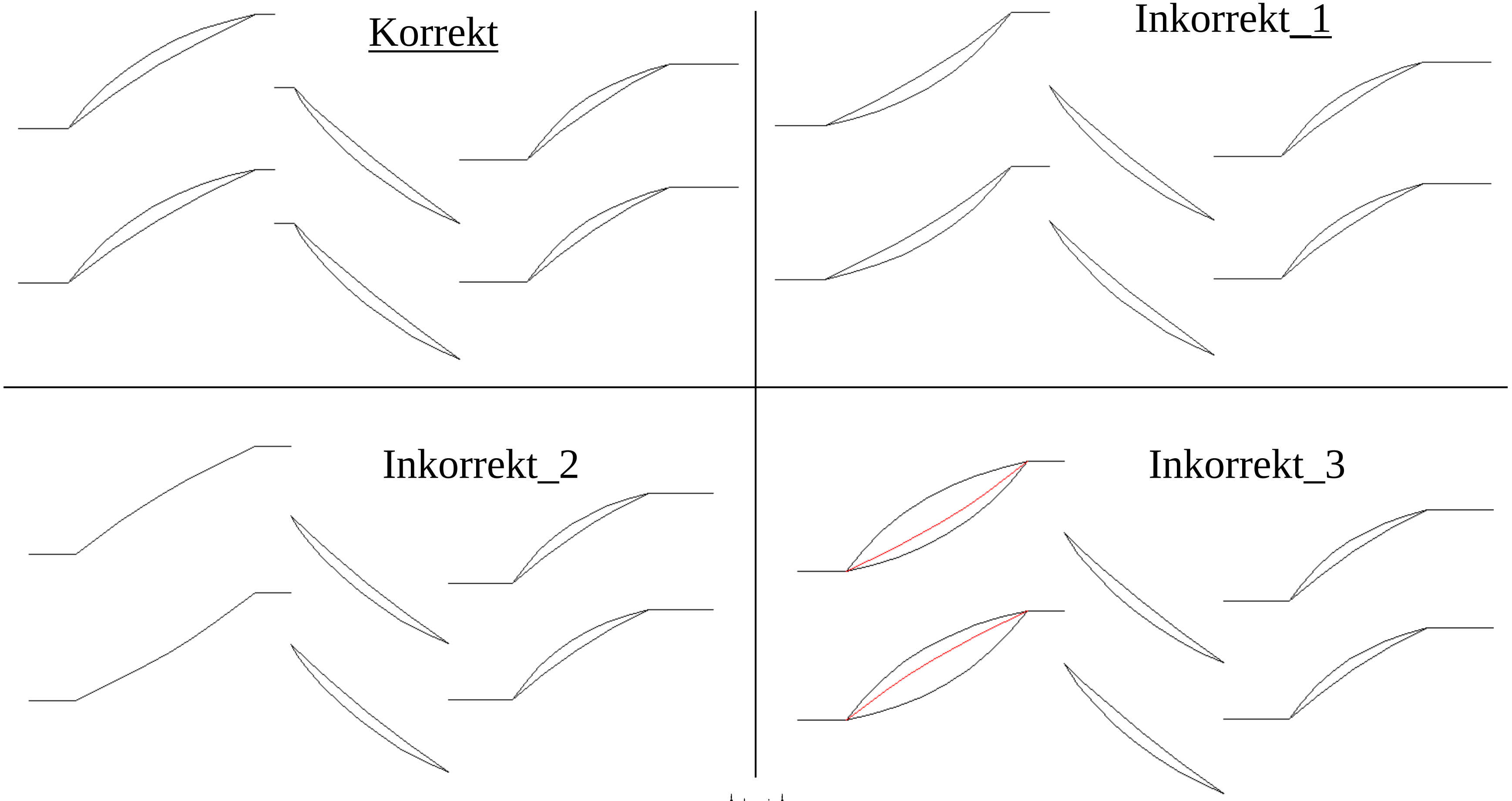
DASFLOW Program - Lapátrács numerikus áramlástani vizsgálata

Probléma leírása



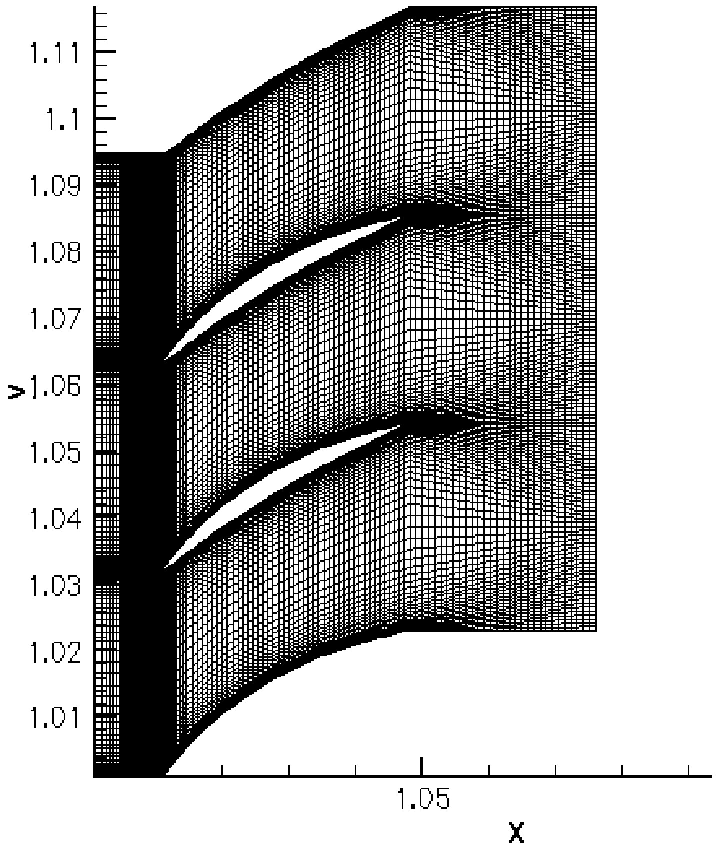
DASFLOW Program - Lapátrács numerikus áramlástani vizsgálata

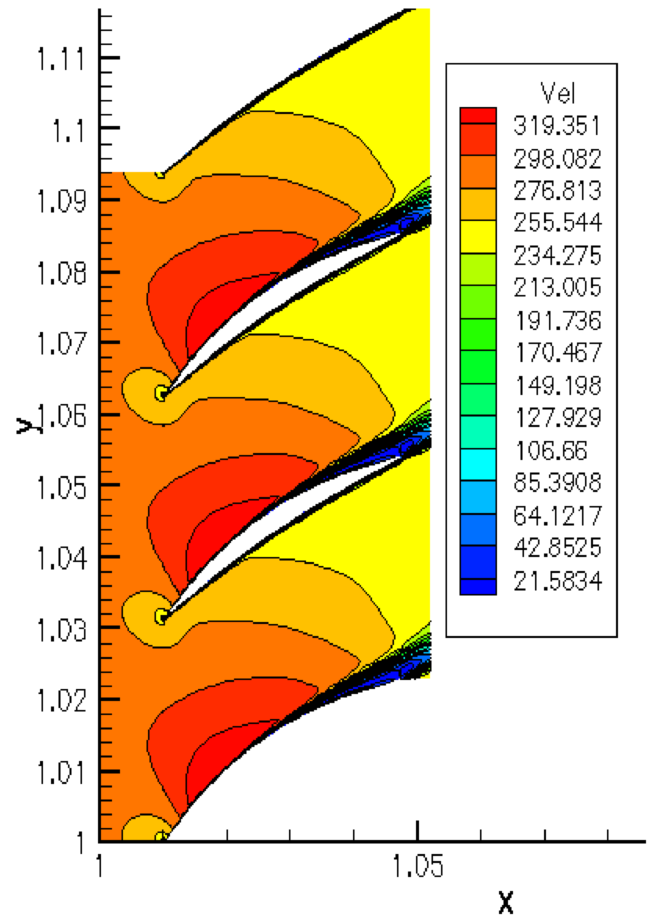
Probléma leírása



DASFLOW Program - Lapátrács numerikus áramlástani vizsgálata

Korrekt



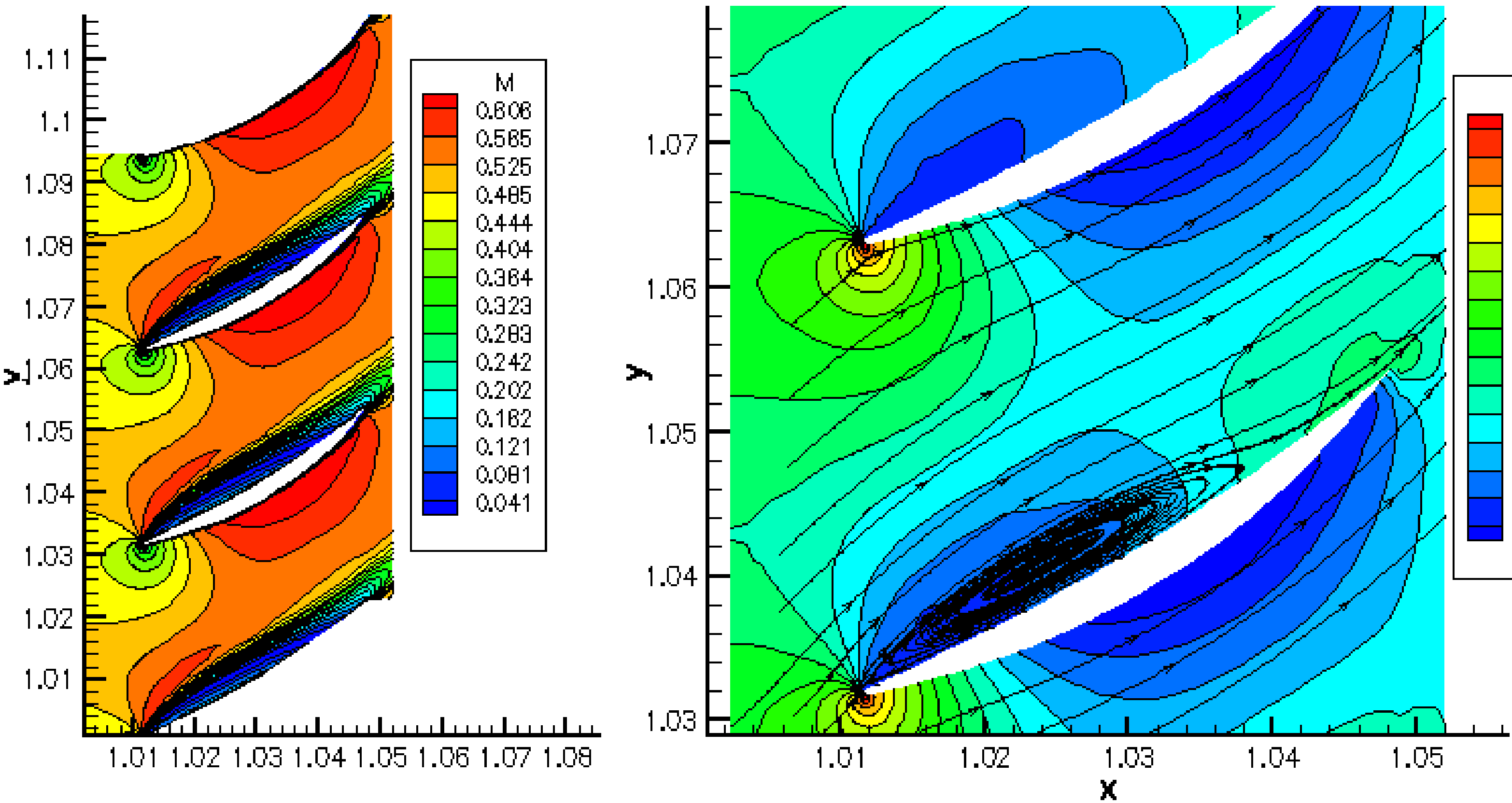


p^{to}_{in}= 796127 Pa, T^{to}_{in}= 530.7 K, $alfa_{in}$=43°

p_{out}= 654600. Pa, $p^{to}_{out\_ave}$=780525. Pa, $T^{to}_{out\_ave}$=530.5 K, alfa_r_{out_ave}= -43.7°

DASFLOW Program - Lapátrács numerikus áramlástani vizsgálata

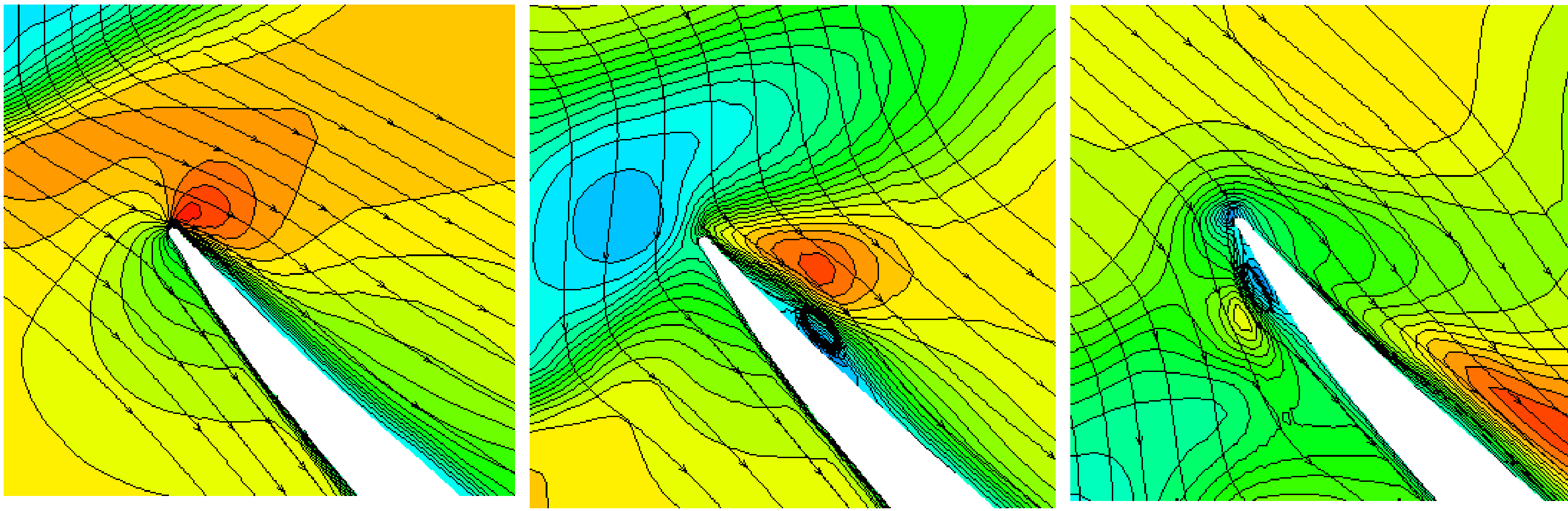
Inkorrekt_1



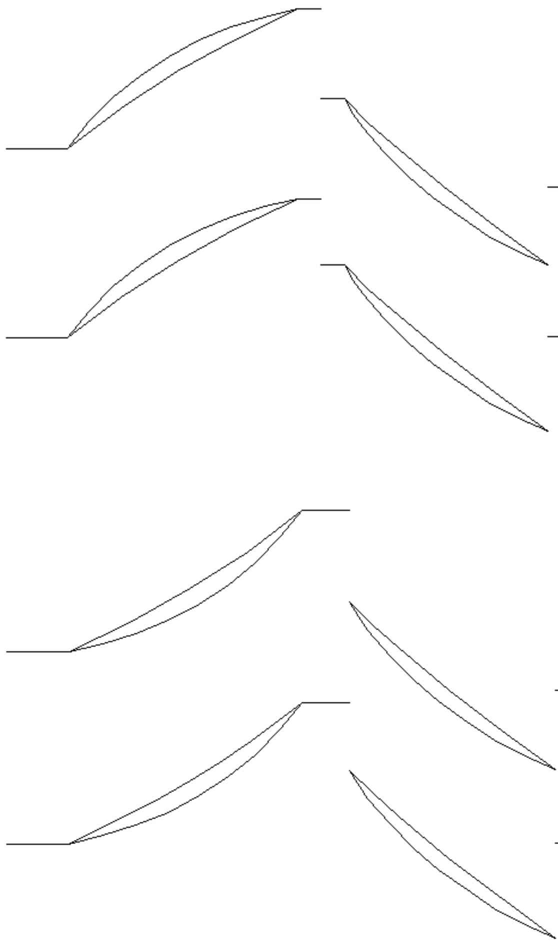
p^{to}_{in}= 796127 Pa, T^{to}_{in}= 530.7 K, $alfa_{in}$=43^o

