



Budapesti Műszaki és Gazdaságtudományi Egyetem
Vasúti Járművek, Repülőgépek és Hajók Tanszék
Budapest, 1111, Sztoczek u. 6. J. ép. 4. em.
Tel: +36-1-463-1922, Fax: +36-1-463-30-80

Hő- és áramlástanai számítások

BMEKOV RM606

Dr. Veress Árpád
egyetemi docens

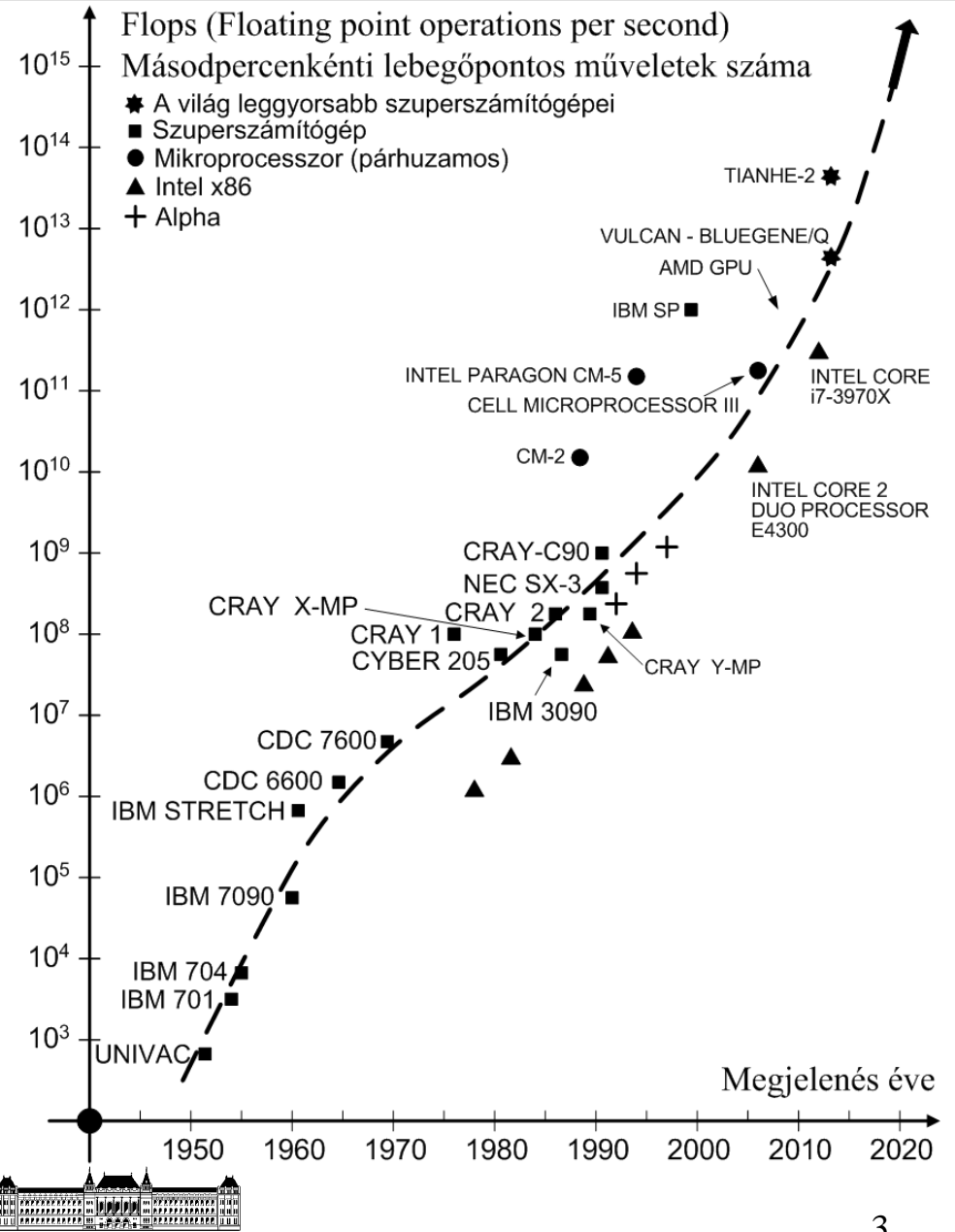
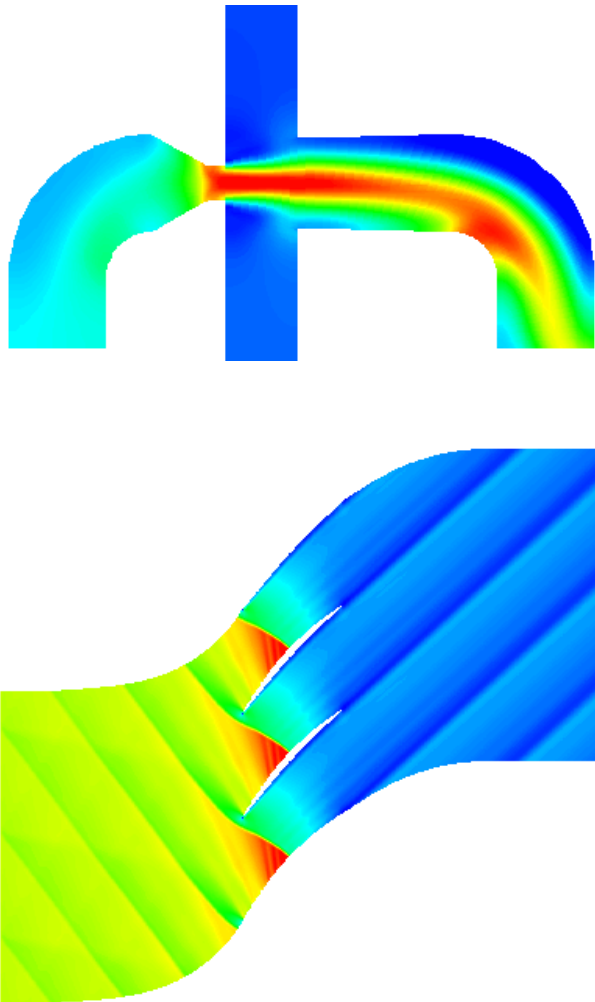
Budapest, 2020-09-07



Bevezetés



Aktualitás

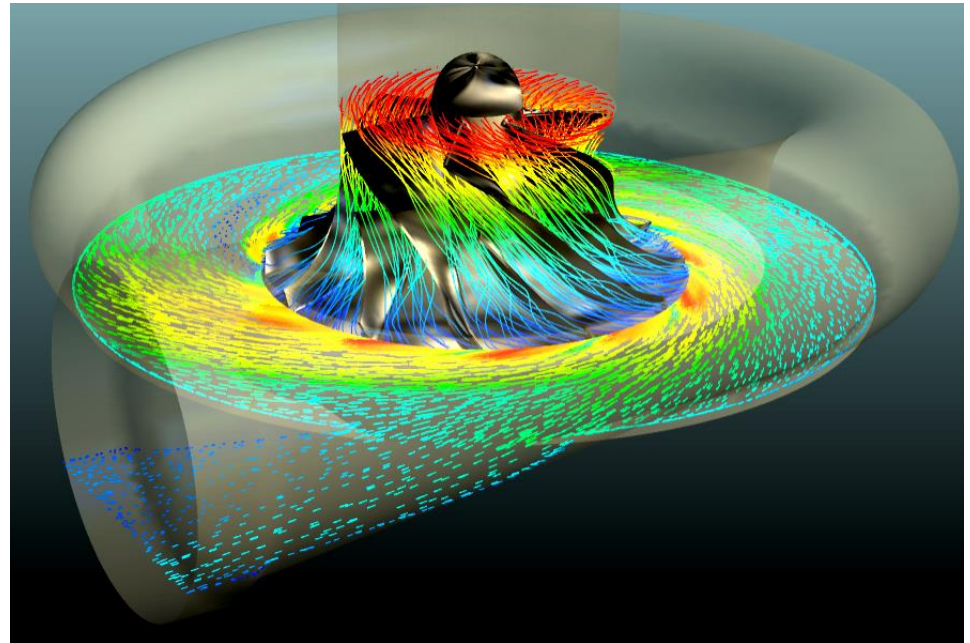
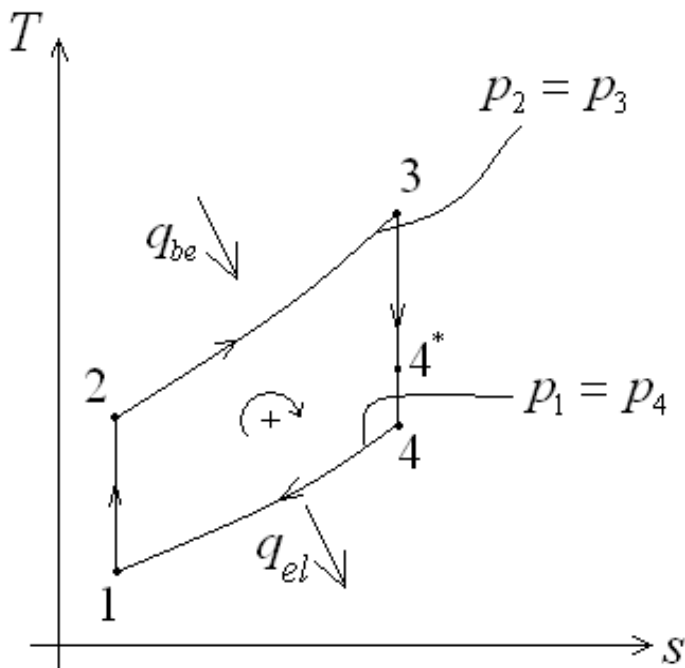


Közelítési elvek és alkalmazhatósági feltételek

Fizikai folyamatok leírása → mérnöki számítások → alapvető közelítési elvek :

1. Koncentrált paraméterű

2. Elosztott paraméterű



Termodinamika, hőközlés és áramlástan → Egyenletek alkalmazhatóságának vizsgálata



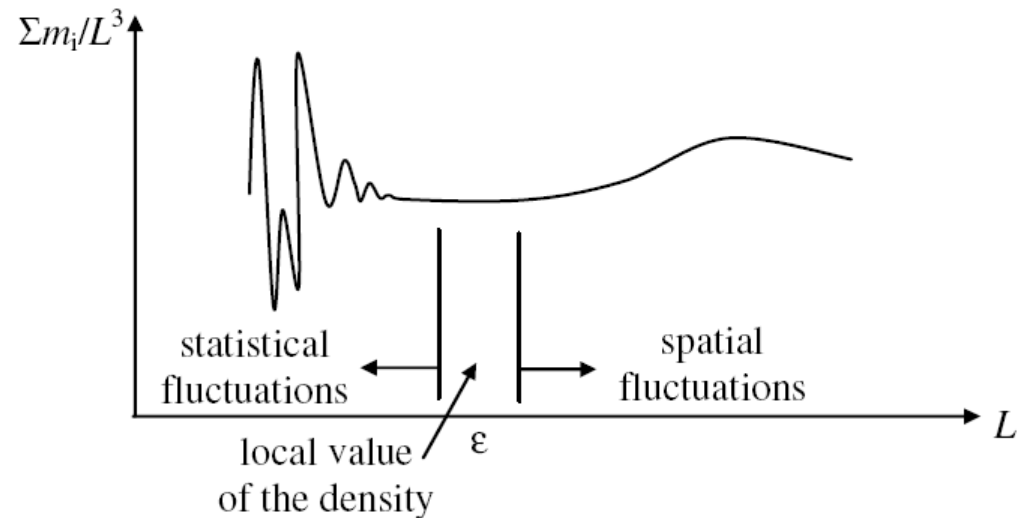
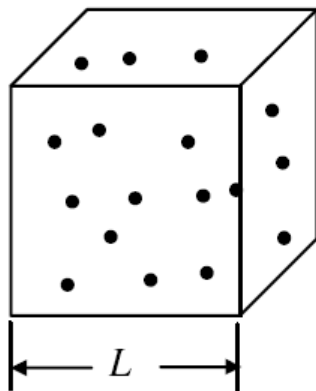
Közelítési elvek és alkalmazhatósági feltételek

Fizikai folyamatok leírása → alapvető közelítési elvek :

1. Statisztikai (Statisztikus mechanika, Kinetikus gázelmélet)
2. Kontinuum mechanikai

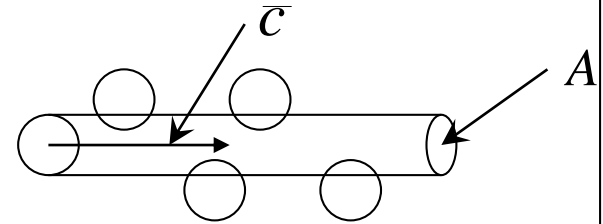
Jellemző hossz mérték – Length scale

$$\rho \equiv \lim_{L \rightarrow \epsilon} \frac{\sum m_i}{L^3}$$



Közelítési elvek és alkalmazhatósági feltételek

Jellemző hossz mérték – Length scale



Egy molekula által időegység alatt végigsöpört térfogat: $\dot{V} = \bar{c}A$ [m^3 / s]

Ekkor másik molekulával időegység alatt létrejött ütközések száma:

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

Két ütközés között eltelt átlagos idő: $t' = 1/\dot{n} = 1/(n\dot{V}) = 1/(n\bar{c}A)$ [s]

A két ütközés alatt megtett átlagos úthossz (a molekulák szabad köz. úthossza):

$$\lambda = \bar{c}t' = 1/(nA) \quad [m] \quad \text{Levegőre standard állapot esetén: } n = 2,7e19 \left[\frac{db}{cm^3} \right]$$

$$160 \text{ Km magasan: } \lambda = 80[m]$$

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

$$\text{Lökéshullámban: } \lambda = 1[\mu m] = 1e-4 [cm]$$

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

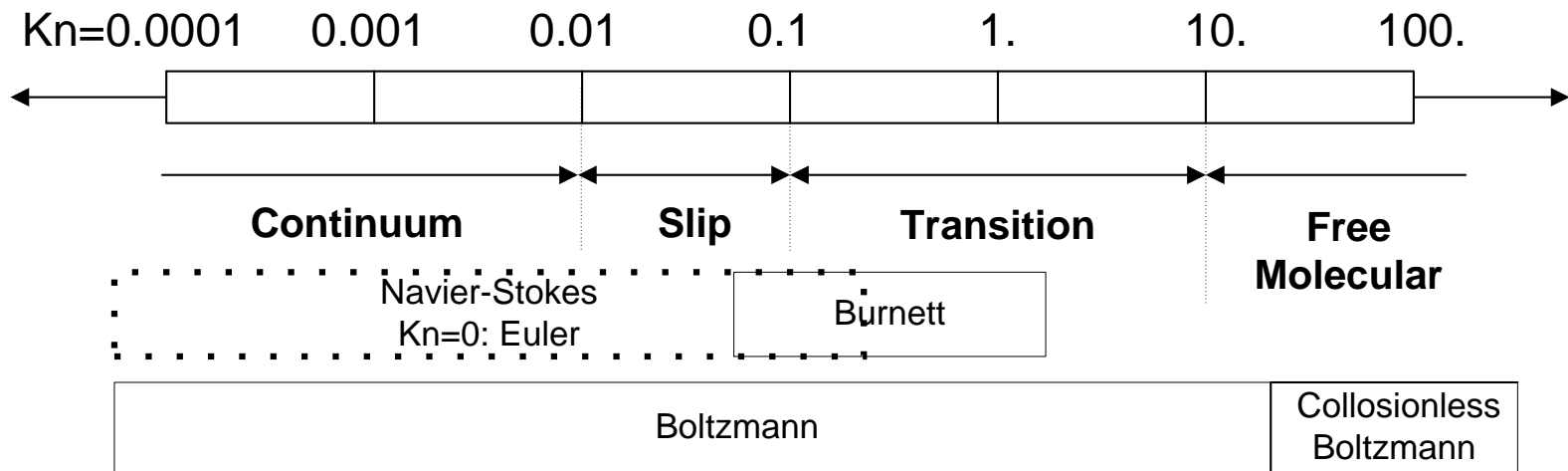


Közelítési elvek és alkalmazhatósági feltételek

Áramlások osztályozása a lokális Knudsen szám alapján

1. Statisztikai (Statisztikus mechanika, Kinetikus gázelmélet)
2. Kontinuum mechanikai

A Knudsen szám jelentősége a kontinuum mechanikai közelítés alkalmazhatóságában:

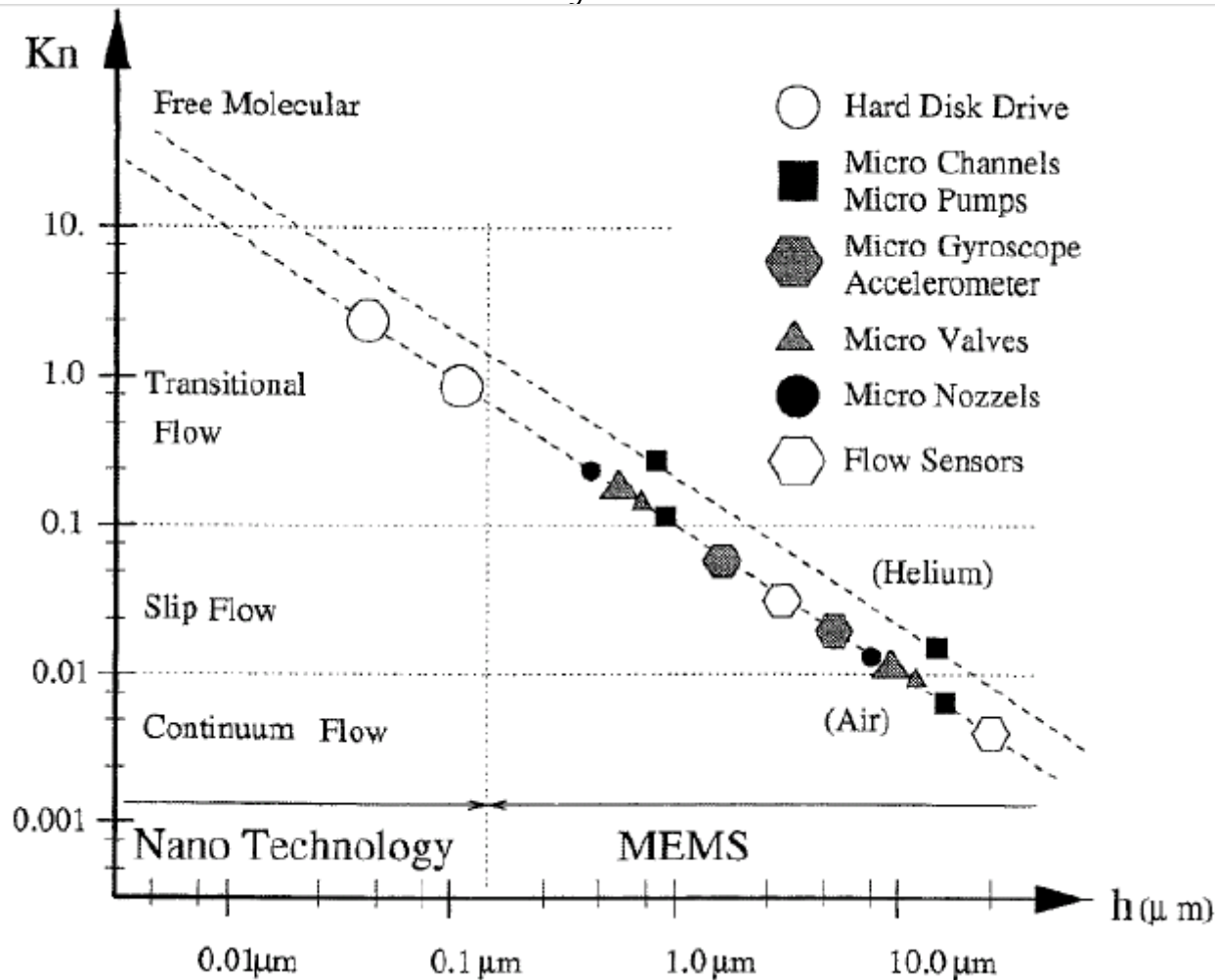


$$\text{Kn} = \frac{\lambda}{L} = \frac{3,7e-7[\text{m}]}{0,03[\text{m}]} \ll 0,01$$



Közelítési elvek és alkalmazhatósági feltételek

Áramlások osztályozása a lokális Knudsen szám alapján



Forrás: **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.)