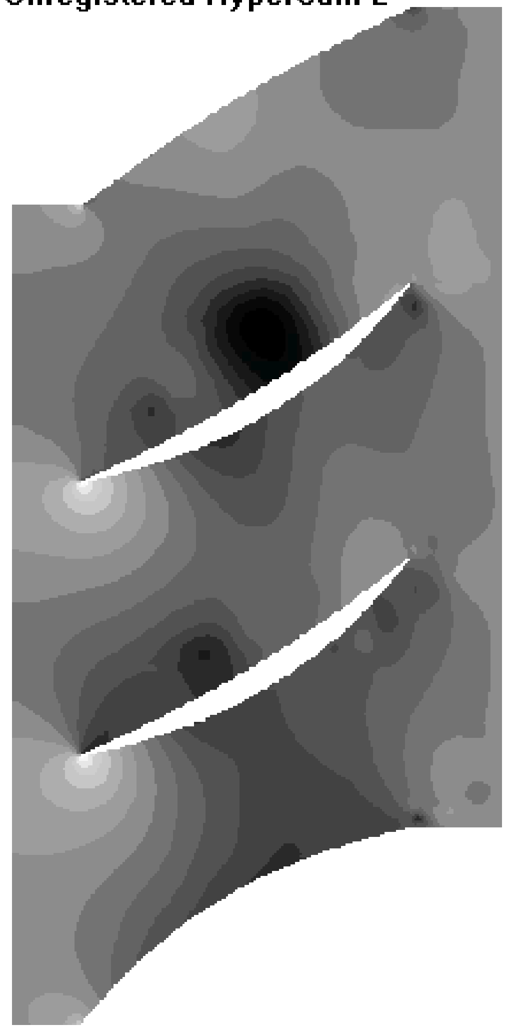
p_{out}= 654600. Pa, $p^{to}_{out\_ave}$=780525. Pa, $T^{to}_{out\_ave}$=530.5 K, alfa_$r_{out-ave}$= -43.7^o

DASFLOW Program - Lapátrács numerikus áramlástani vizsgálata



DASFLOW Program - Lapátrács numerikus áramlástani vizsgálata





Tüzelőanyag sugárszivattyú direkt numerikus optimalizálása - Ipari alkalmazás

2D-s összenyomhatatlan áramlás Euler egyenletei konzervatív és dimenziós alakban:

$$\frac{\partial U^i}{\partial t}+\frac{\partial F^i}{\partial x}+\frac{\partial G^i}{\partial y}=0$$

$$U^i=\left(0,u,v\right)^T \qquad F^i=(u,u^2+\frac{p}{\rho},vu)^T$$

$$G^i=(v,uv,v^2+\frac{p}{\rho})^T$$

Chorin módszere szerint:

$$\frac{\partial U^i}{\partial t}+\frac{\partial F^i}{\partial x}+\frac{\partial G^i}{\partial y}=0$$

$$U^i=\left(P,u,v\right)^T \qquad F^i=(u\beta^2,u^2+P,vu)^T$$

$$G^i=(v\beta^2,uv,v^2+P)^T$$

$$P=\frac{p}{\rho}$$

Tüzelőanyag sugárszivattyú direkt numerikus optimalizálása - Ipari alkalmazás

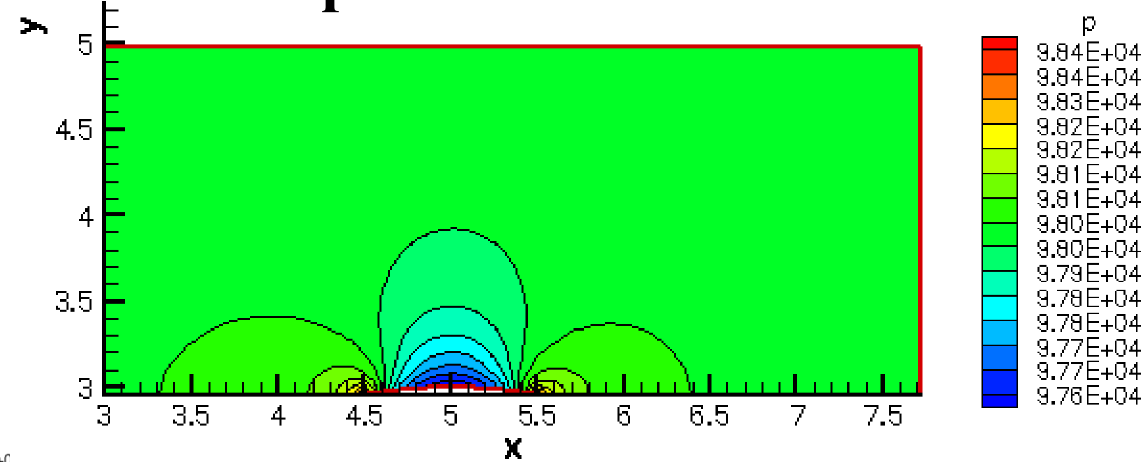
• Integrál egyenletek

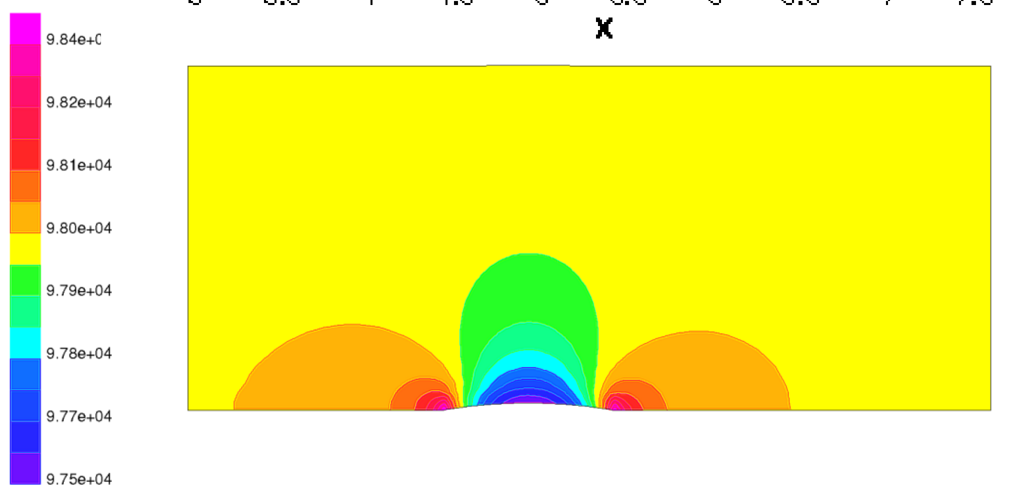
$$\begin{cases} \dfrac{\partial}{\partial t}\iint\limits_{\Omega} U d\Omega + \int_{\Gamma} H_n d\Gamma = 0 \\ H_n = \vec{H}\vec{n} = \begin{pmatrix} \beta^2 V_n \\ uV_n + Pn_x \\ vV_n + Pn_y \end{pmatrix} \end{cases}$$

$$\left.\begin{aligned} \frac{\partial}{\partial t} U_j &= -\frac{1}{\Omega_j}\sum_{k=1}^{N_f}\left[H_n\right]_{j,k}\Gamma_{j,k} + \frac{1}{\Omega_j} D \\ U^0 &= U^n \\ U^k &= U^0 + \alpha_k \Delta t \Re\left(U^{k-1}\right) \qquad k = 1,m \\ U^{n+1} &= U^m \end{aligned}\right\}$$

• Runge-Kutta módszer

Tüzelőanyag sugárszivattyú direkt numerikus optimalizálása - Ipari alkalmazás - Validáció

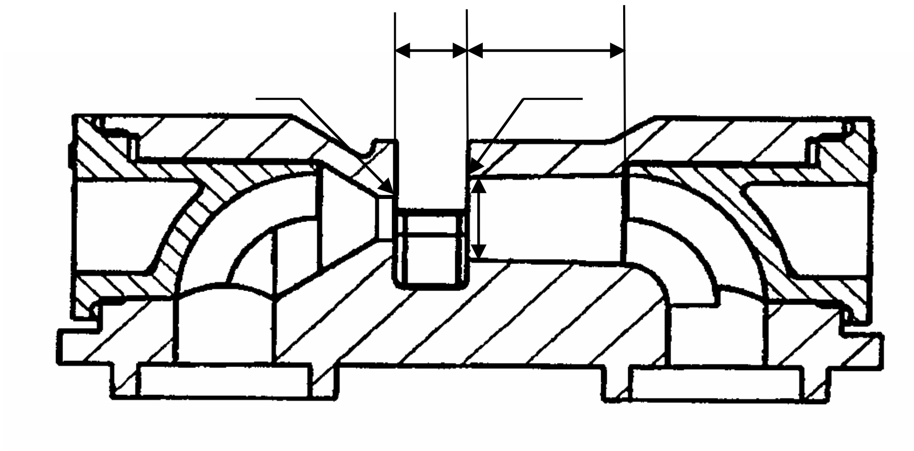


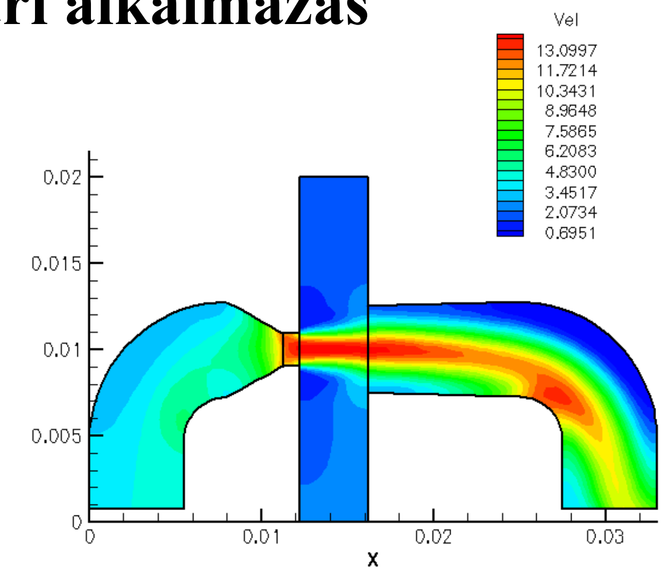


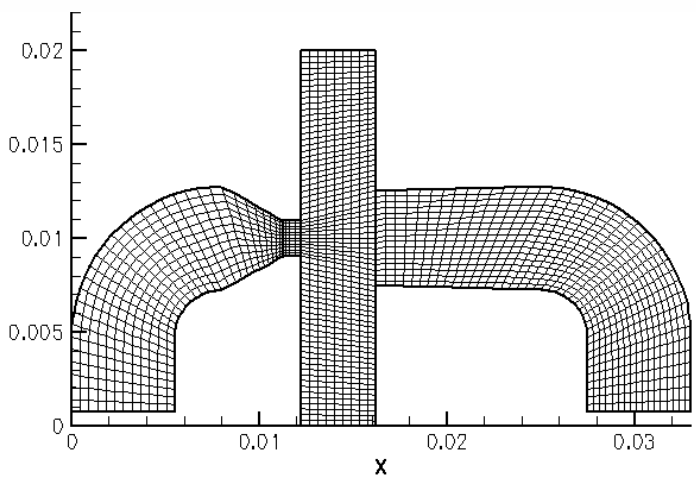
Contours of Static Pressure (pascal)

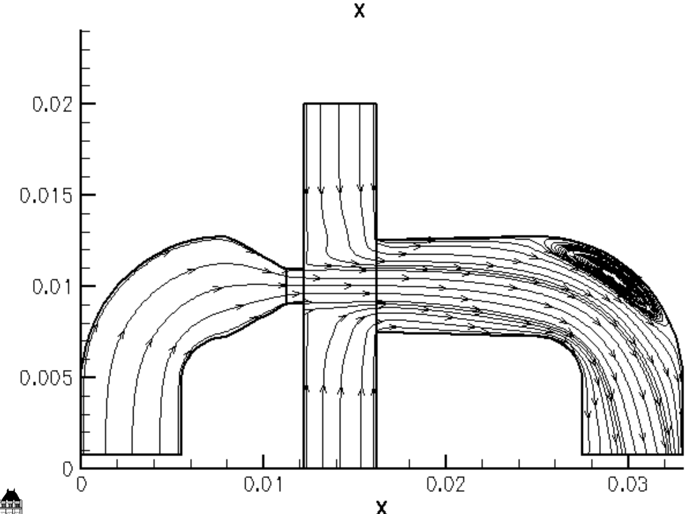
Sep 13, 2003
FLUENT 5.2 (2d, segregated)