Áramlások matematikai modelljei



Áramlásmodellezés – Kontinuum-mechanika alapján

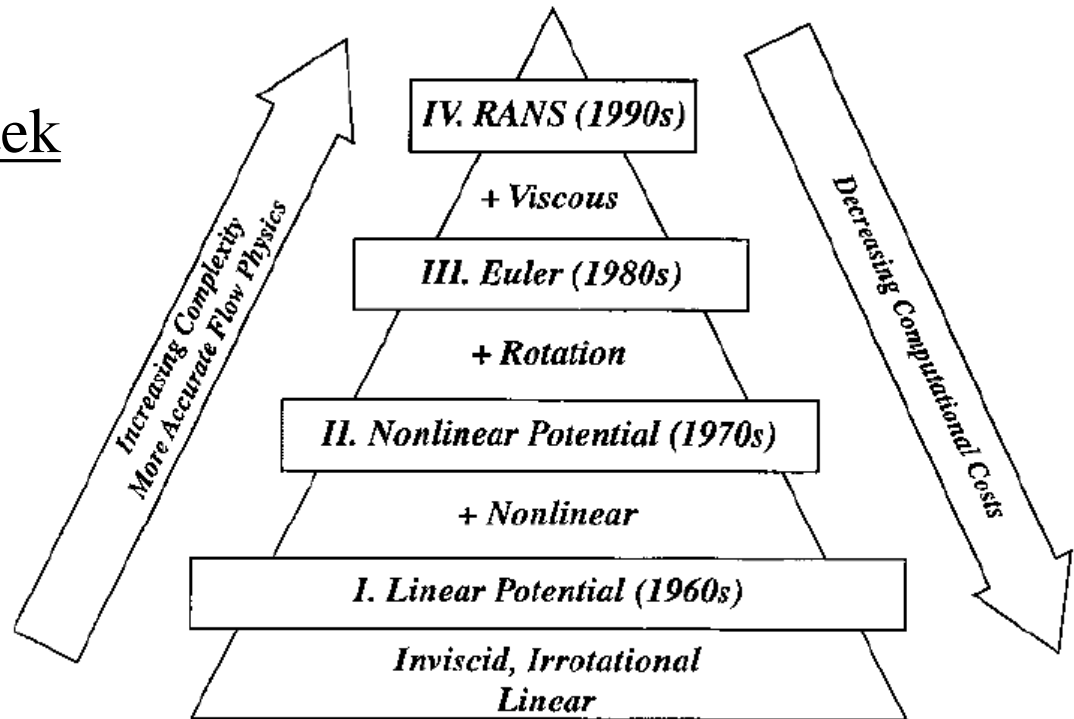
Navier-Stokes egyenletek

*Ideális =
Súrlódásmentes =
Nem viszkózus*

Euler egyenletek

*Izentrópikus,
örvénymentes*

Potenciál egyenletek



Hierarchy of Fluid Flow Models

Forrás: 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.)



Áramlásmodellezés – Kontinuum-mechanika alapján

Navier-Stokes egyenletek

Forrás: VKI-LS, Introduction to CFD



Lagrange és Euler közelítési elvek

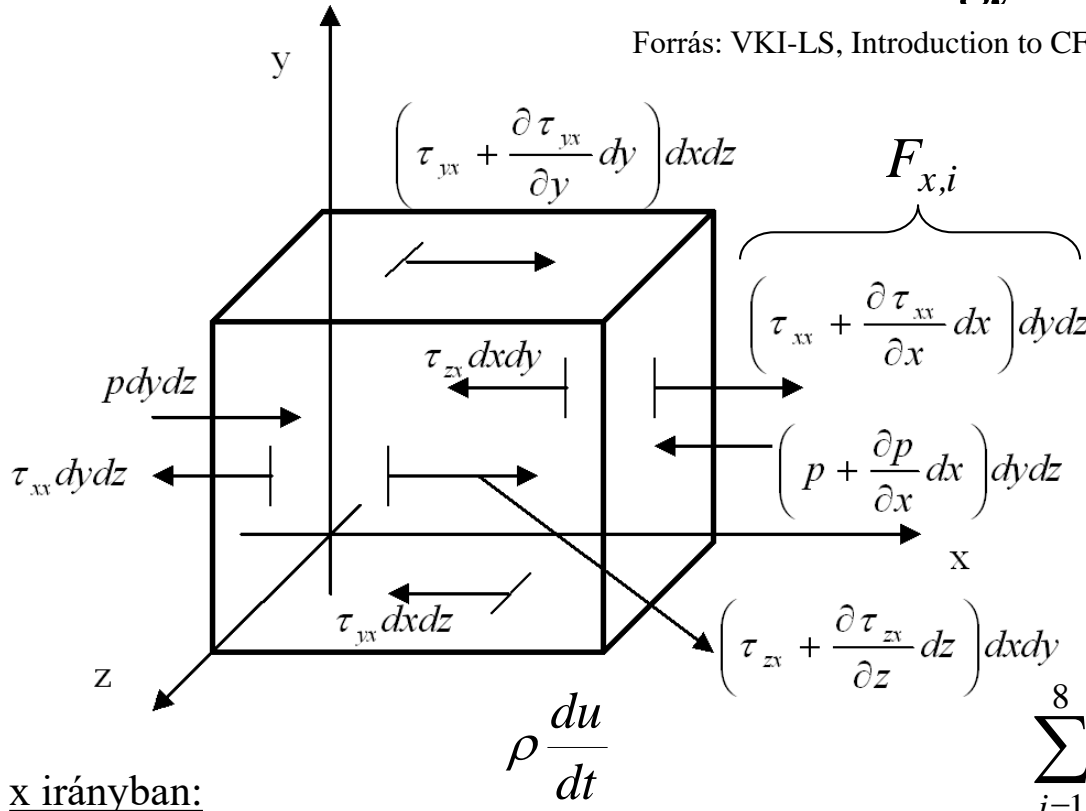
Newton II. törvénye x irányban:

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

$$m = \rho dx dy dz$$

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

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



x irányban:

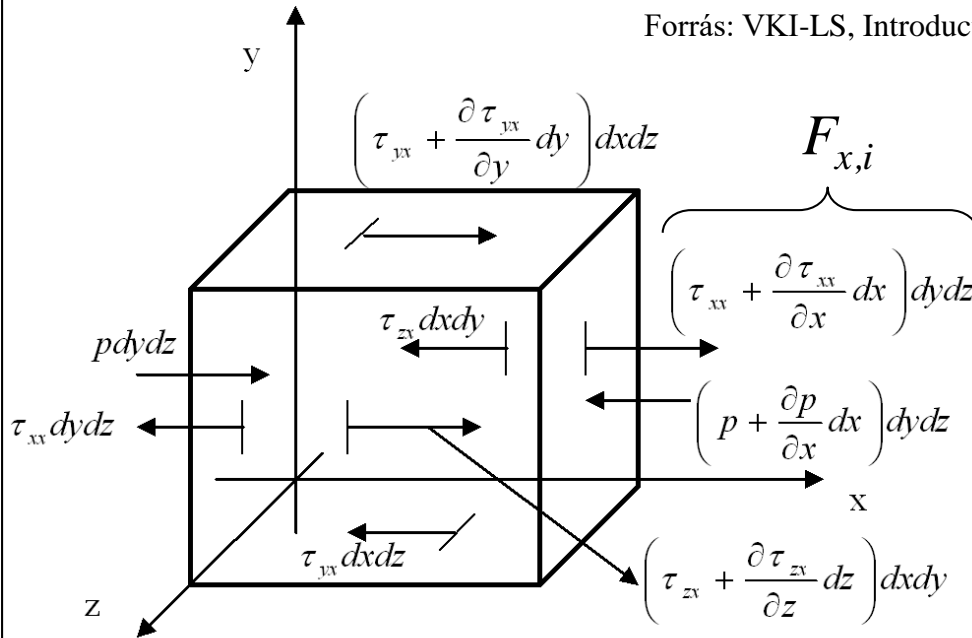
$$dx dy dz \left(\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} + \frac{\partial(\rho uw)}{\partial z} \right) = \left(-\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} \right) dx dy dz$$



Áramlásmodellezés – Kontinuum-mechanika alapján

Navier-Stokes egyenletek

Forrás: VKI-LS, Introduction to CFD



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

Mivel:

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

és:

$$\rho \bar{V} \nabla u = \nabla(\rho u \bar{V}) - u \nabla(\rho \bar{V})$$

Ezért:

$$\rho \frac{du}{dt} = \frac{\partial(\rho u)}{\partial t} - u \left[\frac{\partial \rho}{\partial t} + \nabla(\rho \bar{V}) \right] + \nabla(\rho u \bar{V})$$

0 (kontinuitás)

x irányban:

$$\rho \frac{du}{dt}$$

$$\sum_{i=1}^8 F_{x,i}$$

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

$$dx dy dz \left(\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} + \frac{\partial(\rho uw)}{\partial z} \right) = \left(-\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} \right) dx dy dz$$



Áramlásmodellezés – Kontinuum-mechanika alapján

Navier-Stokes egyenletek

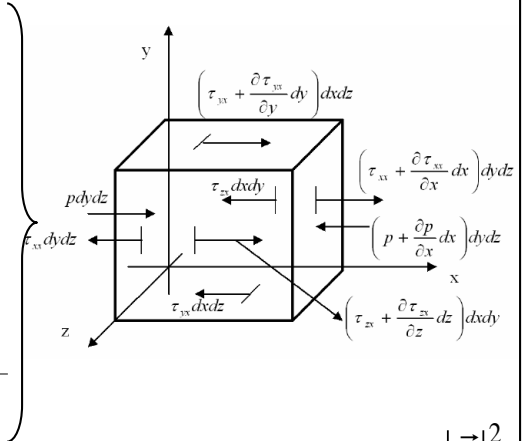
$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \quad \longrightarrow \quad \frac{\partial(\rho u)}{\partial x} = 0 \quad \xrightarrow{\iiint_V} \quad \iint_A \rho u dA = \rho u A = \text{Állandó}$$

$$\boxed{F = ma}$$

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2 + p)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} + \frac{\partial(\rho uw)}{\partial z} = \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z}$$

$$\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho vu)}{\partial x} + \frac{\partial(\rho v^2 + p)}{\partial y} + \frac{\partial(\rho vw)}{\partial z} = \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z}$$

$$\frac{\partial(\rho w)}{\partial t} + \frac{\partial(\rho wu)}{\partial x} + \frac{\partial(\rho wv)}{\partial y} + \frac{\partial(\rho w^2 + p)}{\partial z} = \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z}$$



$$\boxed{P = Fv}$$

$$\frac{\partial(\rho E)}{\partial t} + \frac{\partial(\rho u H)}{\partial x} + \frac{\partial(\rho v H)}{\partial y} + \frac{\partial(\rho w H)}{\partial z} = \frac{\partial(u \tau_{xx} + v \tau_{xy} + w \tau_{xz} + k \partial T / \partial x)}{\partial x} + \frac{\partial(u \tau_{yx} + v \tau_{yy} + w \tau_{yz} + k \partial T / \partial y)}{\partial y} + \frac{\partial(u \tau_{zx} + v \tau_{zy} + w \tau_{zz} + k \partial T / \partial z)}{\partial z}$$

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

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

$$\downarrow p = \rho R T$$



Áramlásmodellezés – Kontinuum-mechanika alapján

Navier-Stokes egyenletek

Az előző oldalon felírt nemlináris parciális differenciál-egyenletrendszer (a matematikai modell) érvényességi (alkalmazhatósági) feltételei fizikai szempontból:

- Összenyomható ideális gáz relatív, nyugvó rendszerben,
- Folytonos anyag (kontinuum), lehet lamináris és turbulens áramlás is,
- Homogén (egyféle anyag) izotróp anyagtulajdonsággal,
- Tranziens folyamatok figyelembevétele,
- Valóságos (súrlódásos) áramlás,
- Erőtérmentes áramlás (nincs gravitáció, mágneses tér, stb.),
- Forrás és nyelőmentes áramlás,
- Konzervatív forma → szakadások (kontakt szakadás, örvényvonal és lökéshullám) leírására alkalmas.

$$v_{n,1} - v_{n,2} = [v_n] = 0; [p] = 0;$$

$$[\rho] \neq 0; [v_t] = 0;$$

$$[v_n] = 0; [p] = 0;$$

$$[\rho] \neq 0; [v_t] \neq 0;$$

$$[v_n] \neq 0; [p] \neq 0;$$

$$[\rho] \neq 0; [v_t] = 0;$$



CFD



Mi az a CFD (Computational Fluid Dynamics) ?

- A CFD a számítógépes áramlás-modellezés rövid angol elnevezése
- Az áramlás matematikai-fizikai modellezése számítógépek segítségével
- A modern tervezési folyamatok alapvető összetevője
- Hozzásegít jobb, olcsóbb és biztonságosabb járművekhez, termékekhez és folyamatokhoz
- Hatékony eszköz lehet el nem végezhető, költséges folyamatok szimulációjára
- A verifikáció mellett, érvényessége kísérletekkel, illetve egyéb tesztesetekkel (benchmark) ellenőrizendő



A CFD előnyei

- Az ipari alkalmazások 80 %-ban 5-10% százalék alatti pontossággal képes visszaadni a valóságot.
- A termék teljes életciklusában alkalmazható (az előfejlesztéstől az újrahasznosításig).
- Alkalmazásával jelentős költség, kapacitás és idő takarítható meg a költséges mérésekkel szemben, hiszen a virtuális valóságban módosítani és analizálni egy terméket összehasonlíthatatlanul gyorsabb, mint újra legyártani és mérni.
- A sokféle vizualizációs technikának köszönhetően jobban megérthetők a fizikai jelenségek, mint kísérletek esetén.
- Többféle fizika vizsgálható egyszerre elfogadható számítógépi kapacitás-igénnyel.



A CFD előnyei

- Olyan esetekben is alkalmazható, ahol nem lehet mérni (pl. működő atomerőmű belsejében, idegen bolygók légkörében), illetve a mérőműszer jelentősen megzavarná a mérendő mennyiséggel kapcsolatos folyamatot.
- A numerikus áramlástan számítások parametrizálhatók, könnyen reprodukálhatók és automatizálhatók.
- Az alkalmazott numerikus módszerek optimalizációs algoritmusokkal kapcsolhatók össze
- Egy teljes fejlesztési folyamat nem alapulhat kizárólag számításokra. Validációra mindenképp szükség van.



Fontosabb alkalmazási területek

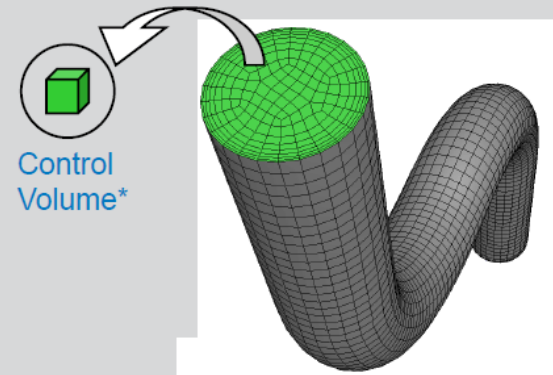
- Gépjárműgyártás: kocsiszekrény aerodinamika, utastér ventiláció, légkondicionálás, motor, stb.
- Repülőipar (külső, belső és lapátrácsba való áramlás)
- Hő- és áramlástan gépek
- Biztonságtechnika: tűz- és füstterjedés előrejelzés, robbanás és egyéb véletlenszerű események modellezése
- Gyártási folyamatok: hatásfoknövelés a könnyű-, a nehéz-, a vegy-, és az élelmiszeriparban
- Meteorológia: hosszú és rövid távú időjárás-előrejelzés, globális klíma modellek
- Környezetvédelem: szennyezés terjedés az atmoszférában
- Épületgépészet: épület fűtés és szellőzés analízis
- Csillagászat



A CFD működésének rövid összefoglalása

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



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

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

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

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

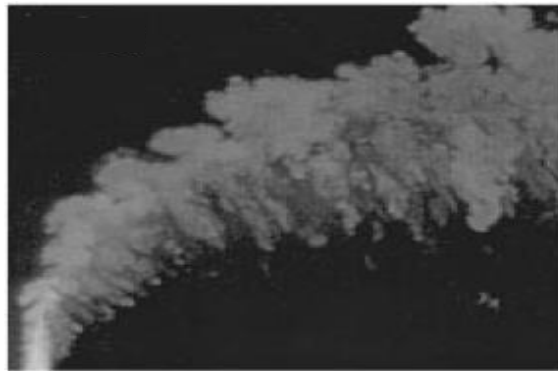


Áramlások matematikai modelljei - Turbulencia



Áramlásmodellezés – Kontinuum-mechanika alapján

Navier-Stokes egyenletek – turbulens áramlás



Áramlásmodellezés – Kontinuum-mechanika alapján

Navier-Stokes egyenletek – turbulens áramlás - elmélet

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

$$Re_L = \frac{\rho \cdot U \cdot L}{\mu}$$

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

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

- Transition to Turbulence varies depending on the type of flow:

- External flow

- along a surface : $Re_x > 500\,000$

- around on obstacle : $Re_L > 20\,000$

- Internal flow

- : $Re_D > 2\,300$

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



Áramlásmodellezés – Kontinuum-mechanika alapján

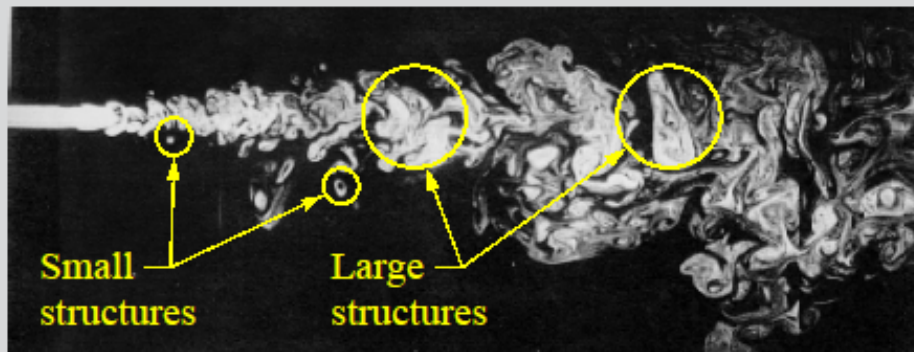
Navier-Stokes egyenletek – turbulens áramlás - elmélet

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.)



Áramlásmodellezés – Kontinuum-mechanika alapján

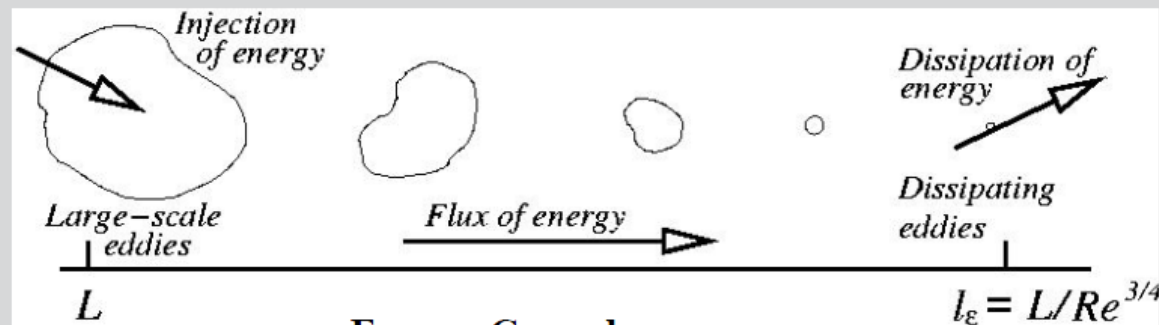
Navier-Stokes egyenletek – turbulens áramlás - elmélet

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.)



Áramlásmodellezés – Kontinuum-mechanika alapján

Navier-Stokes egyenletek – turbulens áramlás - elmélet

Turbulent Flow Structures

- Characteristics of the Turbulent Structures:

- Length scale : l [m]
- Velocity scale : \sqrt{k} [m/s]
- Time scale : $\frac{l}{\sqrt{k}}$ [s]
- Shape (non-isotropic larger structures)

- Turbulent kinetic energy : $k = \frac{1}{2} (\overline{u'^2} + \overline{v'^2} + \overline{w'^2})$ [m²/s²]

- Turbulent kinetic energy dissipation : ε [m²/s³] $\sim k^{3/2}/l$ (dimensional analysis)

- Turbulent Reynolds : $Re_t = k^{1/2} \cdot l / \nu \sim k^2 / \nu \varepsilon$ [-]

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

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

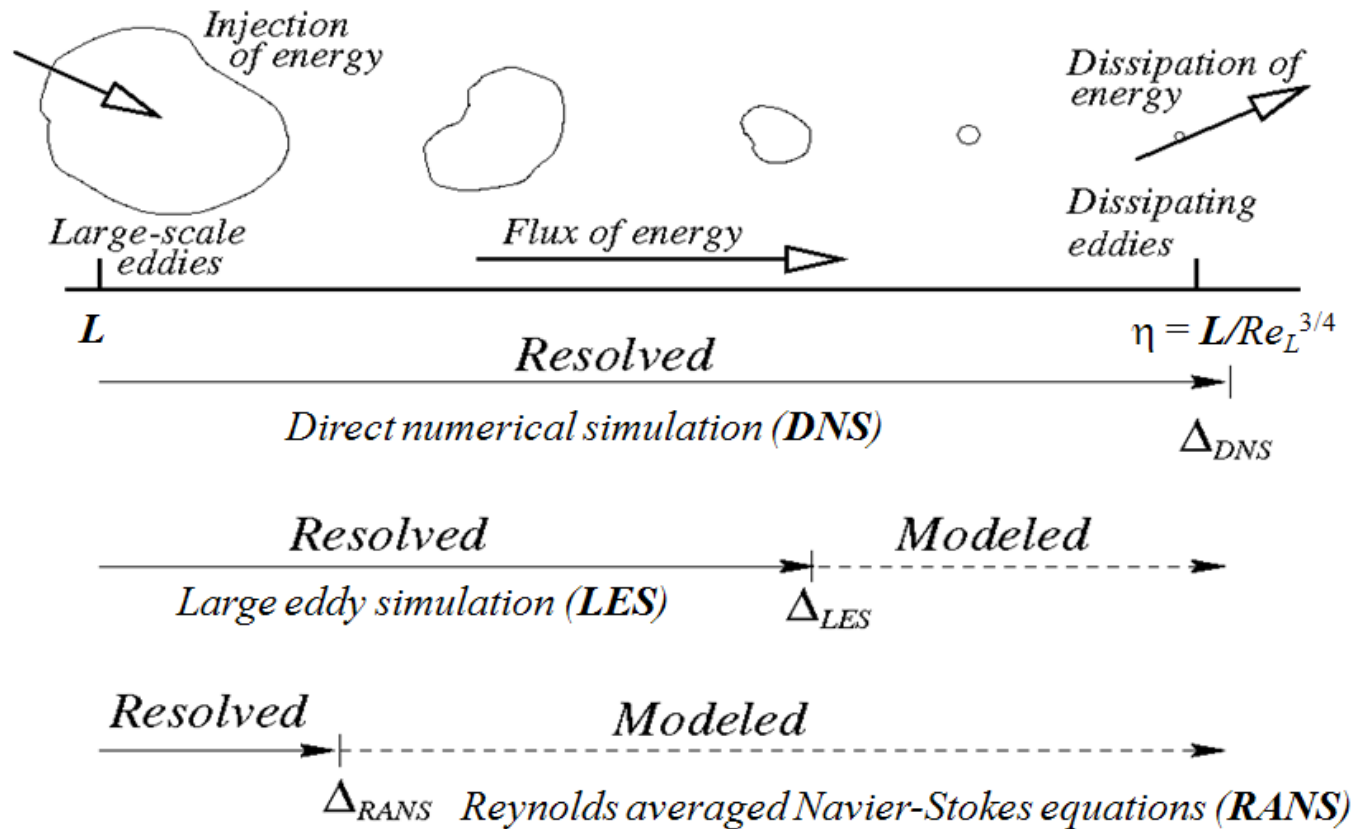
↑ Instantaneous component ↑ Time-average component ↑ Fluctuating component