Tüzelőanyag sugárszivattyú direkt numerikus optimalizálása - Ipari alkalmazás

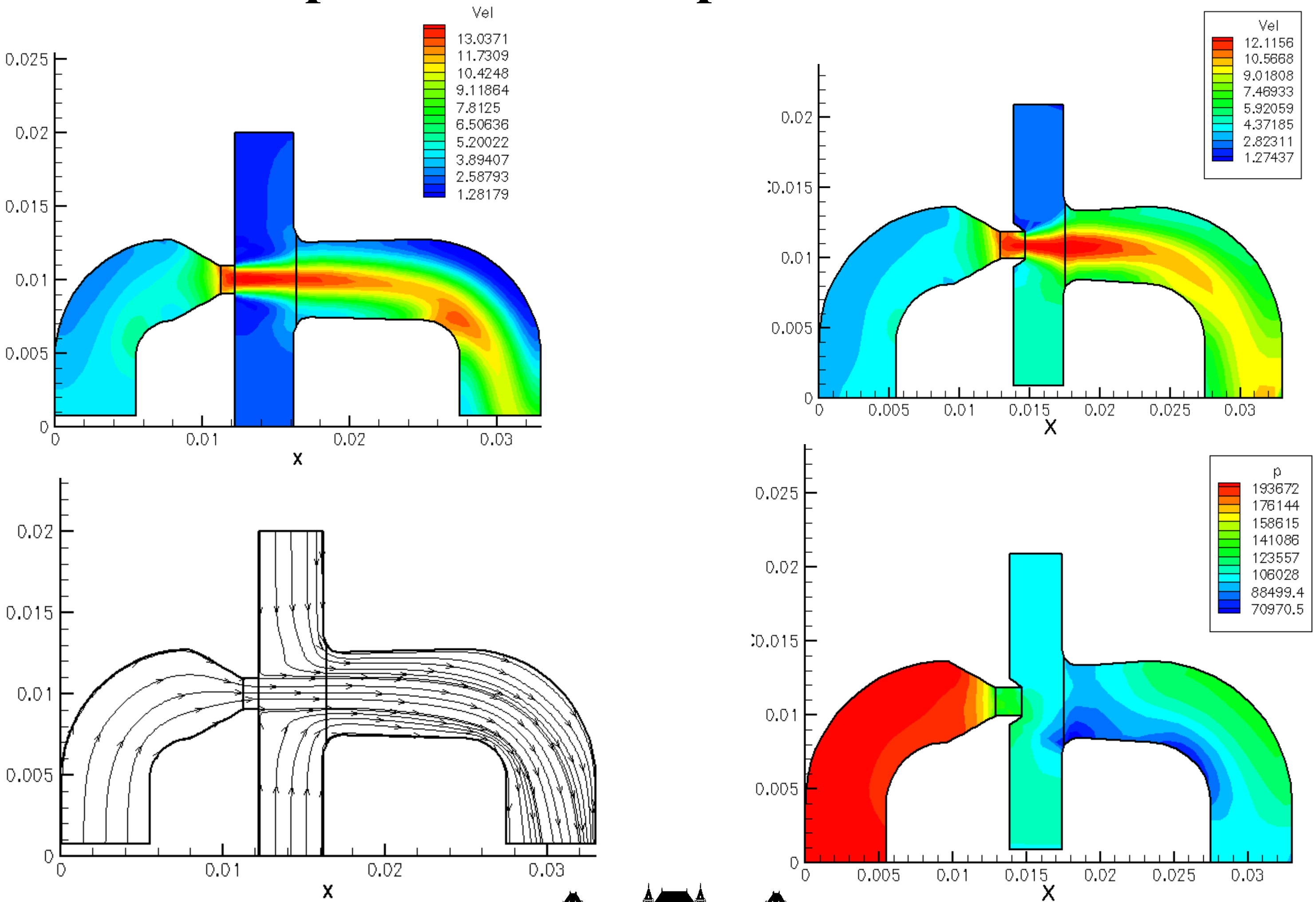








Tüzelőanyag sugárszivattyú direkt numerikus optimalizálása - Ipari alkalmazás

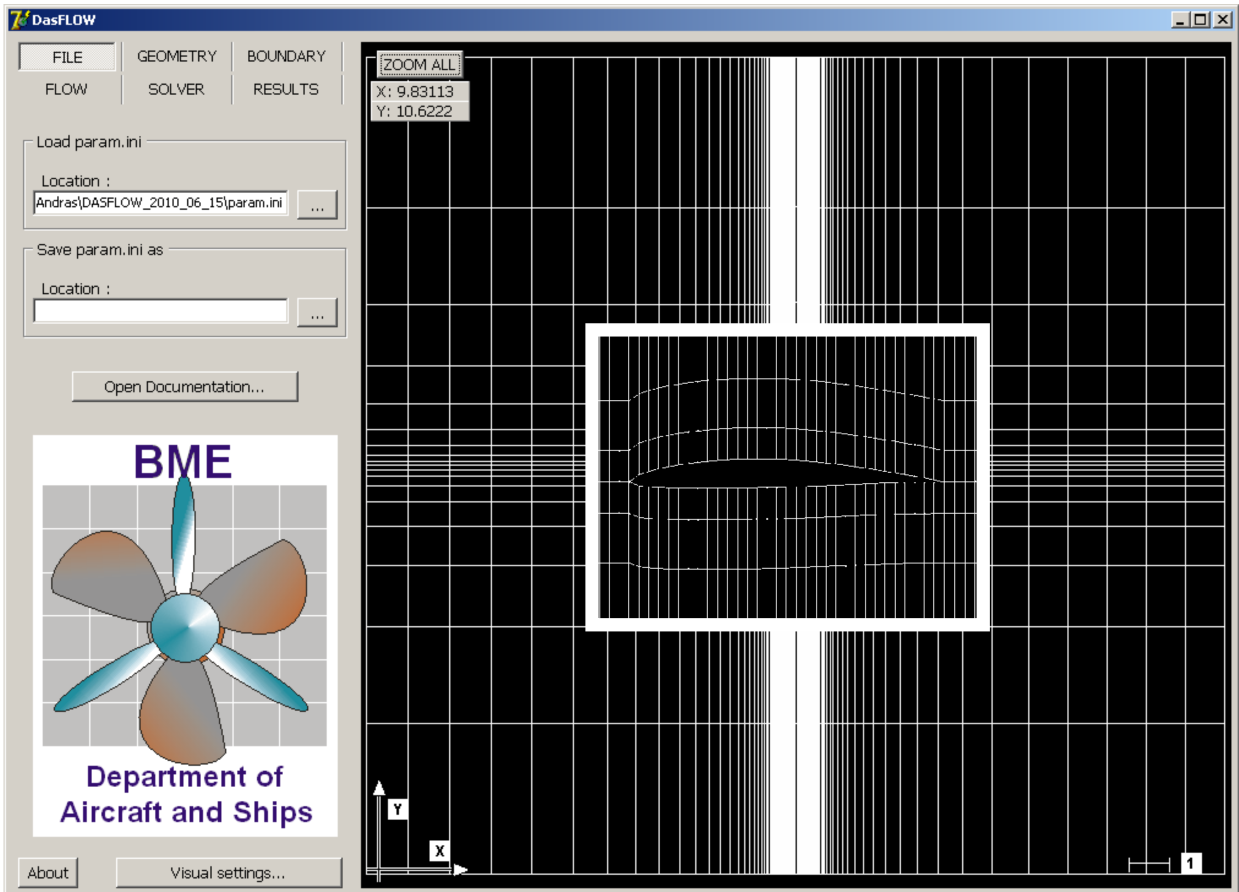


Tüzelőanyag sugárszivattyú direkt numerikus optimalizálása - Ipari alkalmazás

Modell	**Alap**	**Letörés**	**Optimalizált 1**	**Optimalizált 2**
m_{be} [l/h]	129,8	128,9	126	115,9
m_{ki} [l/h]	307	326,5	350,33	369,1
Sz.k.	2,36	2,53	2,78	3,18

Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

Szárnyprofilra, NACA 65-4101



GUI view of the DASFLOW in-house 2D CFD analysis and design software

Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

Szárnyprofilra, NACA 65-4101

ScienceDirect

Home | Publications | Search | My settings | My alerts | Shopping cart

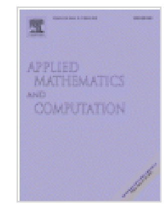
Purchase Export citation Jump to references More options...

ELSEVIER

Applied Mathematics and Computation

Volume 219, Issue 13, 1 March 2013, Pages 7115–7126

ESCO 2010 Conference in Pilsen, June 21- 25, 2010



Coupled problem of the inverse design and constraint optimization

Árpád Veress[a], Attila Felföldi[a], Tamás Gausz[a], László Palkovics[b]

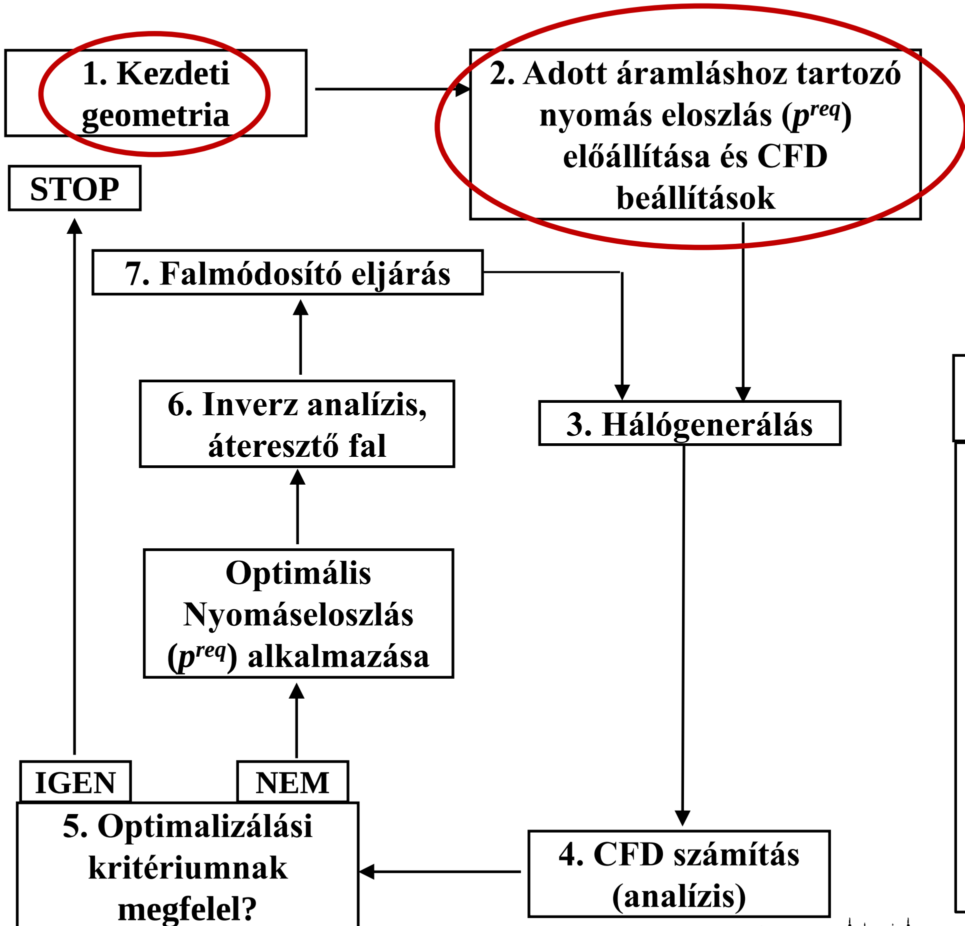
[a] Department of Aircraft and Ships, Budapest University of Technology and Economics, Sztoczek u. 6, J ép. 4. em. 426, H-1111 Budapest, Hungary

[b] Knorr-Bremse R&D Center Budapest, Major u. 69, H-1119 Budapest, Hungary

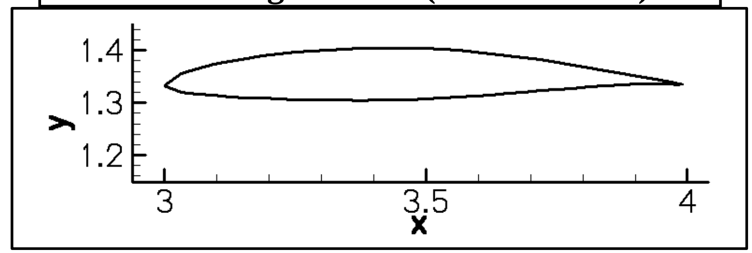
Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

Szárnyprofilra, NACA 65-4101

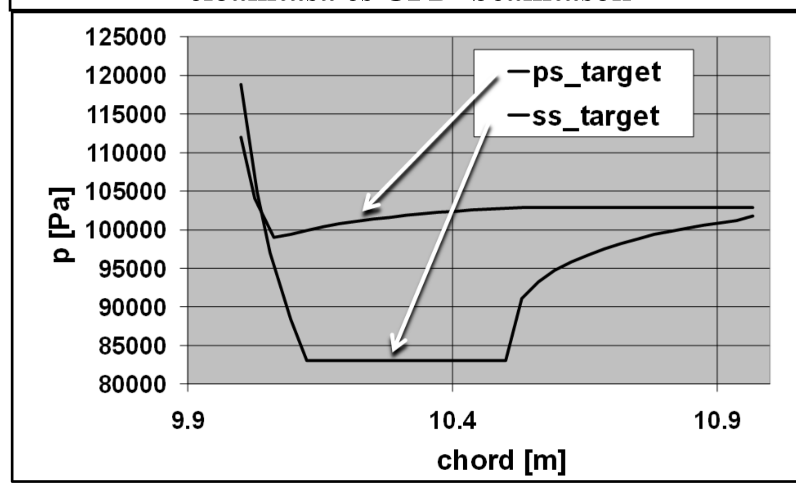
A számítás folyamata:



1. Kezdeti geometria (NACA 65 410)



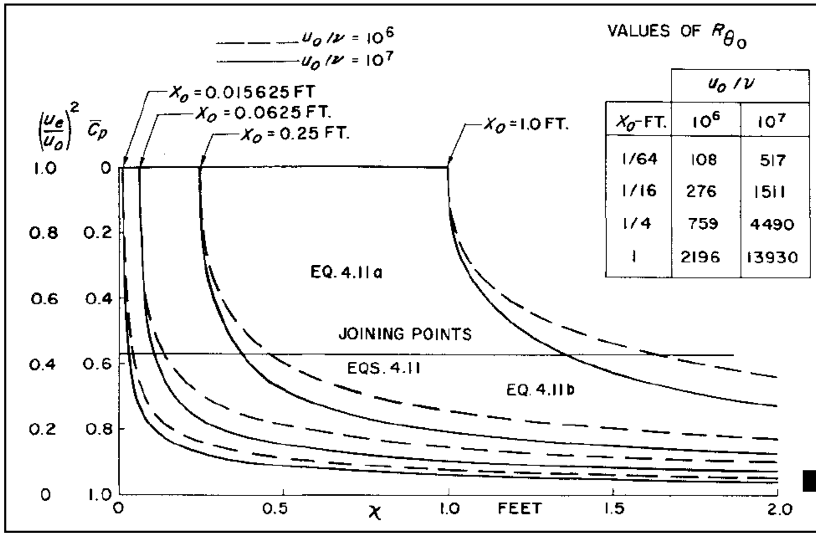
2. Adott áramláshoz tartozó nyomás eloszlás (p^{req}) előállítása és CFD beállítások



Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

Szárnyprofilra, NACA 65-4101

Stratford limiting flow at given Reynolds number



$$\frac{\bar{C}_p[x(d\bar{C}_p/dx)]^{1/2}}{(10^{-6}R)^{1/10}} = S$$

where if $d^2p/dx^2 \geq 0$, then $S = 0.39$; or if $d^2p/dx^2 \leq 0$, then $S = 0.35$. Also, $\bar{C}_p \leq 4/7$.

$$\bar{C}_p = 0.645\{0.435\, R_o^{1/5}[(x/x_o)^{1/5} - 1]\}^{2/n}$$

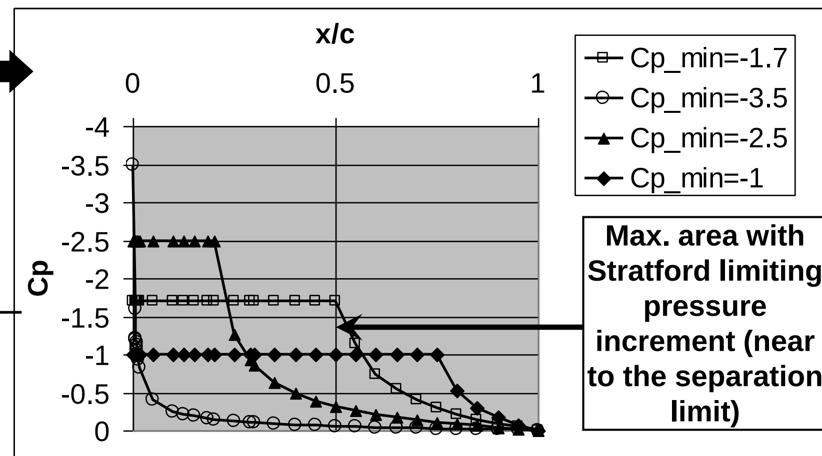
for $\bar{C}_p \leq (n-2)/(n+1)$ ***(n=6)***

$$\bar{C}_p = 1 - \frac{a}{[(x/x_o)+b]^{1/2}} \quad \text{for } \bar{C}_p \geq \frac{n-2}{n+1}$$

Pressure coefficient:

$$C_p = \frac{p - p_\infty}{1/2\rho_\infty V_\infty^2} \equiv \frac{p - p_\infty}{0.7 p_\infty M_\infty^2}$$

p^{req}



Max. area with Stratford limiting pressure increment (near to the separation limit)

Canonical pressure distribution:

$$\bar{C}_p = \frac{p - p_0}{1/2\rho_0 V_0^2}$$

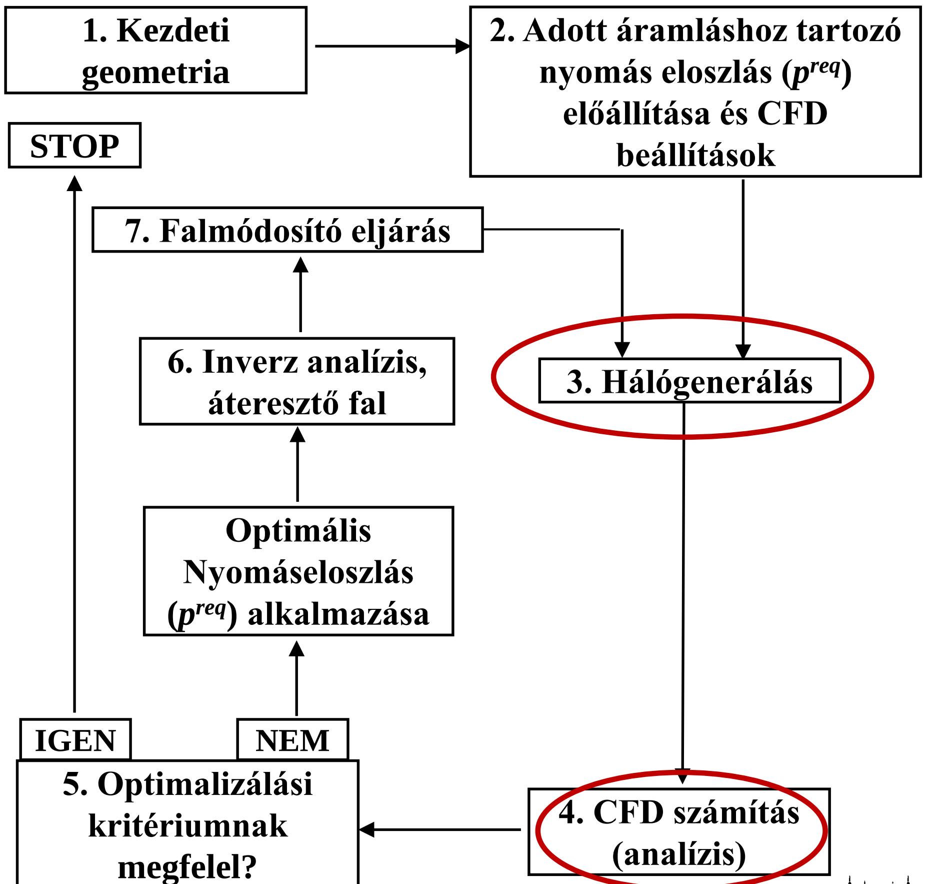
$_0$: Location of the start of the positive pressure gradient

p^{req} is the output of the optimization

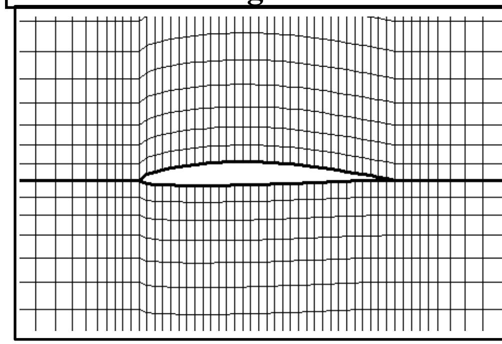
Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

Szárnyprofilra, NACA 65-4101

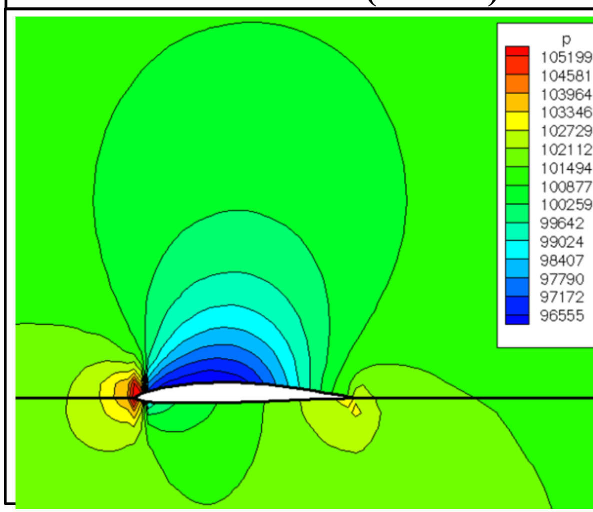
A számítás folyamata:



3. Hálógenerálás



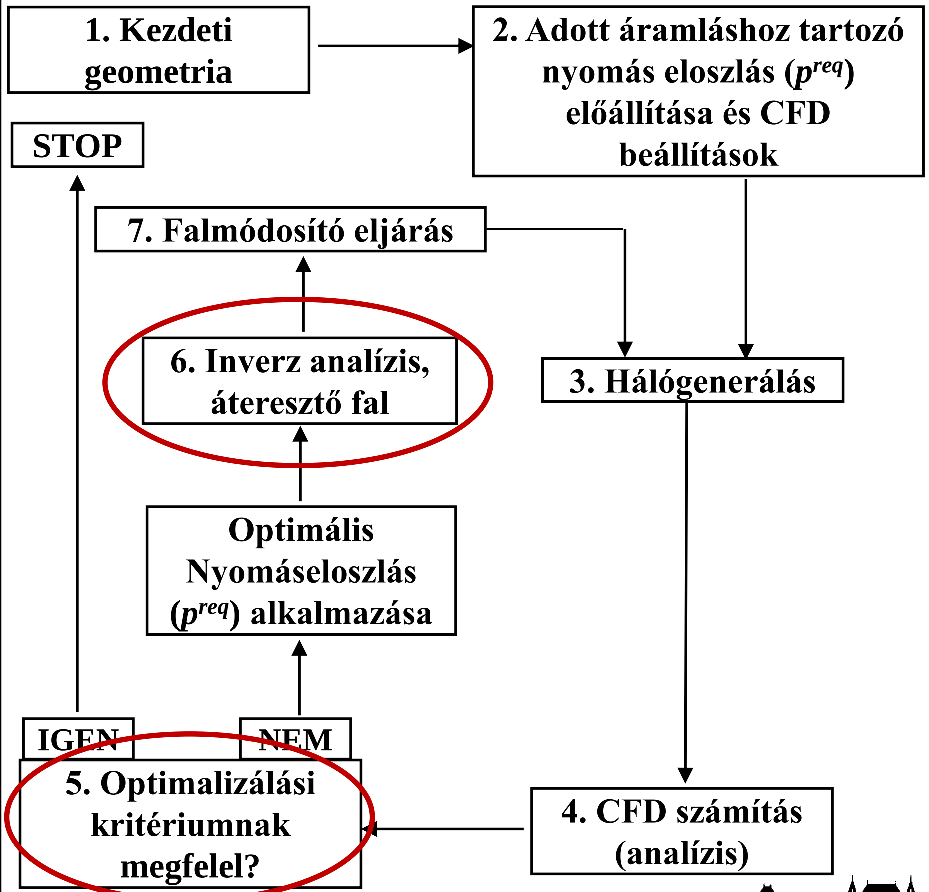
4. CFD számítás (analízis)

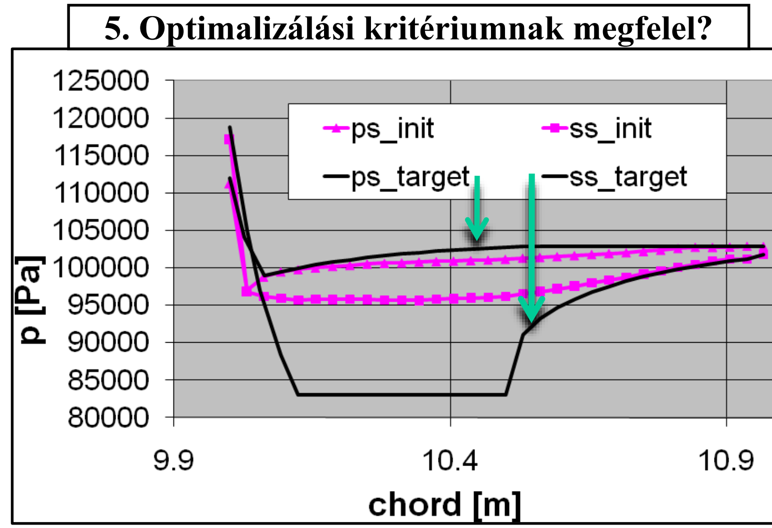


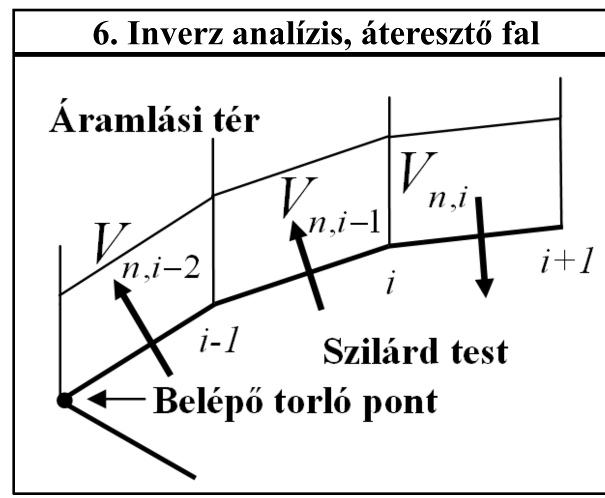
Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

Szárnyprofilra, NACA 65-4101

A számítás folyamata:



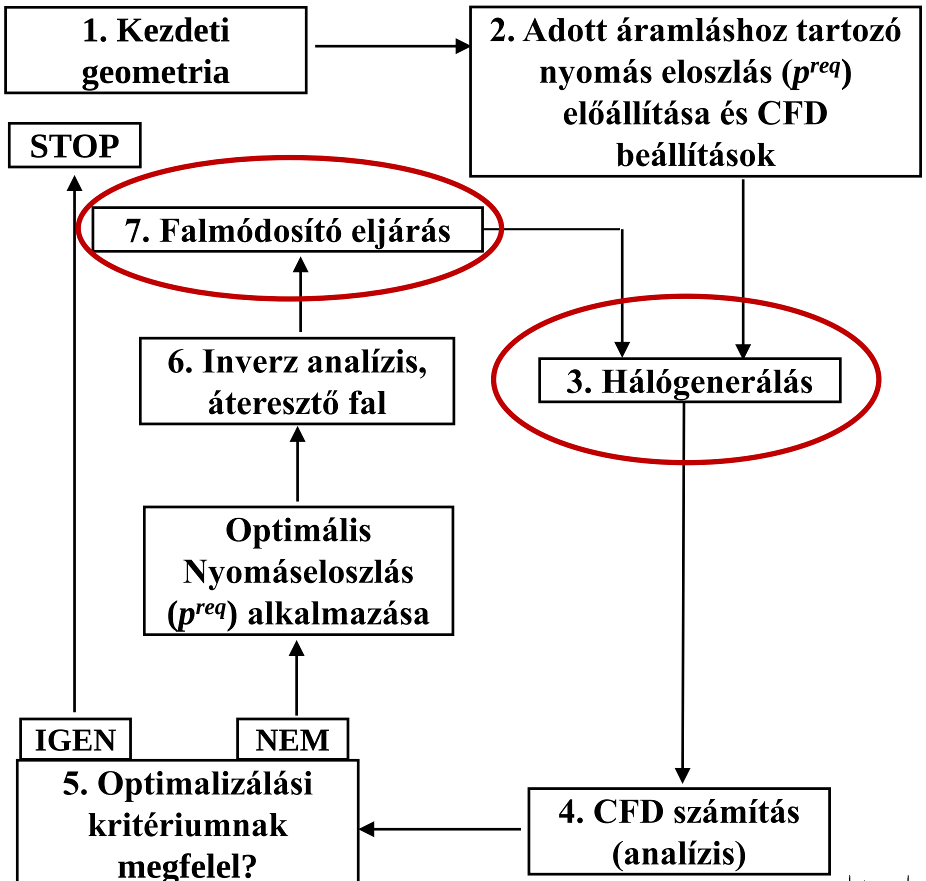


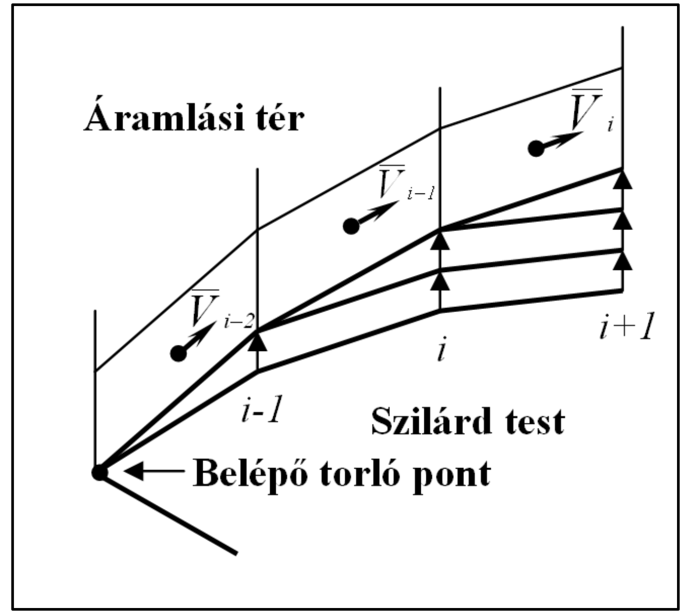


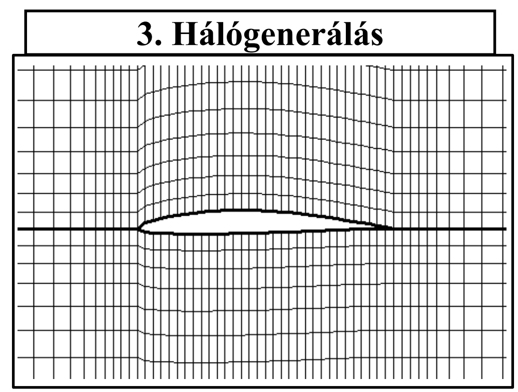
Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

Szárnyprofilra, NACA 65-4101

A számítás folyamata:



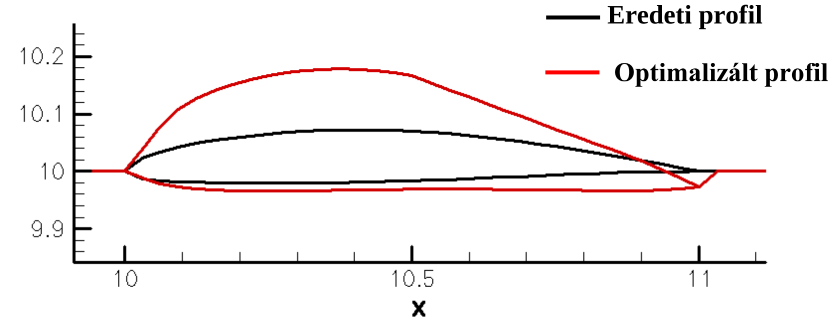


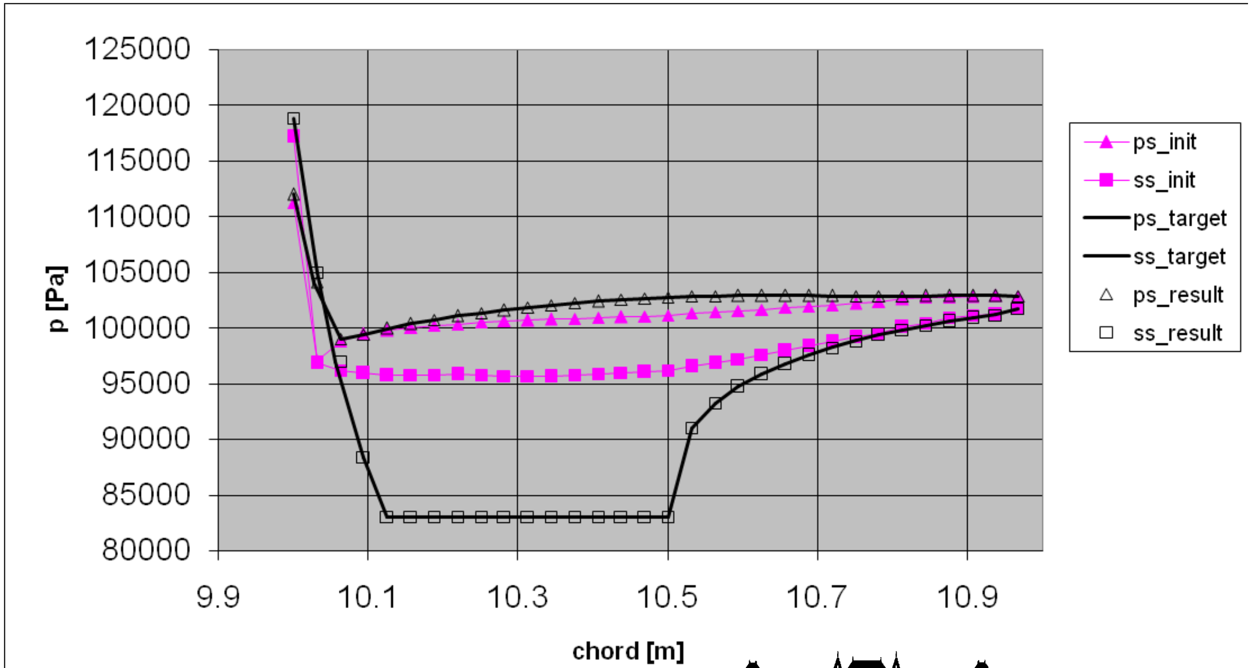


Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

Szárnyprofilra, NACA 65-4101

Eredmények 10 inverz iterációt követően:





Peremfeltételek:

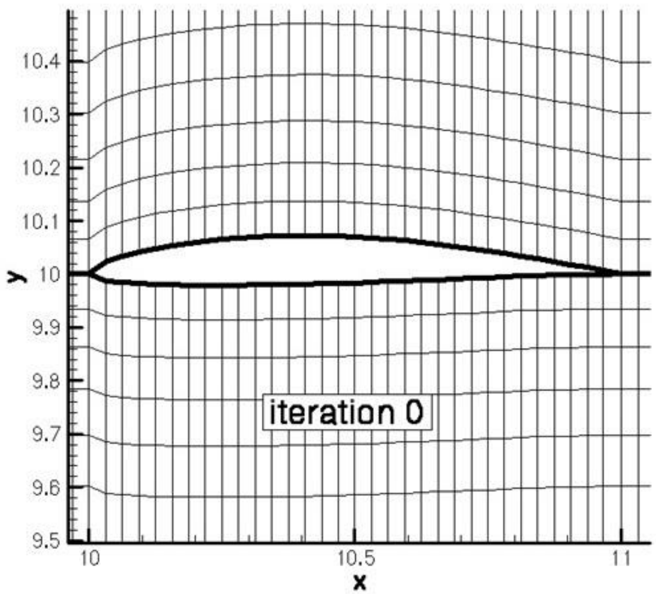
Belépő torlóponti nyomás:
$p_{tot,in}$**=112799 [Pa];**
Belépő torlóponti hőm.:
$T_{tot,in}$**=293.15 [K];**
Kilépő statikus nyomás:
$p_{statot,out}$**=101325 [Pa].**

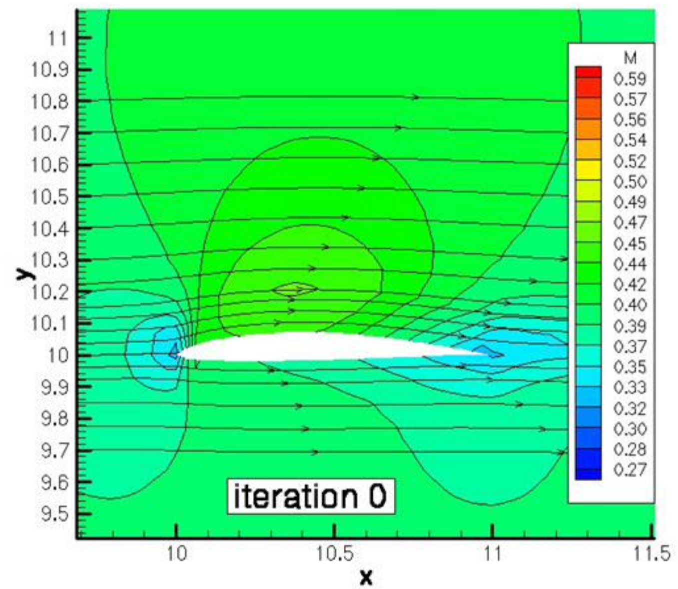
Hálóméret: 87×30
Iteráció szám: 5000
Konvergencia kritérium sűrűség NKÉ: 1e-5.6

Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

Szárnyprofilra, NACA 65-4101

Eredmények 10 inverz iterációt követően:

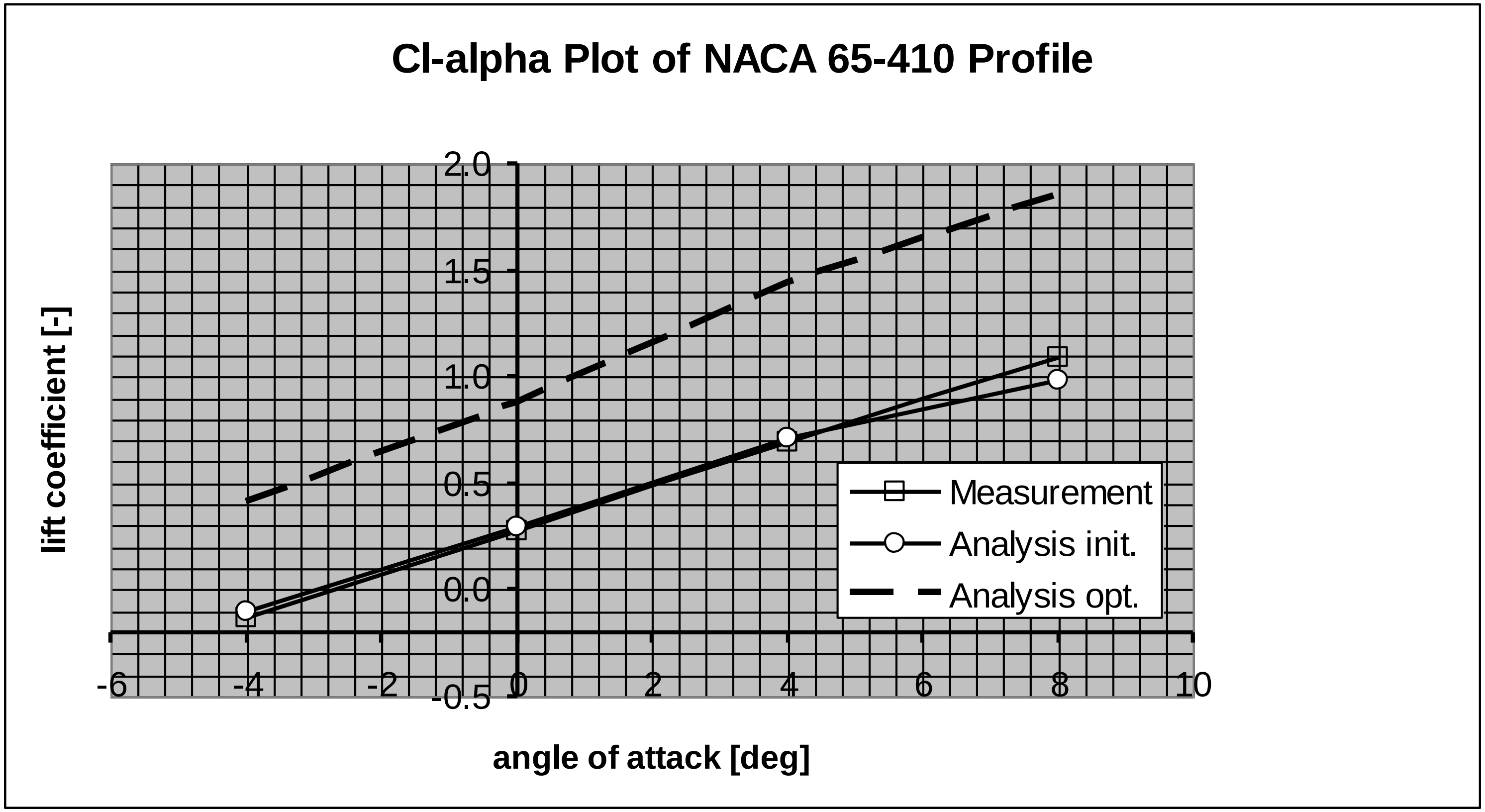




Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

Szárnyprofilra, NACA 65-4101

Eredmények a kezdeti profil esetén és 10 inverz iterációt követően:

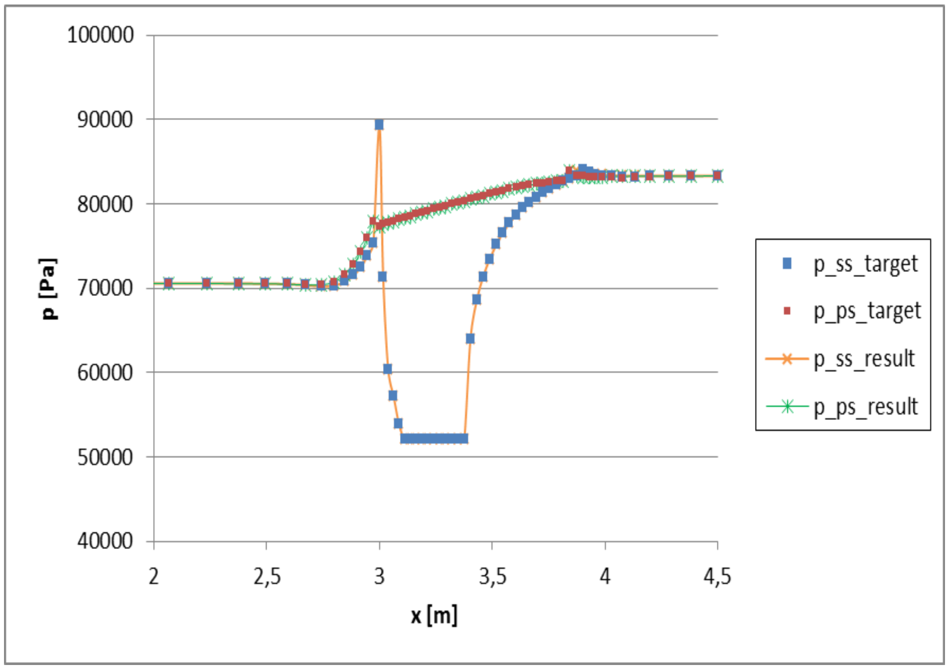


Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

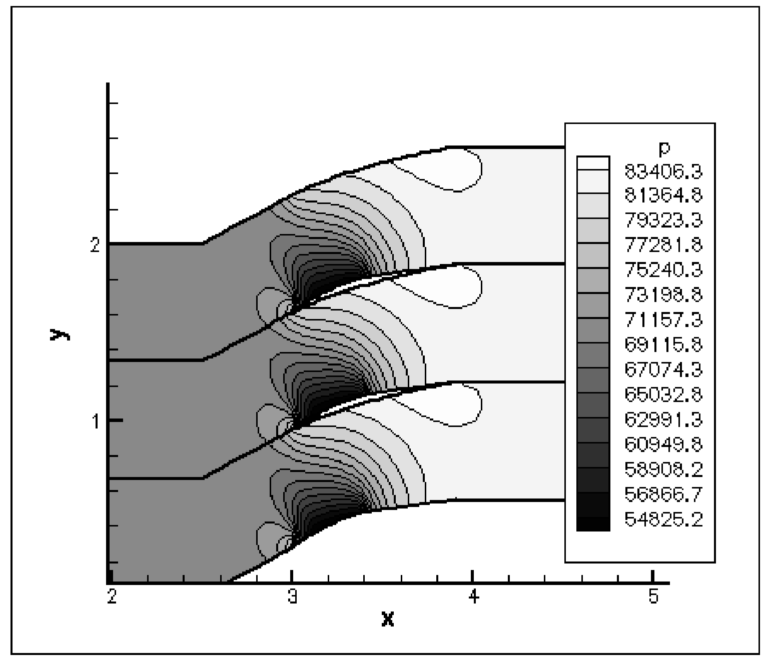
Fokozati kompresszió viszony nagyságát befolyásoló tényezők: $\pi_{ks} = f\left(U(n), C_a, \underline{\beta_1 - \beta_2}\right)$

Eredmények lapátrácsra 10 inverz iterációt követően: $\pi_{ks,static} = 1{,}18$

A 0,62 Mach-szám ($p_{stat,out}$=83325 [Pa]) és C_p = -1,4-hez tartozó nyomáseloszlás



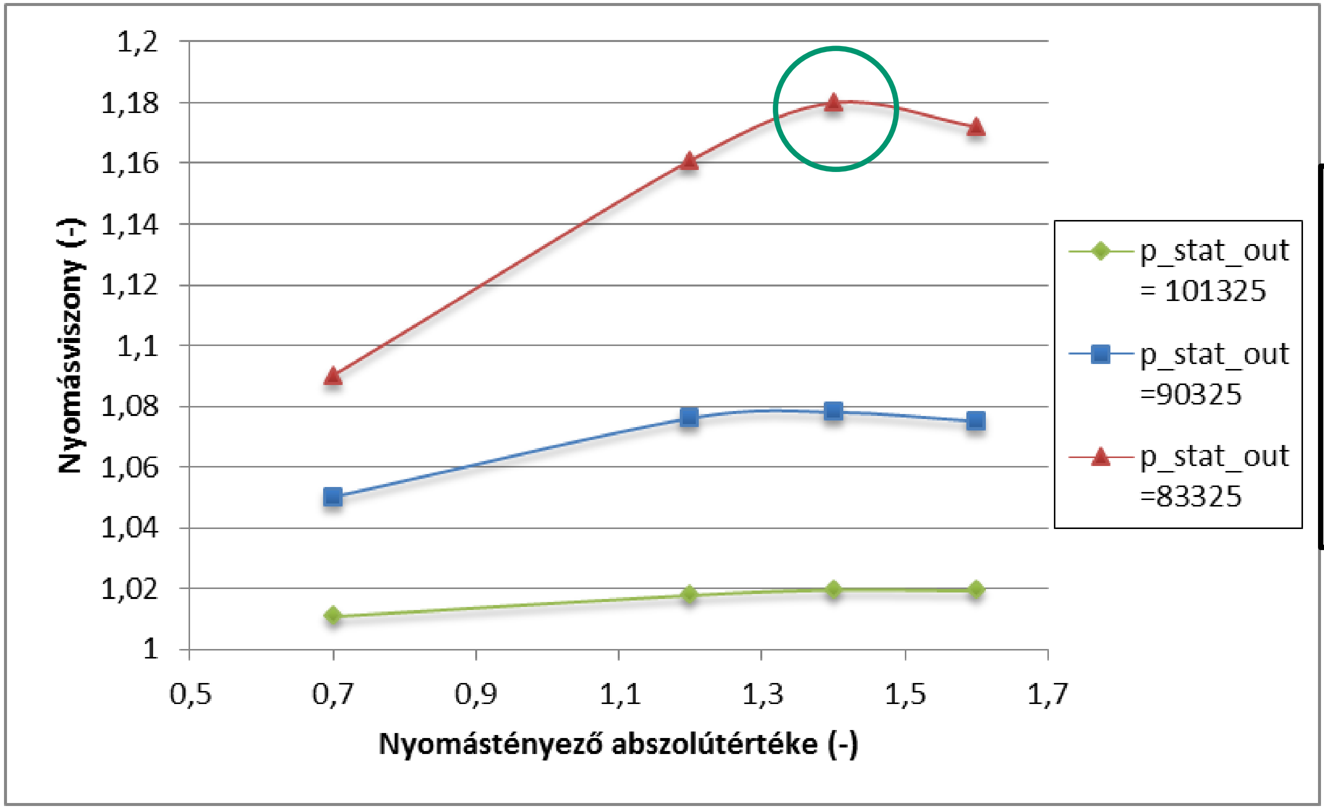
A lapátrácsban kialakuló nyomáseloszlás 0,62 Mach-szám ($p_{stat,out}$=83325 [Pa]) és C_p = -1,4 esetén



Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

Fokozati kompresszió viszony nagyságát befolyásoló tényezők: $\pi_{ks} = f\left(U(n), C_a, \underline{\beta_1 - \beta_2}\right)$

Eredmények lapátrácsra 10 inverz iterációt követően: $\pi_{ks,static} = 1{,}18$



Mach-szám:

← **0,3**

← **0,5**

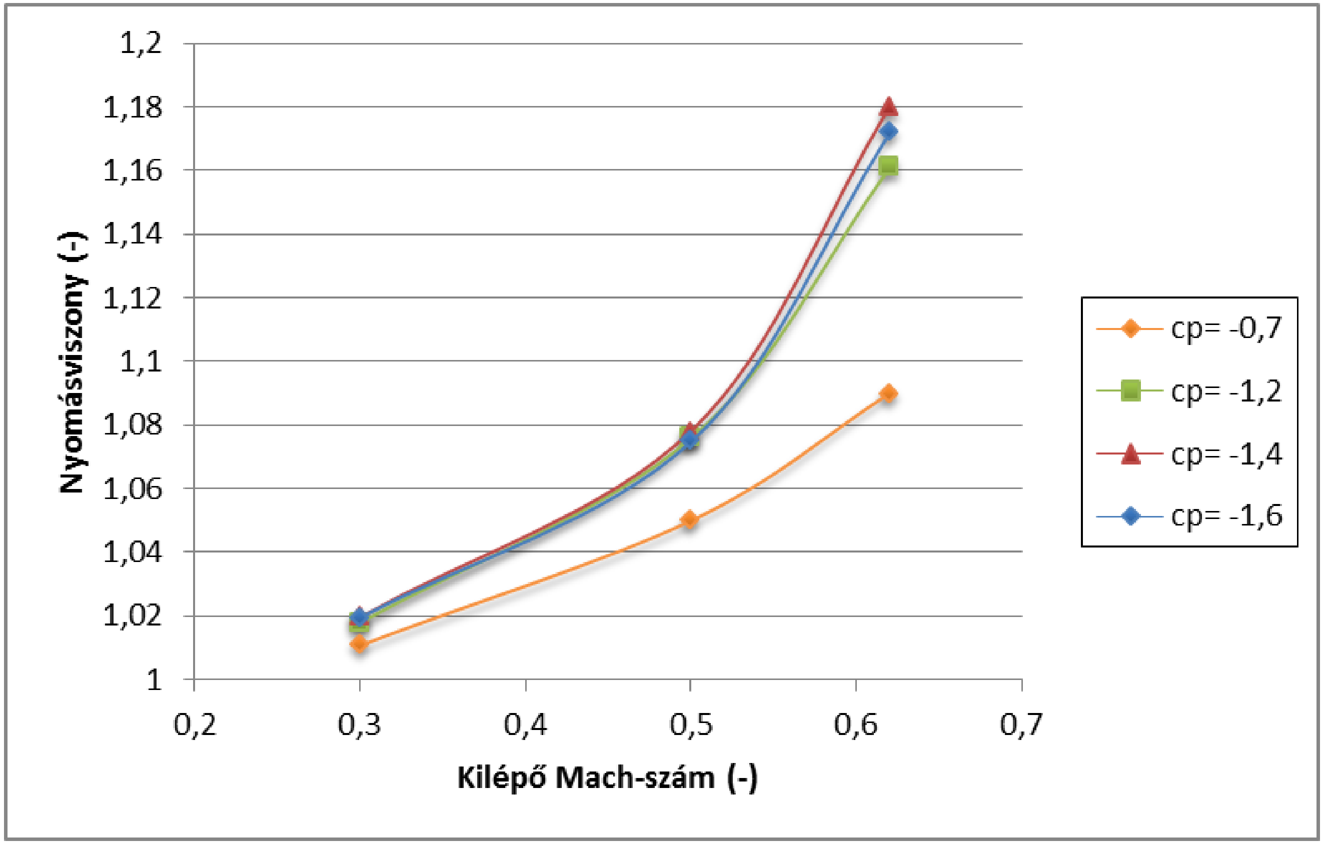
← **0,62**

$$\pi = \frac{p_{stat,out}}{p_{stat,in}} = f\left(p_{stat,out}, Cp\right)$$

Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

Fokozati kompresszió viszony nagyságát befolyásoló tényezők: $\pi_{ks} = f(U(n), C_a, \underline{\beta_1 - \beta_2})$