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



Áramlásmodellezés – Kontinuum-mechanika alapján

NS egyenletek – turbulencia modellezés - elmélet



Forrás:

Jurij SODJA : Turbulence models in CFD

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



Áramlásmodellezés – Kontinuum-mechanika alapján

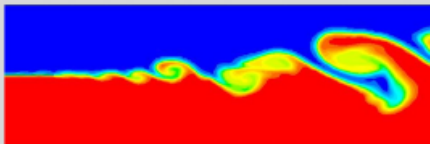
NS egyenletek – szimulációs közelítések - elmélet

Overview of Computational Approaches

- Different approaches to make turbulence computationally tractable

DNS

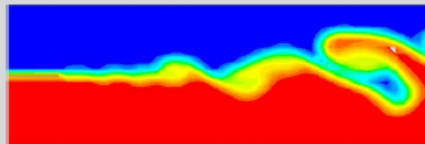
(Direct Numerical Simulation)



- Numerically solving the full unsteady Navier-Stokes equations
- Resolves the whole spectrum of scales
- No modeling is required
- **But the cost is too prohibitive!**
Not practical for industrial flows!

LES

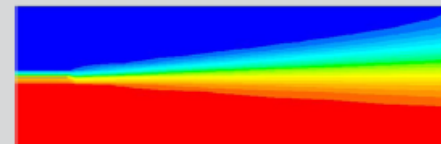
(Large Eddy Simulation)



- 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.)



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés - elmélet

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.)

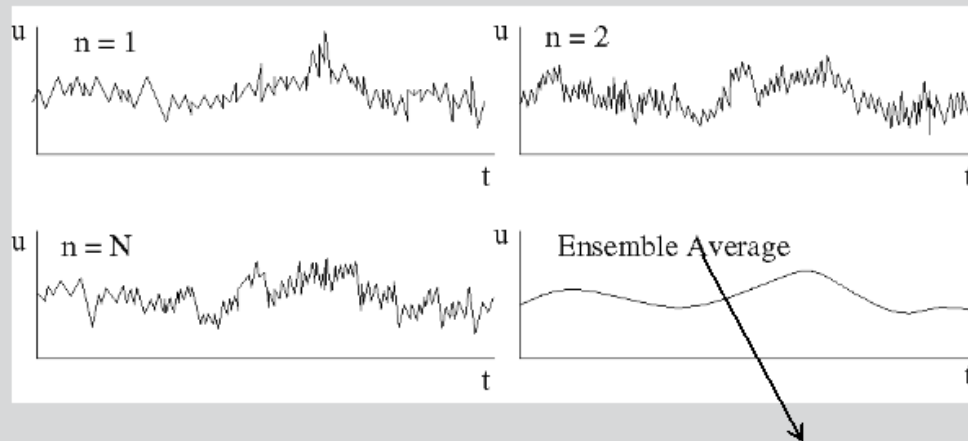


Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés - elmélet

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**



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

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



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés - elmélet

RANS Modeling : Averaging

- Ensemble (time) averaging may be used to extract the mean flow properties from the instantaneous ones

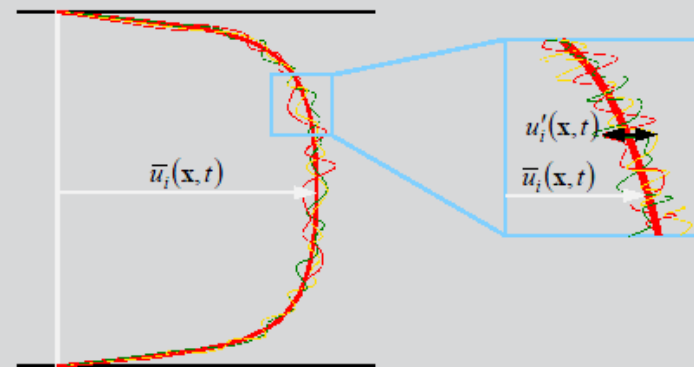
– The instantaneous velocity, u_i , is split into average and fluctuating components

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

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.)

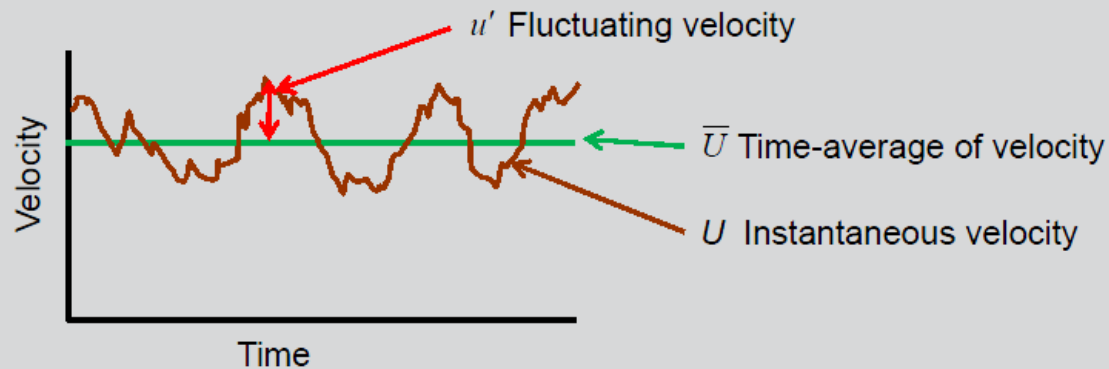


Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés - elmélet

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: $U = \bar{U} + u'$
- 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: $k = 0.5 * (\overline{u'^2} + \overline{v'^2} + \overline{w'^2})$

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



Áramlásmodellezés – Kontinuum-mechanika alapján

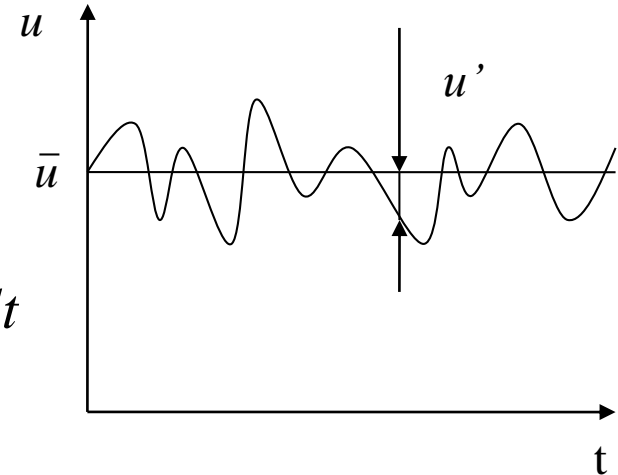
RANS egyenletek – Reynolds és Favre átlagolás - DASFLOW

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

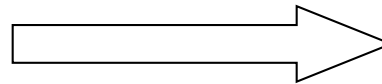
$$w = \bar{w} + w' \quad p = \bar{p} + p'$$

$$\bar{u} = \frac{1}{\Delta t} \int_{t_0}^{t_0+\Delta t} u dt$$

$$\tilde{u} = \frac{1}{\bar{\rho}} \frac{1}{\Delta t} \int_{t_0}^{t_0+\Delta t} (\rho u) dt$$



Reynolds Átlagolás



Favre Átlagolás

Nagy sebességű,
összenyomható áramlás esetén.

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

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



L : laminar transport of the heat

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

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

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

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

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

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

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

$$\begin{aligned} & \frac{\partial}{\partial t} \left[\bar{\rho} \left(\tilde{e} + \frac{1}{2} \sum_{i=1}^3 \tilde{u}_i \tilde{u}_i \right) + \frac{1}{2} \sum_{i=1}^3 \overline{\rho u_i'' u_i''} \right] + \sum_{j=1}^3 \frac{\partial}{\partial x_j} \left[\bar{\rho} \tilde{u}_j \left(\tilde{h} + \frac{1}{2} \sum_{i=1}^3 \tilde{u}_i \tilde{u}_i \right) + \tilde{u}_j \frac{1}{2} \sum_{i=1}^3 \overline{\rho u_i'' u_i''} \right] = \\ & = \sum_{j=1}^3 \frac{\partial}{\partial x_j} \left[-\bar{q}_j - \overline{\rho u_j'' h''} + \sum_{i=1}^3 \overline{\tau_{ji} u_i''} - \frac{1}{2} \sum_{i=1}^3 \overline{\rho u_j'' u_i'' u_i''} \right] + \sum_{j=1}^3 \frac{\partial}{\partial x_j} \left[\sum_{i=1}^3 \left[\tilde{u}_i (\bar{\tau}_{ji} - \overline{\rho u_i'' u_j''}) \right] \right] \end{aligned}$$



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Reynolds és Favre átlagolás - DASFLOW

$$\frac{1}{2} \sum_{i=1}^3 \overline{\rho u_i'' u_i''} = \bar{\rho} k \quad (1.75)$$

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

$$q_{Tj} = \overline{\rho u_j'' h''} \quad (1.76)$$

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:

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

$$\bar{p} = \bar{\rho} R \tilde{T}$$



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Reynolds és Favre átlagolás - 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:

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



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Reynolds és Favre átlagolás - 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:

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

where (for 3 dimensions)

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



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Reynolds és Favre átlagolás - DASFLOW

$$\tau_{xx}^F = 2(\mu + \mu_t) \frac{\partial \tilde{u}}{\partial x} - \frac{2}{3}(\mu + \mu_t) \nabla^T \tilde{V} - \frac{2}{3} \bar{\rho} k = \frac{2}{3} \mu_{eff} \left(2 \frac{\partial \tilde{u}}{\partial x} - \frac{\partial \tilde{v}}{\partial y} - \frac{\partial \tilde{w}}{\partial z} \right) - \frac{2}{3} \bar{\rho} k \quad (1.81)$$

$$\tau_{yy}^F = 2(\mu + \mu_t) \frac{\partial \tilde{v}}{\partial y} - \frac{2}{3}(\mu + \mu_t) \nabla^T \tilde{V} - \frac{2}{3} \bar{\rho} k = \frac{2}{3} \mu_{eff} \left(-\frac{\partial \tilde{u}}{\partial x} + 2 \frac{\partial \tilde{v}}{\partial y} - \frac{\partial \tilde{w}}{\partial z} \right) - \frac{2}{3} \bar{\rho} k \quad (1.82)$$

$$\tau_{zz}^F = 2(\mu + \mu_t) \frac{\partial \tilde{w}}{\partial z} - \frac{2}{3}(\mu + \mu_t) \nabla^T \tilde{V} - \frac{2}{3} \bar{\rho} k = \frac{2}{3} \mu_{eff} \left(-\frac{\partial \tilde{u}}{\partial x} - \frac{\partial \tilde{v}}{\partial y} + 2 \frac{\partial \tilde{w}}{\partial z} \right) - \frac{2}{3} \bar{\rho} k \quad (1.83)$$

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

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

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

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:



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Reynolds és Favre átlagolás - DASFLOW

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

Heat Flux Vector Terms

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

$$q_{Tj} = \overline{\rho u_j'' h''} = -\frac{\mu_t c_p}{\text{Pr}_t} \frac{\partial \tilde{T}}{\partial x_j} \quad (1.88)$$

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

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

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

$$\bar{q} = \bar{q}_L + \bar{q}_T = -k \nabla \tilde{T} \quad (1.90)$$



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Reynolds és Favre átlagolás - DASFLOW

Molecular Diffusion and Turbulent Transport Terms

These terms are often handled together as

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

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.