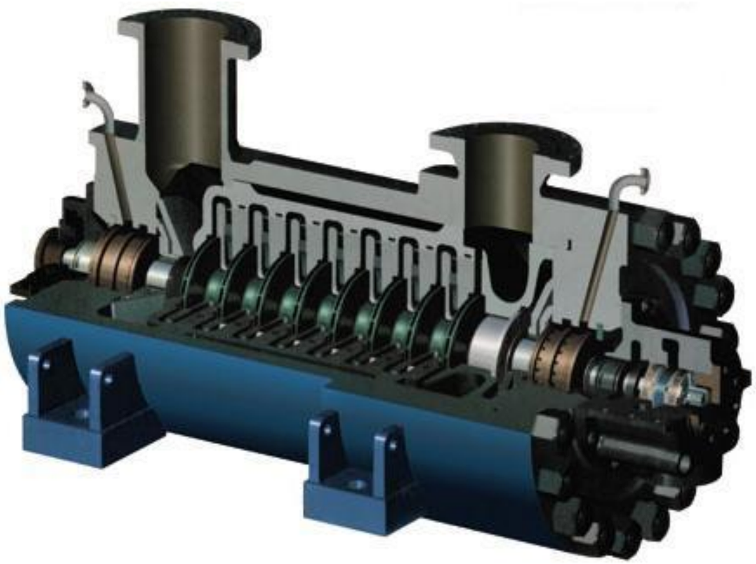
Eredmények lapátrácsra 10 inverz iterációt követően: $\pi_{ks,static} = 1{,}18$

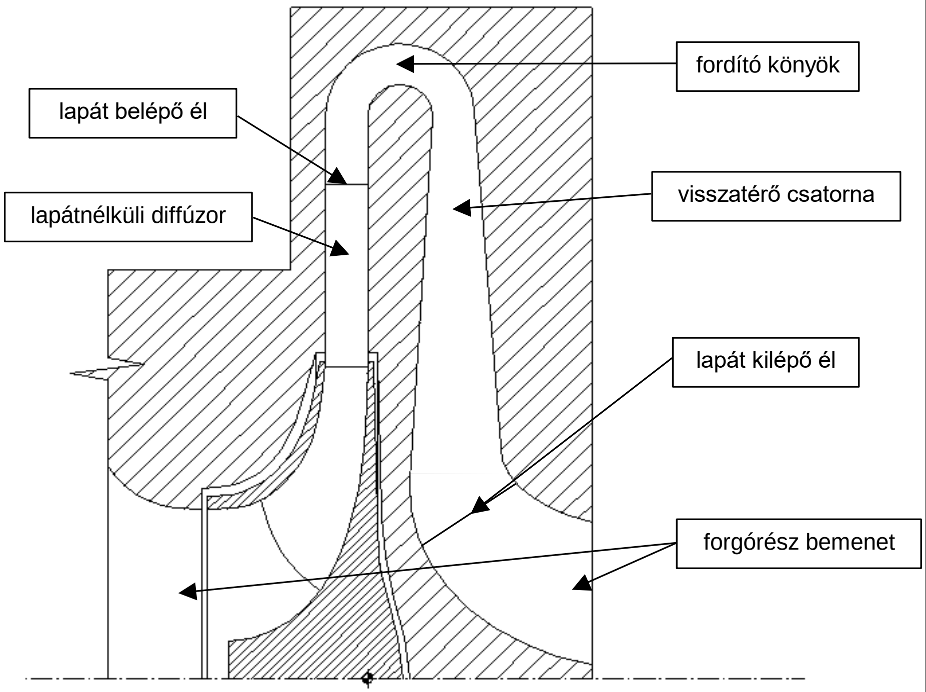


$$\pi = \frac{p_{stat,out}}{p_{stat,in}} = f\left(p_{stat,out}, Cp\right)$$

Appendices III. Industrial Applications of a CFD and Inverse Design Tool Developed at VKI

Többfokozatú centrifugál kompresszor összekötő-csatorna lapátozásának tervezése – Ipari alkalmazás

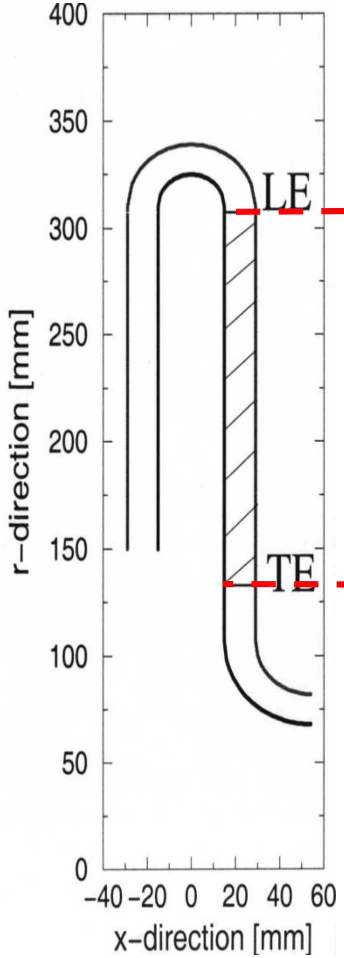


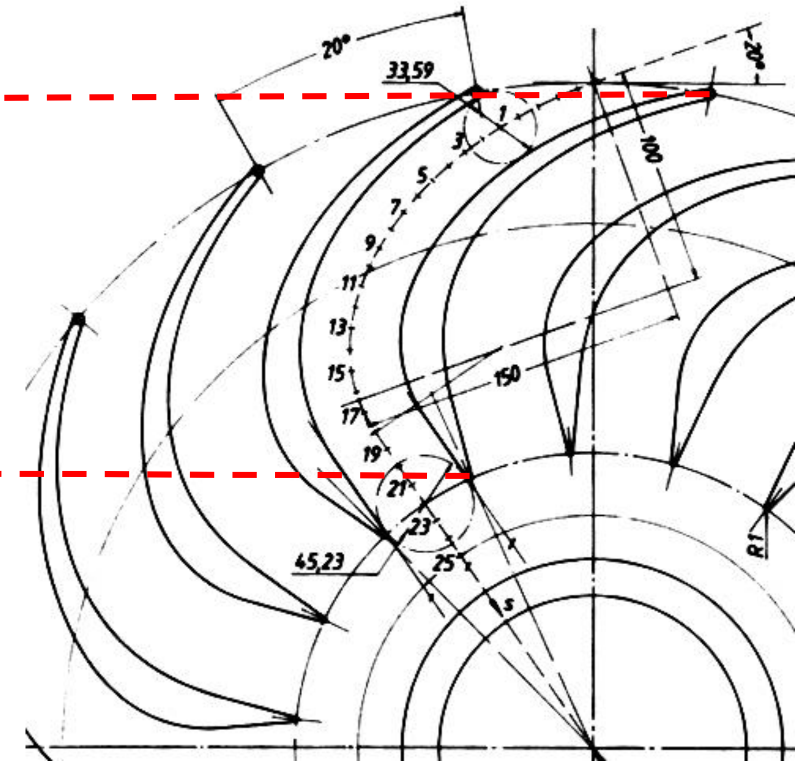


A-C többfokozatú kompresszor

tömegáram: 340 m^3/h –ig,
nyomás: 345 bar-ig

Többfokozatú centrifugál kompresszor összekötő-csatorna lapátozásának tervezése – Ipari alkalmazás





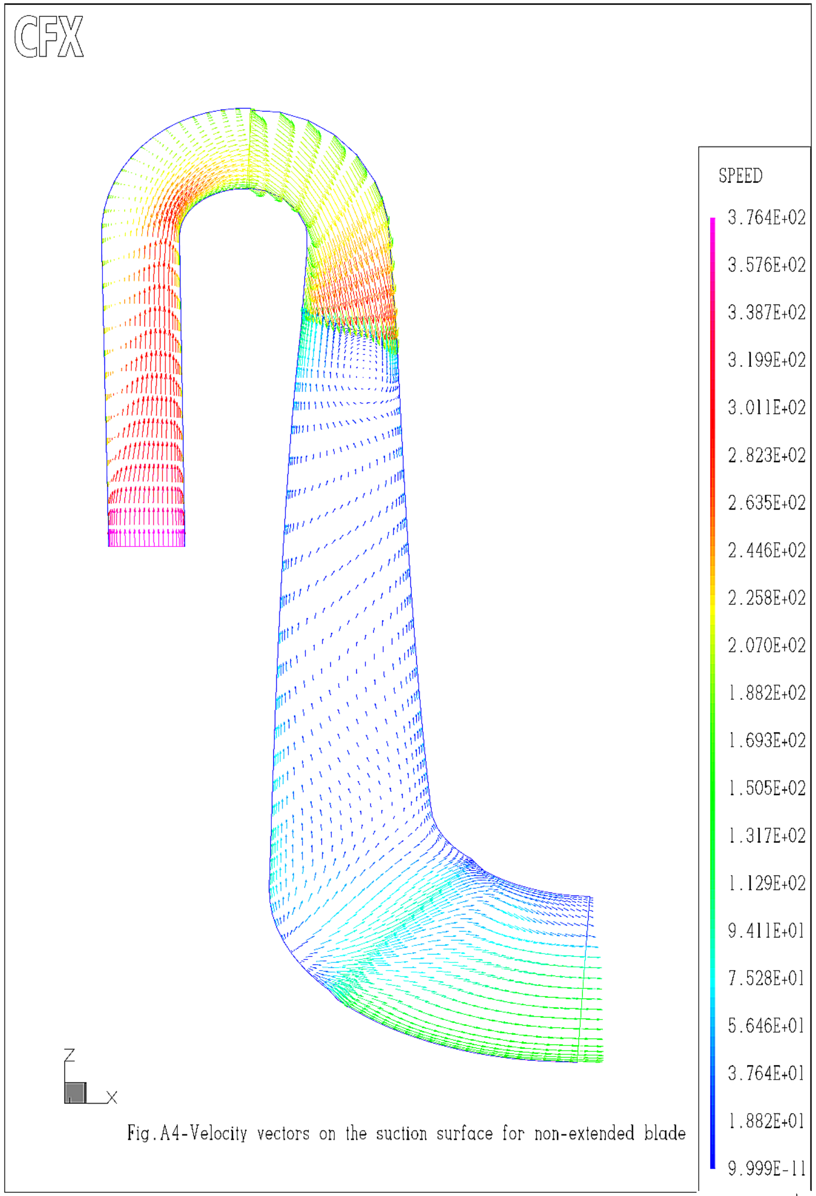


Fig.A4-Velocity vectors on the suction surface for non-extended blade

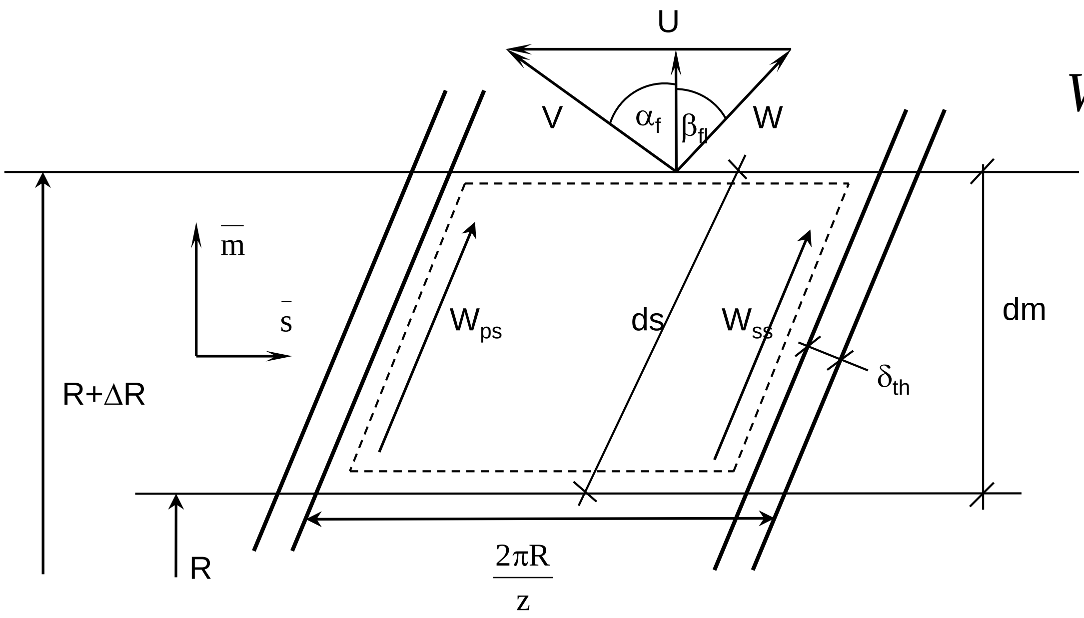
Többfokozatú centrifugál kompresszor összekötő-csatorna lapátozásának tervezése – lapátkiterjesztés – Ipari alkalmazás

$$\omega = \frac{\overline{p}_{in}^{to} - \overline{p}_{out}^{to}}{\overline{p}_{in}^{to} - \overline{p}_{in}^{st}}$$

$$C_p = \frac{\overline{p}_{out}^{st} - \overline{p}_{in}^{st}}{\overline{p}_{in}^{to} - \overline{p}_{in}^{st}}$$

Többfokozatú centrifugál kompresszor összekötő-csatorna lapátozásának tervezése – ÁLT – Ipari alkalmazás

$$\nabla \times \overline{V} = 0 \longrightarrow W_{ps} - W_{ss} = \cos\beta_{bl}\left(\frac{2\pi}{z} - \frac{\delta_{th}}{R\cos\beta_{bl}}\right)\frac{d}{dm}\left(W_m . R . tg\beta_{fl}\right)$$

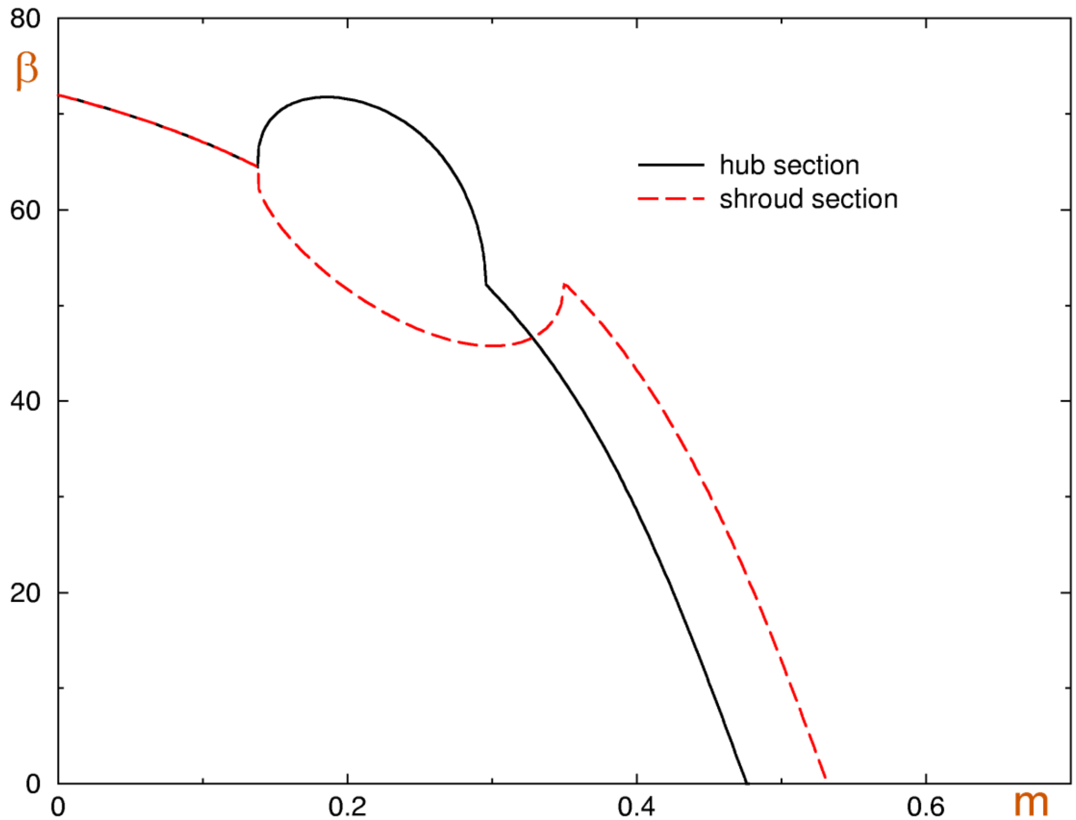


$$W_{ps} - W_{ss} = \left(\frac{2\pi}{z}\right)\frac{C^{t1}}{\cos\ \beta}\frac{d\beta}{dm}$$

$$R.W_m = C^{t1}$$

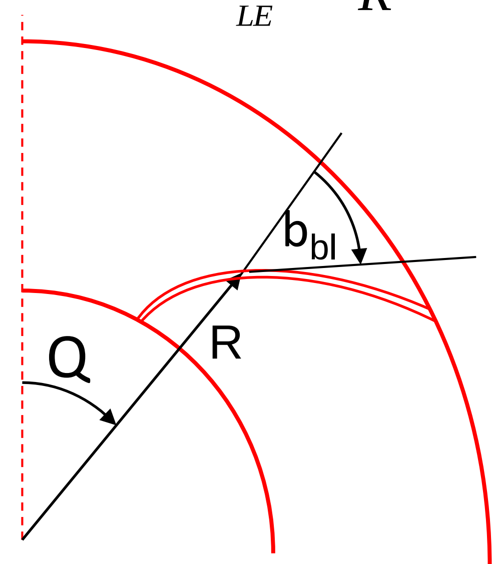
$$\boxed{\beta_{bl} = \frac{C^{t1}}{C^{t2}}\int_{LE}^{TE}\cos\beta\, dm}$$

Többfokozatú centrifugál kompresszor összekötő-csatorna lapátozásának tervezése – ÁLT – Ipari alkalmazás



$$\boxed{\beta_{bl} = \frac{C^{t1}}{C^{t2}} \int_{LE}^{TE} \cos\beta \, dm}$$

$$\theta(m, \beta_{bl}) = \int_{LE}^{TE} \frac{\tan(\beta_{bl})}{R} dm$$



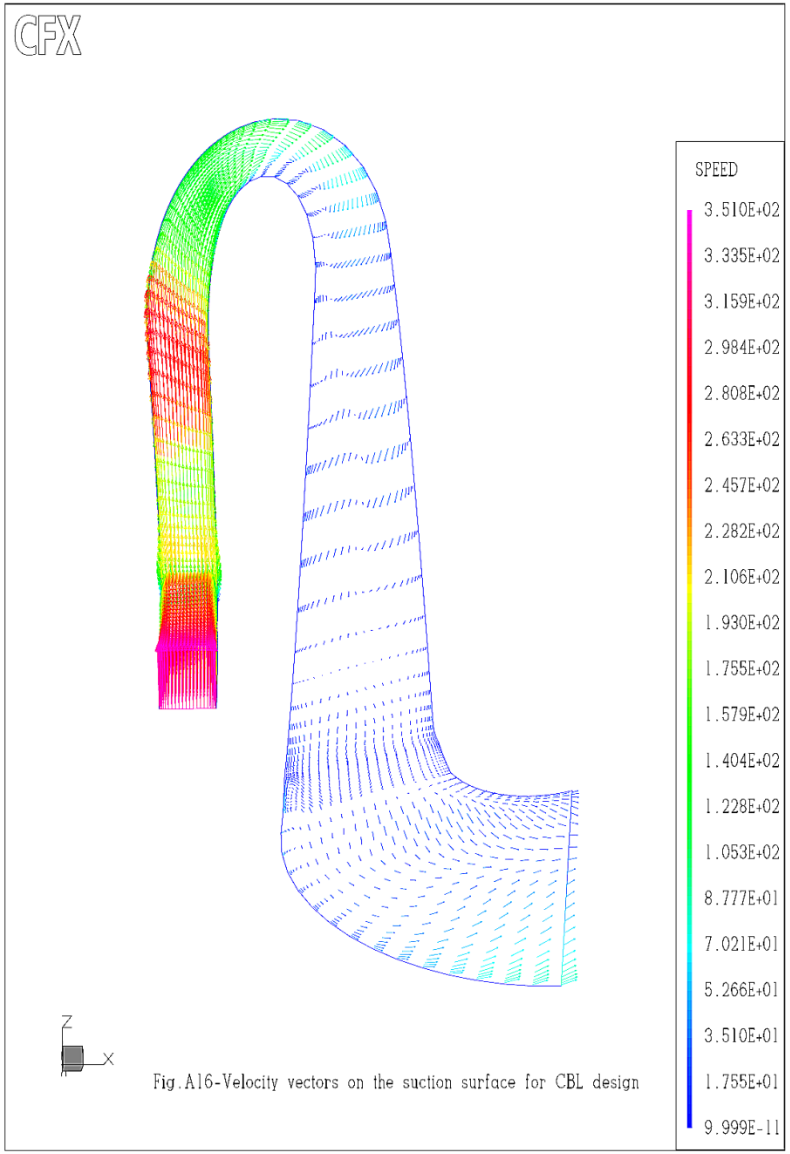


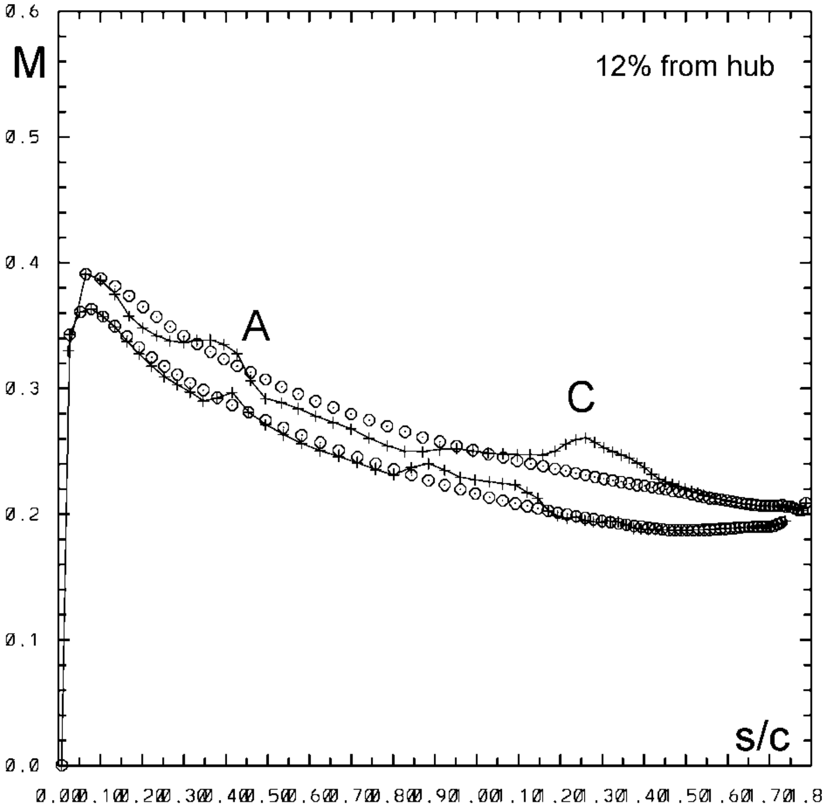
Fig.A16-Velocity vectors on the suction surface for CBL design

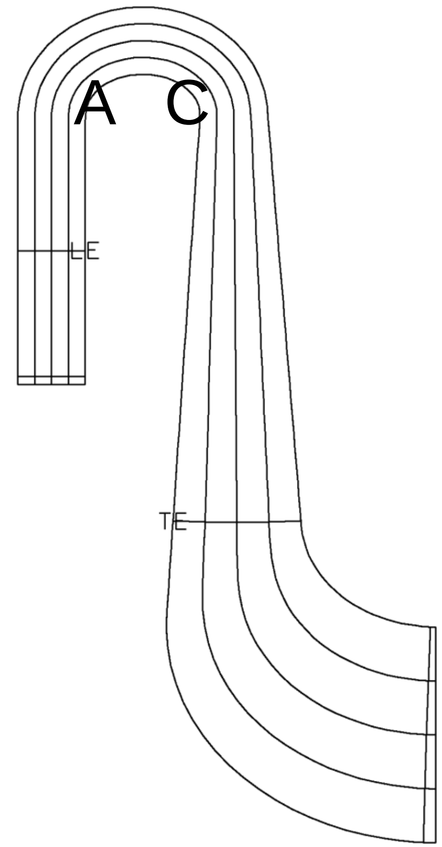
Többfokozatú centrifugál kompresszor összekötő-csatorna lapátozásának tervezése – ÁLT – Ipari alkalmazás

$$\omega = \frac{\overline{p}_{in}^{to} - \overline{p}_{out}^{to}}{\overline{p}_{in}^{to} - \overline{p}_{in}^{st}}$$

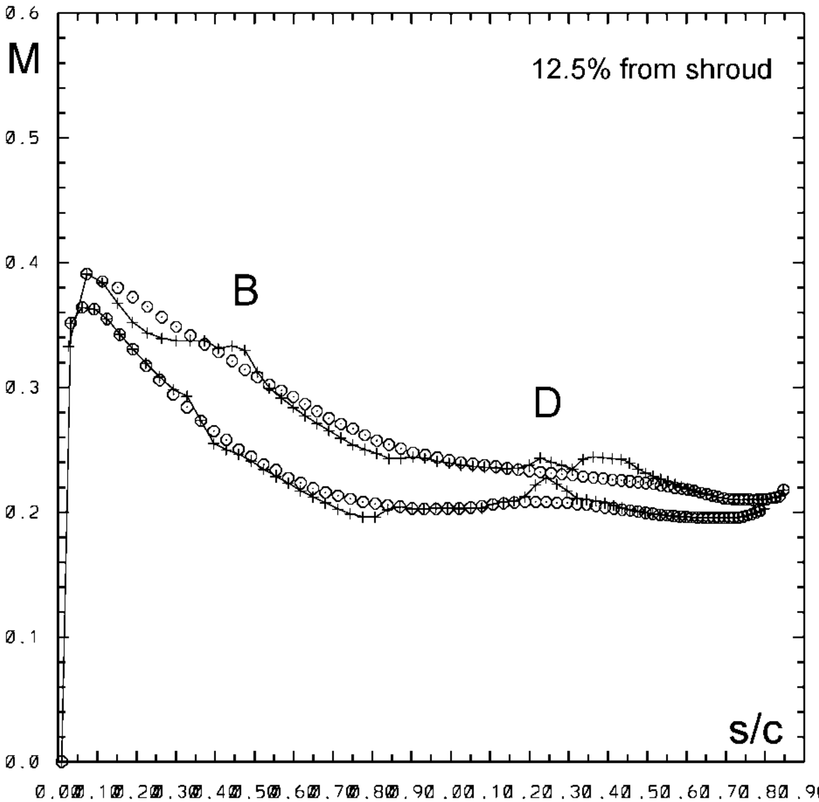
$$C_p = \frac{\overline{p}_{out}^{st} - \overline{p}_{in}^{st}}{\overline{p}_{in}^{to} - \overline{p}_{in}^{st}}$$

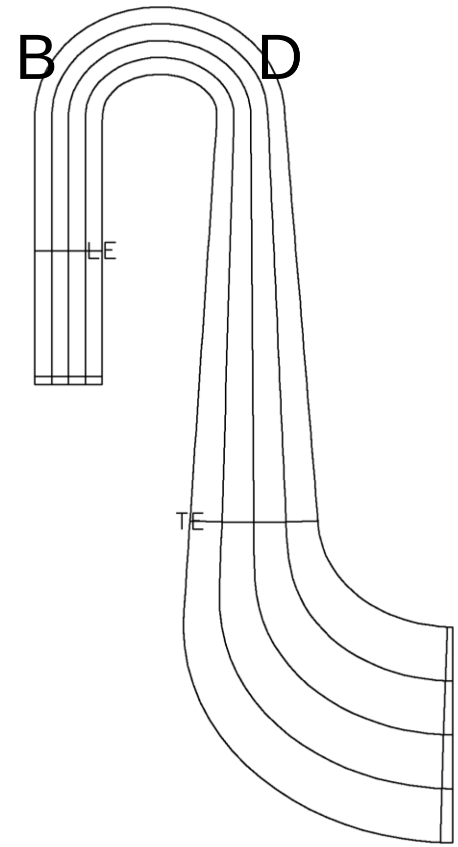
Többfokozatú centrifugál kompresszor összekötő-csatorna lapátozásának tervezése – Inverz Módszer – Ipari alkalmazás



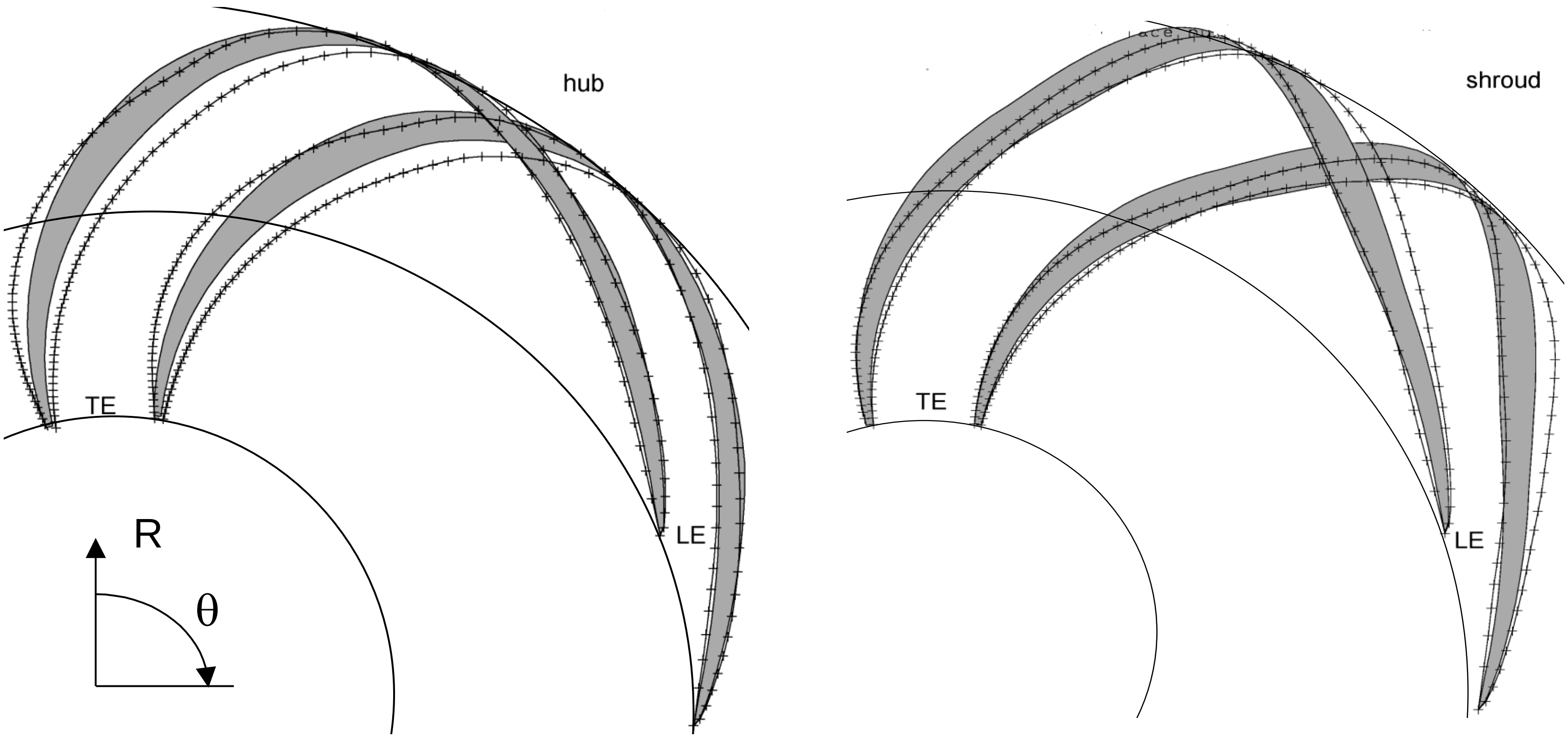


Többfokozatú centrifugál kompresszor összekötő-csatorna lapátozásának tervezése – Inverz Módszer – Ipari alkalmazás





Többfokozatú centrifugál kompresszor összekötő-csatorna lapátozásának tervezése – Inverz Módszer – Ipari alkalmazás