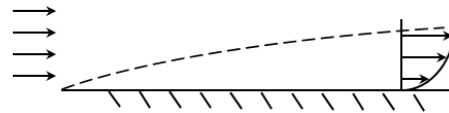


Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés - elmélet

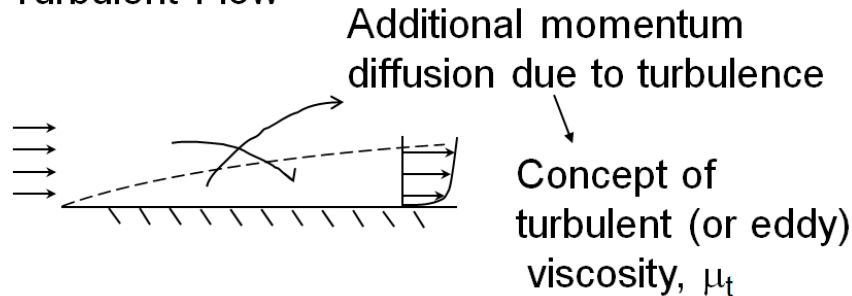
Introduction to Turbulence Modelling

Laminar Flow



Momentum
diffusion
by viscosity

Turbulent Flow



- 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.)



Áramlásmodellezés – Kontinuum-mechanika alapján RANS egyenletek – turbulencia modellezés - elmélet

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)

$$\frac{\partial}{\partial t} (\rho \overline{u'_i u'_j}) + \frac{\partial}{\partial x_k} (\rho \bar{u}_k \overline{u'_i u'_j}) = P_{ij} + F_{ij} + D_{ij}^T + \Phi_{ij} - \varepsilon_{ij}$$

- 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

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

- 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.)



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Reynolds és Favre átlagolás - DASFLOW

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

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

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

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

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

$$H(x) = \begin{cases} 0 & \text{if } x \leq 0 \\ 1 & \text{if } x > 0 \end{cases} \quad S_{ij} = \frac{1}{2} \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right) \quad F(M_t) = [M_t^2 - M_{t0}^2] H(M_t - M_{t0})$$

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



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Reynolds és Favre átlagolás – Konzervatív összevont forma - DASFLOW

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

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



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Reynolds és Favre átlagolás – Konzervatív összevont forma - DASFLOW

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

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



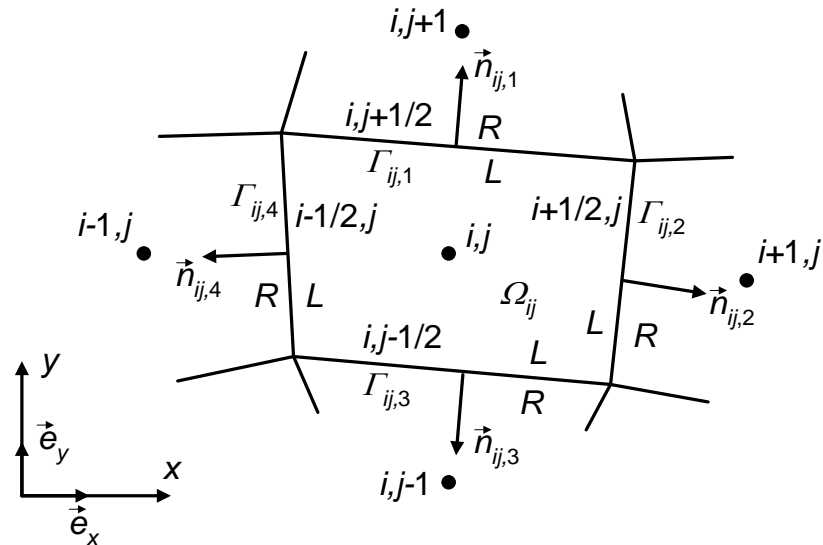
Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Diszkretizáció - DASFLOW

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

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

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



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

$$\left. \begin{aligned} U^0 &= U^n \\ U^k &= U^0 + \alpha_k \Delta t \mathfrak{R}(U^{k-1}) \quad k = 1, m \\ U^{n+1} &= U^m \end{aligned} \right\} \begin{aligned} &\bullet \text{ Runge-Kutta módszer} \\ &m=4 \end{aligned}$$



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés

RANS Modeling : Averaging

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

$$\rho \left(\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_k \frac{\partial \bar{u}_i}{\partial x_k} \right) = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\mu \frac{\partial \bar{u}_i}{\partial x_j} \right) + \frac{\partial R_{ij}}{\partial x_j}$$

$R_{ij} = -\overline{\rho u'_i u'_j}$
 (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

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

→ 6 unknowns ...

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



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés

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)

$$\frac{\partial}{\partial t} (\rho \overline{u'_i u'_j}) + \frac{\partial}{\partial x_k} (\rho \bar{u}_k \overline{u'_i u'_j}) = P_{ij} + F_{ij} + D_{ij}^T + \Phi_{ij} - \varepsilon_{ij}$$

- 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

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

- 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.)

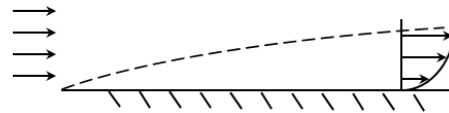


Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés

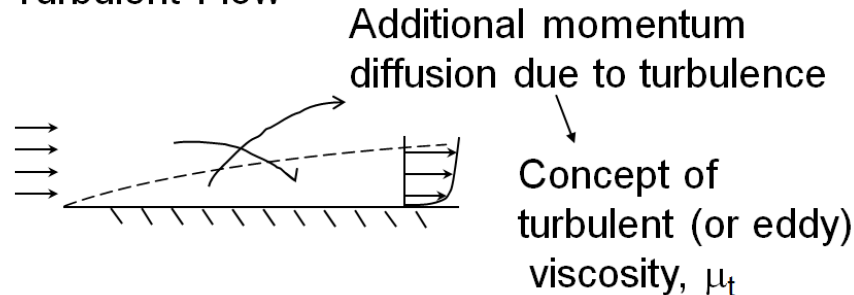
Introduction to Turbulence Modelling

Laminar Flow



Momentum
diffusion
by viscosity

Turbulent Flow



- 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.)



Áramlásmodellezés – Kontinuum-mechanika alapján RANS egyenletek – turbulencia modellezés - modellek

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 $k-\varepsilon$ model.
- 3) RNG $k-\varepsilon$ model.
- 4) Standard $k-\omega$ model.
- 5) Baseline (BSL) zonal $k-\omega$ based model.
- 6) SST zonal $k-\omega$ based model.
- 7) $(k-\varepsilon)_{1E}$ 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.)



Áramlásmodellezés – Kontinuum-mechanika alapján RANS egyenletek – turbulencia modellezés - modellek

k-omega Model

- A pragmatic compromise for RANS Turbulence modeling:
 - $k-\omega$ equations based models
- This models have gained popularity mainly because:
- $k-\omega$ models perform much better than $k-\varepsilon$ models for boundary layer flows
 - For separation, transition, low Re effects, impingement, the $k-\omega$ models is more accurate than the $k-\varepsilon$ 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

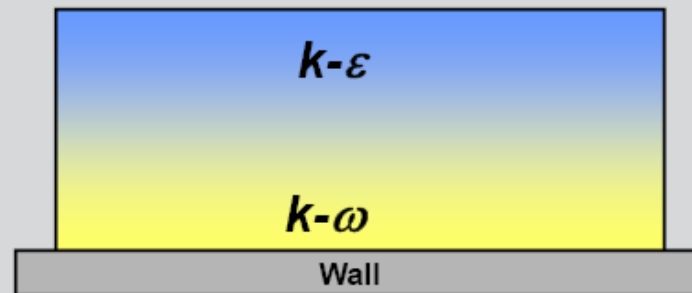
Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)



Áramlásmodellezés – Kontinuum-mechanika alapján RANS egyenletek – turbulencia modellezés - modellek

SST Model

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



- The $k-\varepsilon$ and $k-\omega$ models are blended such that the SST model functions like the $k-\omega$ close to the wall and the $k-\varepsilon$ model in the freestream

SST is a good compromise between $k-\varepsilon$ and $k-\omega$ models

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



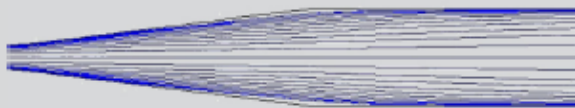
Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés - modellek

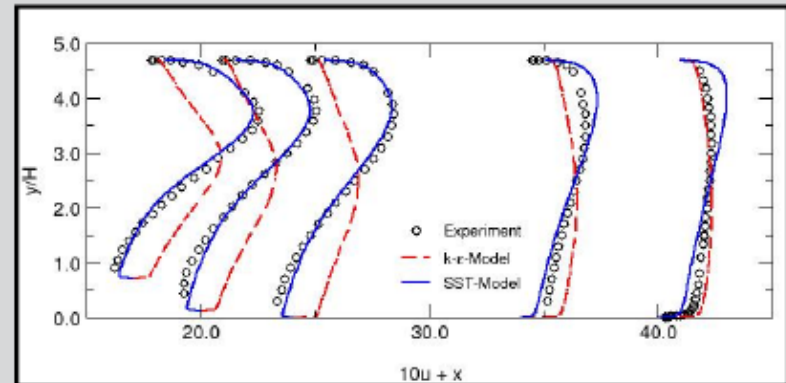
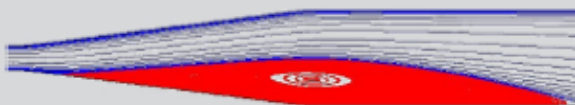
SST Model

- 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 $k-\varepsilon$ 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.)

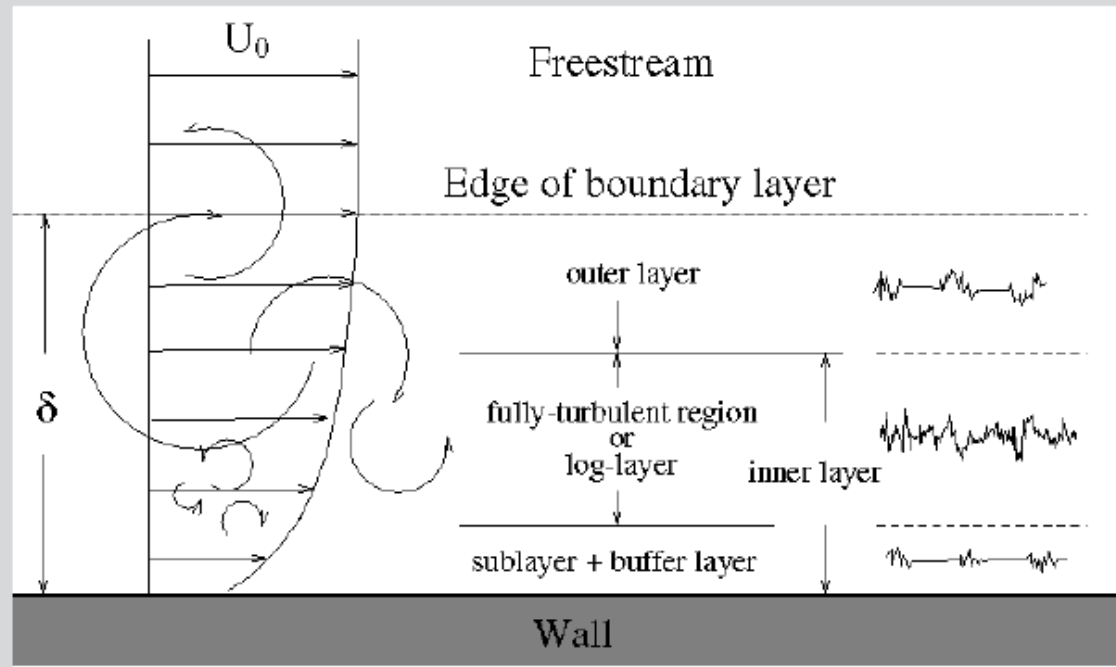


Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés – fal modellek

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.)



Áramlásmodellezés – Kontinuum-mechanika alapján

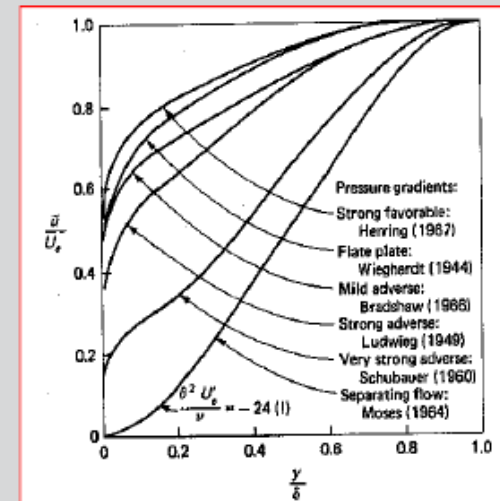
RANS egyenletek – turbulencia modellezés - fal modellek

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.)

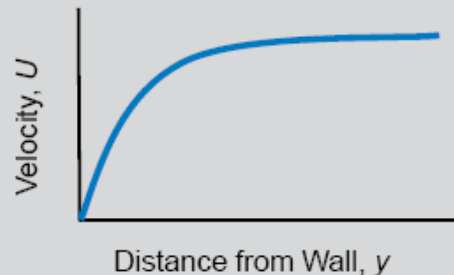


Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés - fal modellek

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/U_τ ($u_\tau = \sqrt{\frac{\tau_{\text{wall}}}{\rho}}$)
 - 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:

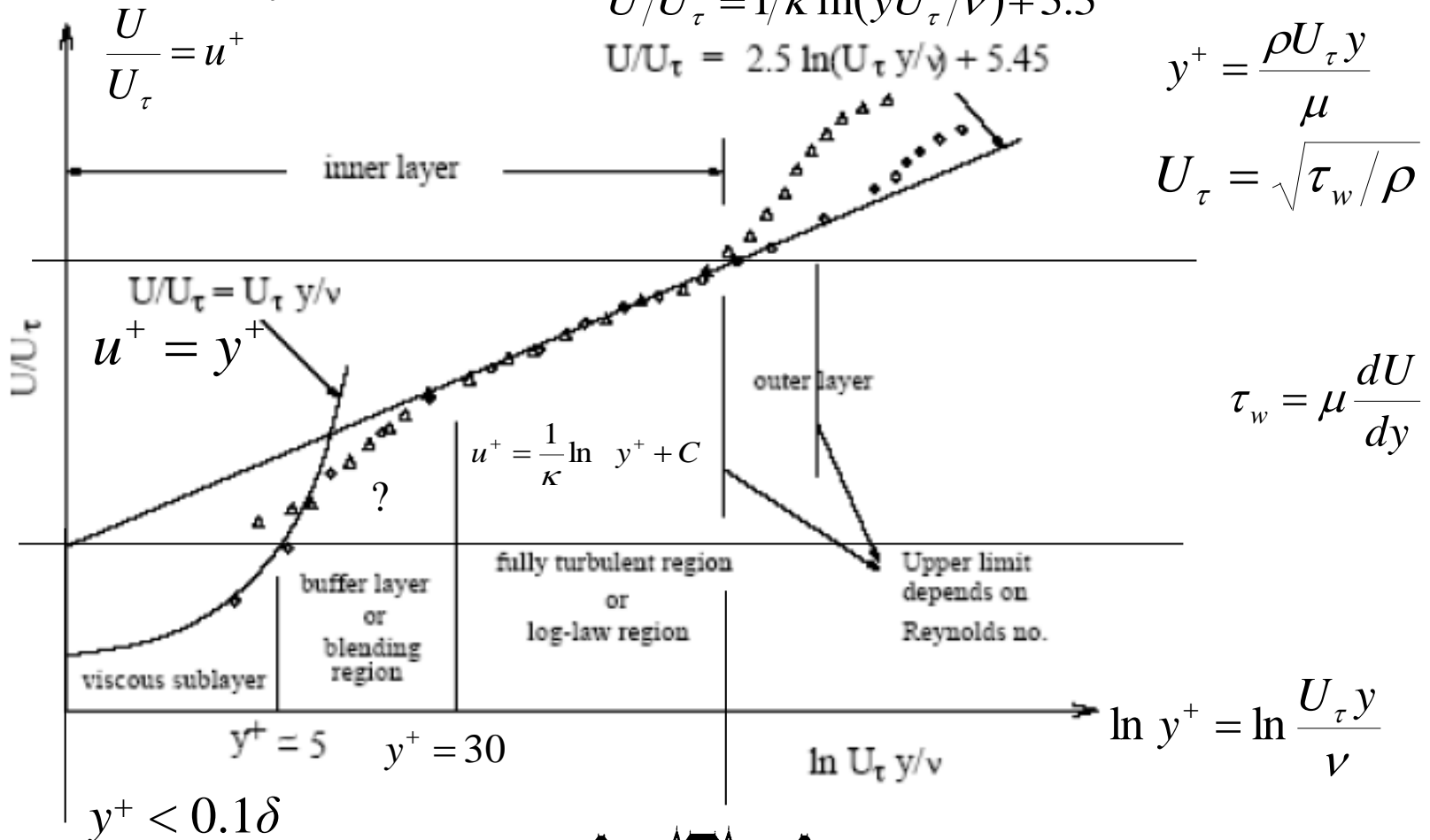
Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés - fal modellek

Law of the wall by Prandtl



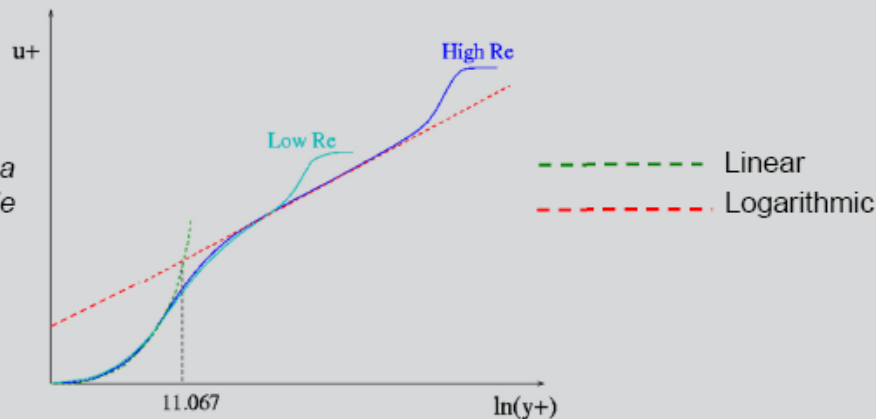
Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés - fal modellek

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.)

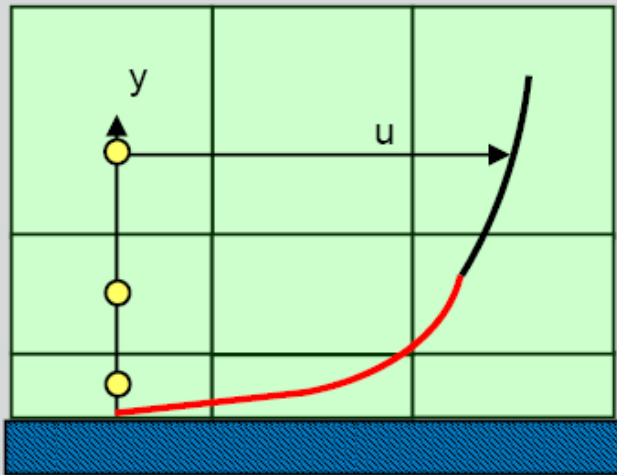


Áramlásmodellezés – Kontinuum-mechanika alapján

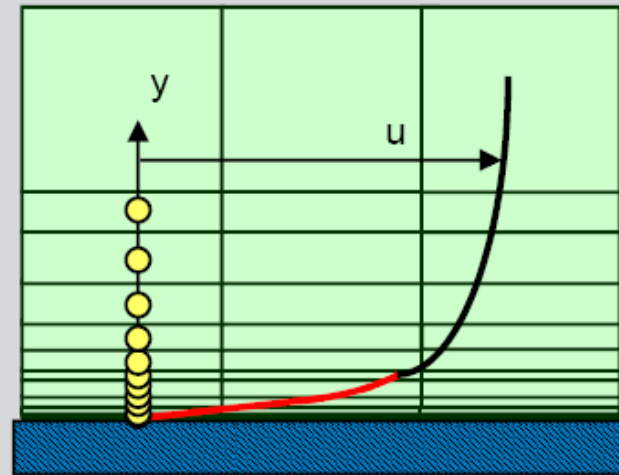
RANS egyenletek – turbulencia modellezés - fal modellek

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.)