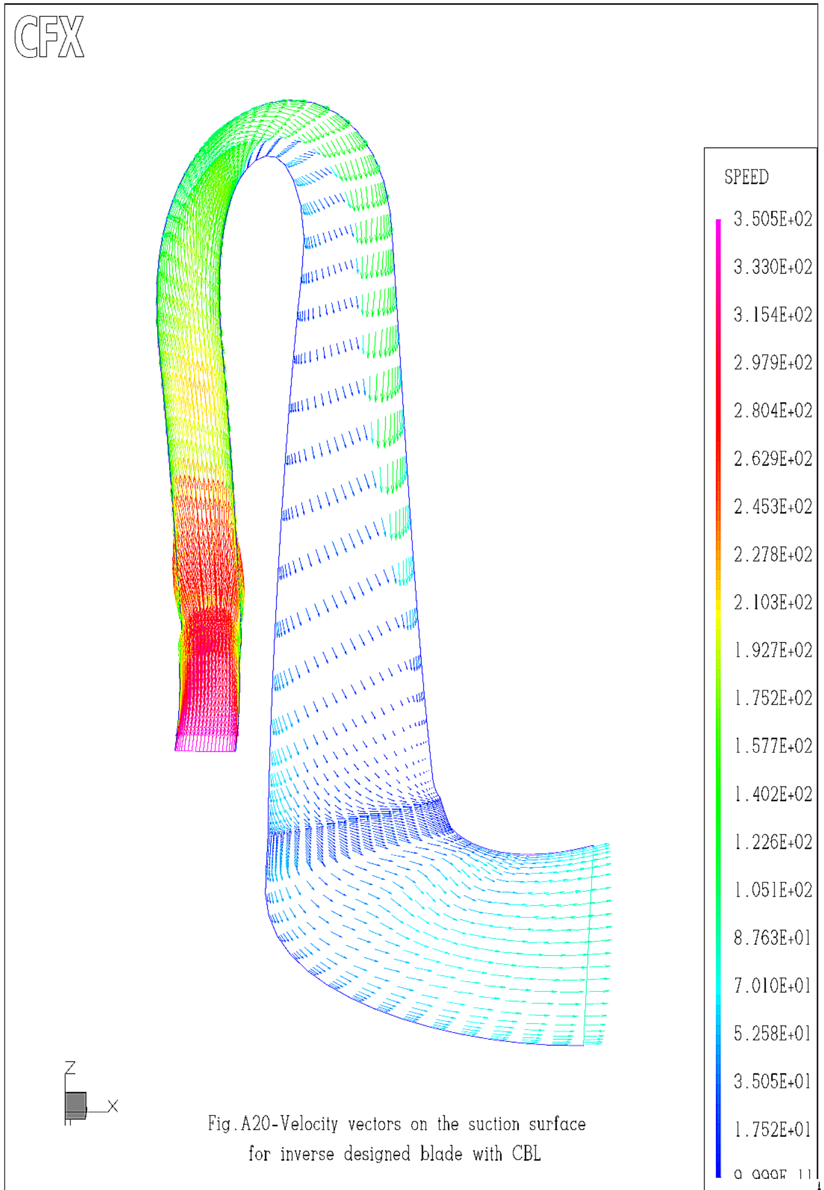


Fig.A20-Velocity vectors on the suction surface for inverse designed blade with CBL

Többfokozatú centrifugál kompresszor összekötő-csatorna lapátozásának tervezése – Inverz Módszer – Ipari alkalmazás

$$\omega = \frac{\bar{p}_{in}^{to} - \bar{p}_{out}^{to}}{\bar{p}_{in}^{to} - \bar{p}_{in}^{st}}$$

$$C_p = \frac{\bar{p}_{out}^{st} - \bar{p}_{in}^{st}}{\bar{p}_{in}^{to} - \bar{p}_{in}^{st}}$$

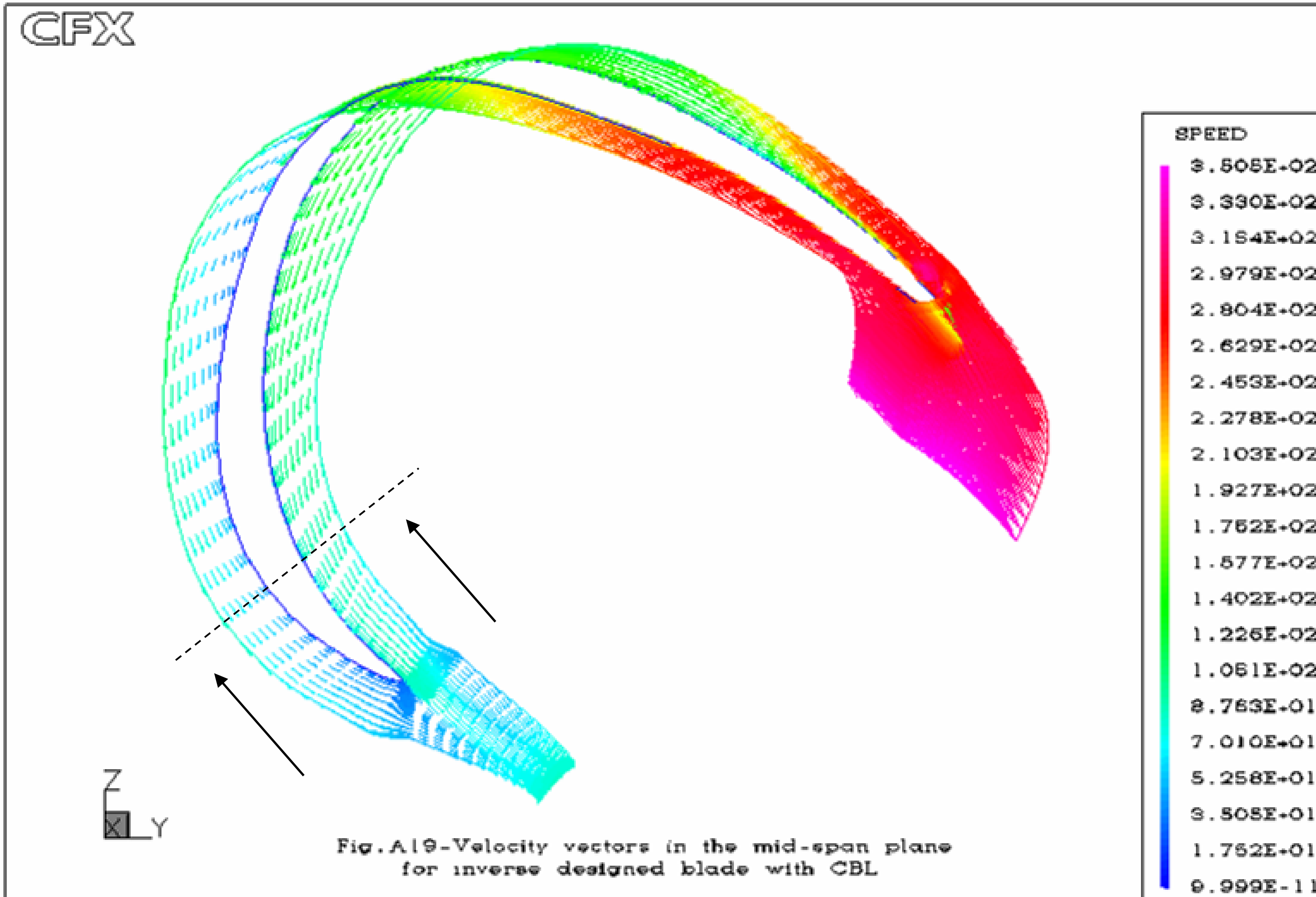
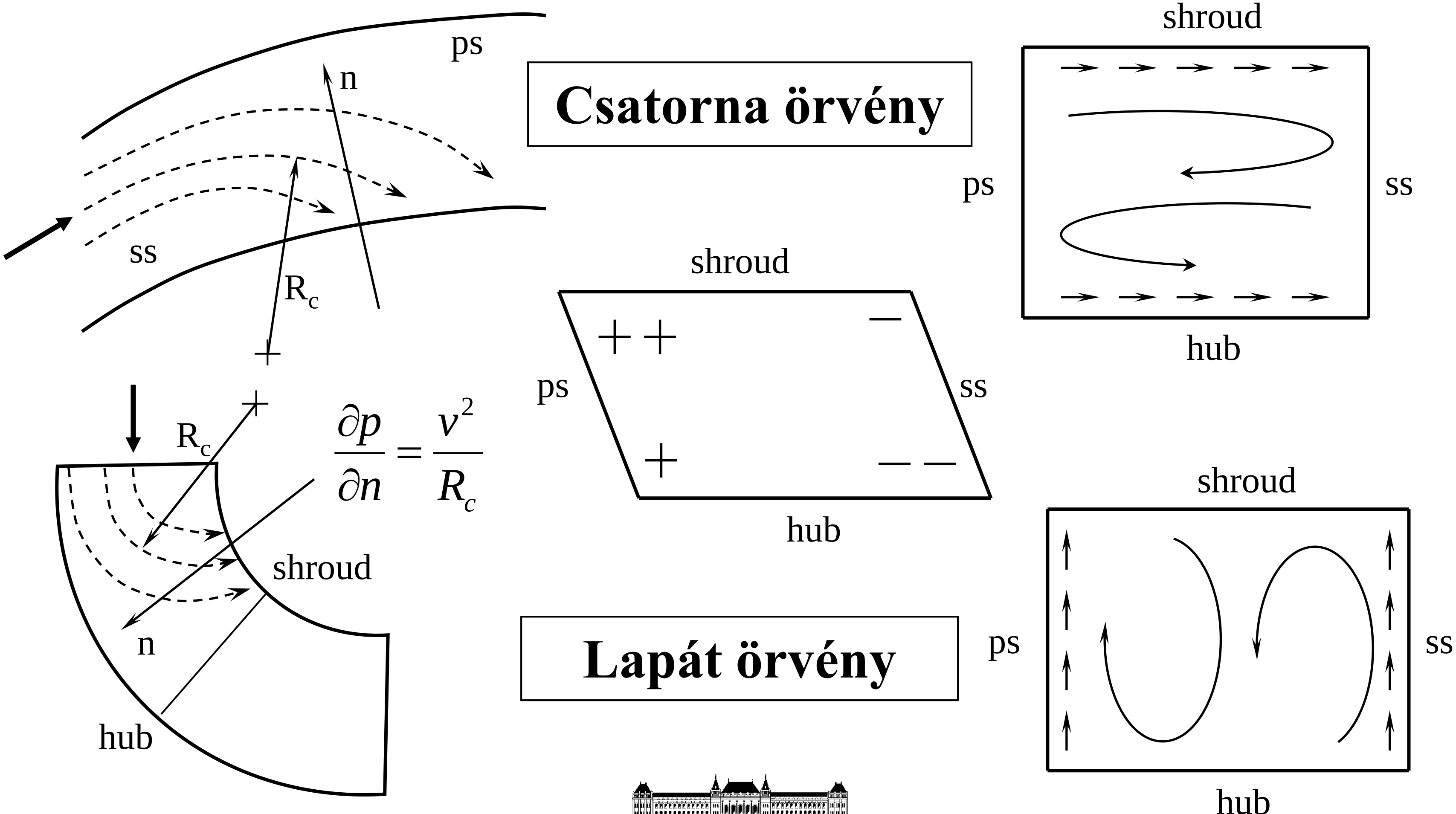


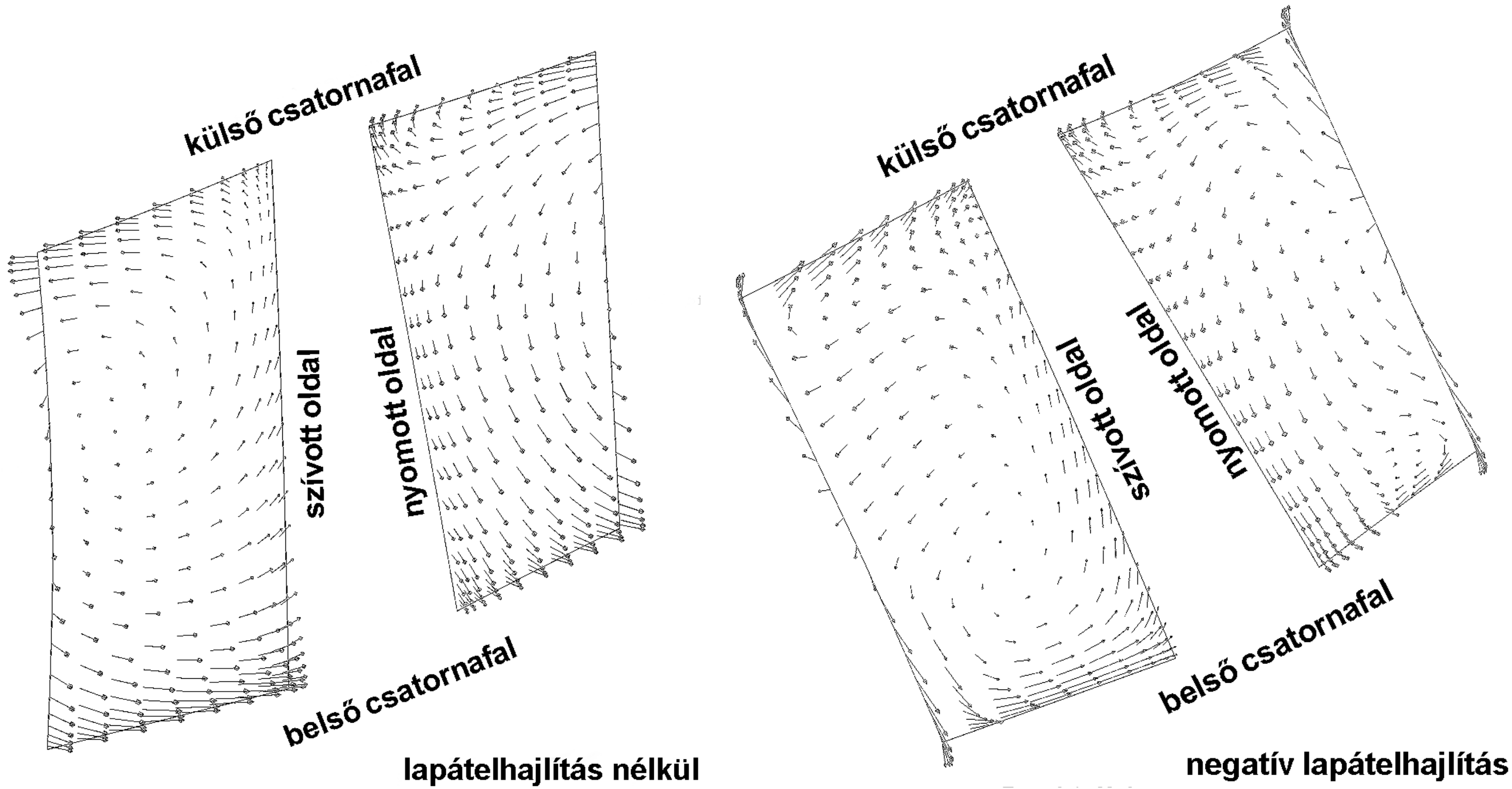
Fig.A19-Velocity vectors in the mid-span plane
for inverse designed blade with CBL

Többfokozatú centrifugál kompresszor összekötő-csatorna lapátozásának tervezése – Lapátelhajlítás – Ipari alkalmazás



$$\frac{\partial p}{\partial n} = \frac{v^2}{R_c}$$

Többfokozatú centrifugál kompresszor összekötő-csatorna lapátozásának tervezése – Lapátelhajlítás – Ipari alkalmazás



Többfokozatú centrifugál kompresszor összekötő-csatorna lapátozásának tervezése – Eredmények - Ipari alkalmazás

Tervezés	Nem kiterjesztett	ÁLT	ÁLT + Inverz Tervezés	+ negatív lapátelh.
P_2^o [Pa]	299699.1	299526.6	299696.6	299698.5
P_2^s [Pa]	159038.5	182298.2	174936.4	169050.8
P_3^o [Pa]	236665.3	261108.8	264954.2	265275.3
P_3^s [Pa]	225215.1	258238.	258414.2	258119.0
$\boldsymbol{\omega}$	**0.44813**	**0.3277**	**0.27847**	**0.263**
$\boldsymbol{C_p}$	**0.4705**	**0.6478**	**0.669106**	**0.681**
[kg/s]	4.68	4.5	4.64	4.5

Összefoglalás

- A numerikus áramlástani módszerek segítségével jobban megérthetők a fizikai folyamatok többek között a vizualizációs eszközöknek köszönhetően.
- Kapcsolt fizikai folyamatok modellezése is lehetséges elfogadható számítógépi kapacitással.
- A numerikus módszereket optimalizációs algoritmusokkal is lehet csatolni.
- Alkalmazásukkal jelentős költség- és kapacitás-csökkenés érhető el.
- Kivitelezhetetlen, extrém körülmények közötti, illetve nagy költségű mérések kiváltására is alkalmas.
- Az analízisek paraméterezhetőek, könnyen megismételhetőek minimális ráfordítással az előírt geometriai változtatásokat követően.
- A számítási eredmények validációjára és paraméter-érzékenységi vizsgálatok elvégzésére minden esetben szükség van.

Forrás: ANSYS, Inc., *ANSYS CFX-Solver Theory Guide, Release 14.5*, ANSYS, Inc. Southpointe, 275 Technology Derive Canonsburg, PA 15317, ansysinfo@ansys.com, http://www.ansys.com, USA, 2012

Forrás: ANSYS, Inc., *ANSYS CFX-Solver Theory Guide, Release 13*, ANSYS, Inc. Southpointe, 275 Technology Derive Canonsburg, PA 15317, ansysinfo@ansys.com, http://www.ansys.com, USA, 2010

Thank you for your kind attentions.

BME, Vasúti Járművek, Repülőgépek és Hajók Tanszék
Stoczek u. 6. J. ép. 4. em. 426
H-1111, Budapest
Telefon: +36 1 463-1922
Fax: +36 1 463-3080
e-mail: averess@vrht.bme.hu