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés - fal modellek

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:

$$y_p^+ = \frac{y_p u_\tau}{\nu} \Rightarrow y_p = \frac{y_p^+ \nu}{u_\tau} \quad u_\tau = \sqrt{\frac{\tau_w}{\rho}} = U_e \sqrt{\frac{C_f}{2}}$$
$$\text{Flat Plate: } \frac{\bar{C}_f}{2} \approx \frac{0.037}{\text{Re}_L^{1/5}} \quad \text{Duct: } \frac{\bar{C}_f}{2} \approx \frac{0.039}{\text{Re}_{D_h}^{1/4}}$$

$y^+ \sim 10-15$ 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.)

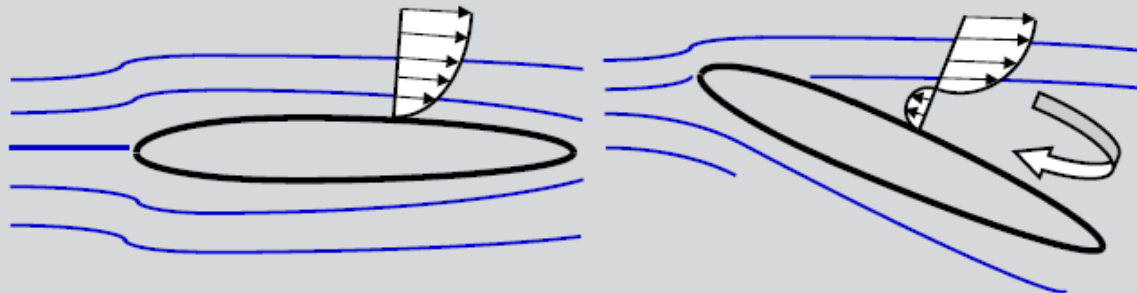


Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés - fal modellek

Limitations of Wall Function

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



Wall functions applicable

Wall functions not applicable

- 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.)



Áramlásmodellezés – Kontinuum-mechanika alapján RANS egyenletek – turbulencia modellezés - fal modellek

y^+ for the SST and k - ω 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

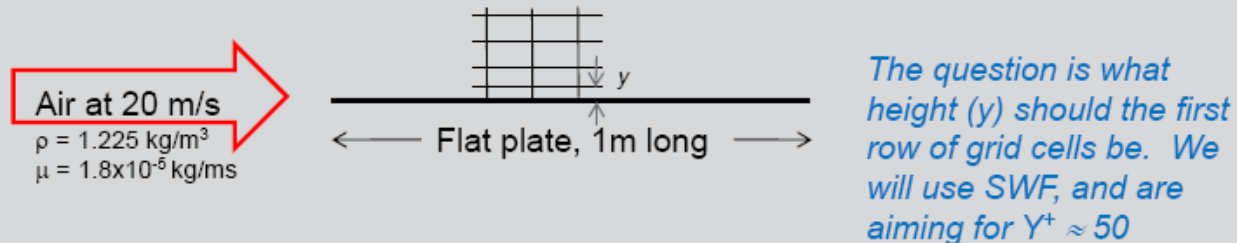
Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)



Áramlásmodellezés – Kontinuum-mechanika alapján RANS egyenletek – turbulencia modellezés - fal modellek

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:



- For a flat plate, Reynolds number ($Re_L = \frac{\rho VL}{\mu}$) gives $Re_L = 1.4 \times 10^6$
- (Recall from earlier slide, flow over a surface is turbulent when $Re_L > 5 \times 10^5$)

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



Áramlásmodellezés – Kontinuum-mechanika alapján RANS egyenletek – turbulencia modellezés - fal modellek

Example in predicting near-wall cell size [2]

- A literature search suggests a formula for the skin friction on a plate¹ thus

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

- Use this value to predict the wall shear stress τ_w

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

- From τ_w compute the velocity U_τ

$$U_\tau = \sqrt{\frac{\tau_w}{\rho}} \quad U_\tau = 0.82 \text{ m/s}$$

- Rearranging the equation shown previously for y^+ gives a formula for the first cell height, y , in terms of U_τ

$$y = 9 \times 10^{-4} \text{ m}$$

- 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.)



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés - fal modellek

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 $y^+ = 1$:

$$V = 20 \text{ m/s}, \quad \rho = 1.225 \text{ kg/m}^3, \quad \mu = 1.8 \times 10^{-5} \text{ kg/ms}$$

$$Re_l = \frac{\rho VL}{\mu} \rightarrow Re_l = 1.4 \times 10^6$$

$$\rightarrow C_f = 0.0034$$

$$\rightarrow \tau_w = 0.83 \text{ kg/ms}^2$$

$$C_f = 0.058 Re_l^{-0.2} \rightarrow U_\tau = 0.82 \text{ m/s}$$

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

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

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

aiming for y^+ of 1:
our first cell height y
should be $\approx 0.02 \text{ mm}$

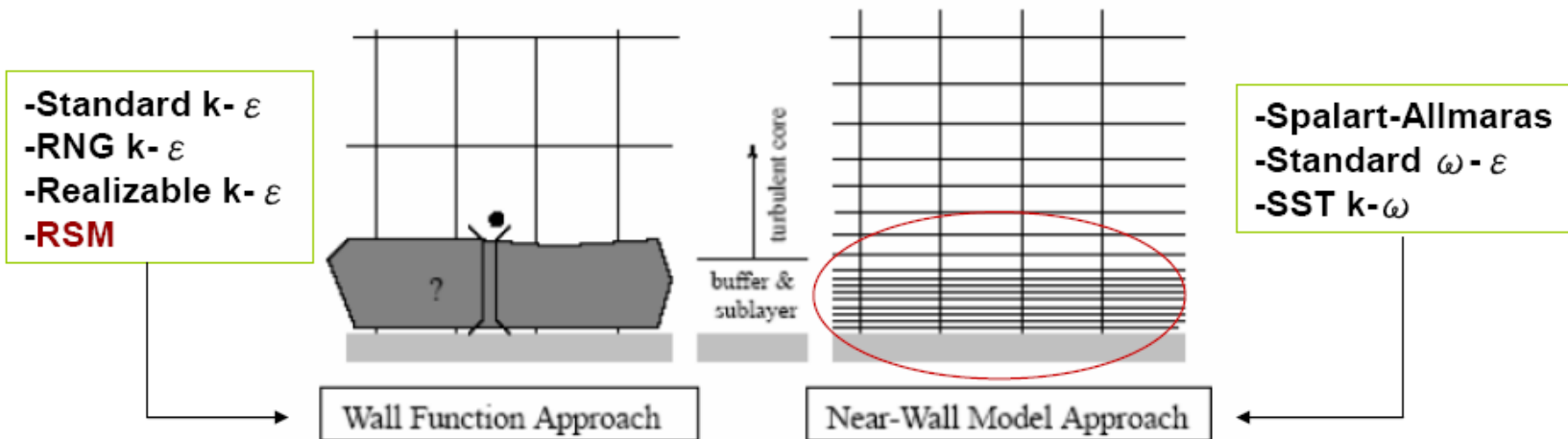
Forrás: Introduction to ANSYS CFX, Lecture 07 - Turbulence, CFX-Intro_14.0_L07_Turbulence .pdf (2013.09.01.)



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés - fal modellek

Wall Boundary Conditions



- 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



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés - fal modellek

Placement of The First Grid Point

■ Wall function $\sim y^+ \doteq 30-300$

■ Near-wall function $\sim y^+ \doteq 1$

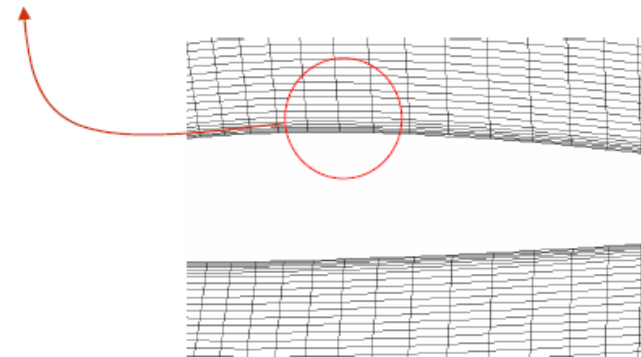
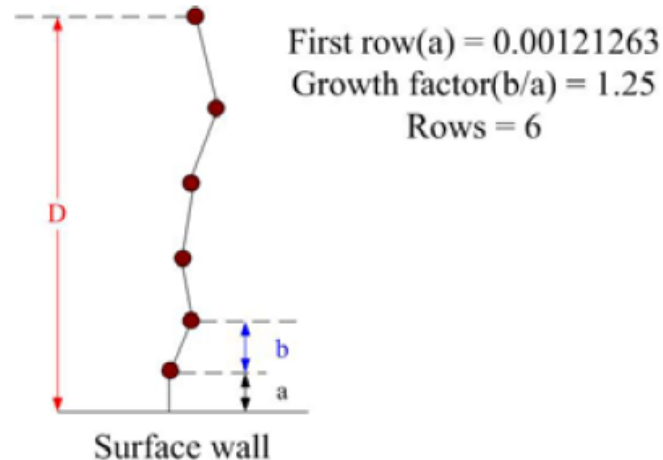
Non-dimensional distance from a wall in a turbulent boundary layer is given by y^+

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

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



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés - fal modellek

ANSYS CFX

Formulation

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

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

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

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

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

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

The definition of Δy^+ for this estimate is:

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

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

Using the definition

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

Forrás: Introduction to ANSYS CFX



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés - fal modellek

ANSYS CFX

u_τ can be eliminated in Equation 16 to yield:

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

c_f can be eliminated using Equation 15 to yield:

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

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:

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

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



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés - fal modellek

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:

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

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:

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

to be:

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

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:

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

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:

$$n(15) - n(1) \leq \delta \quad \text{Equation 25.}$$

Forrás: Introduction to ANSYS CFX



Áramlásmodellezés – Kontinuum-mechanika alapján RANS egyenletek – turbulencia modellezés - fal modellek

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 Drive Canonsburg, PA 15317, ansysinfo@ansys.com, <http://www.ansys.com>, USA, 2012



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés – belépő perem

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

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

$$\text{Turbulent viscosity ratio : } \mu_t / \mu$$

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



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés – belépő perem

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.)



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés – belépő perem

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.)



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés – belépő perem

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.)



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés – összefoglalás

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.)



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – turbulencia modellezés – összefoglalás

RANS Turbulence Model usage

Model	Behavior and Usage
Standard $k-\epsilon$	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 $k-\epsilon$	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 $k-\omega$	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 $k-\omega$	Offers similar benefits as standard $k-\omega$. 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.)



Áramlásmodellezés – Kontinuum-mechanika alapján RANS egyenletek – turbulencia modellezés – összefoglalás

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.)



Áramlások numerikus modellezése



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Diszkretizáció - 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.

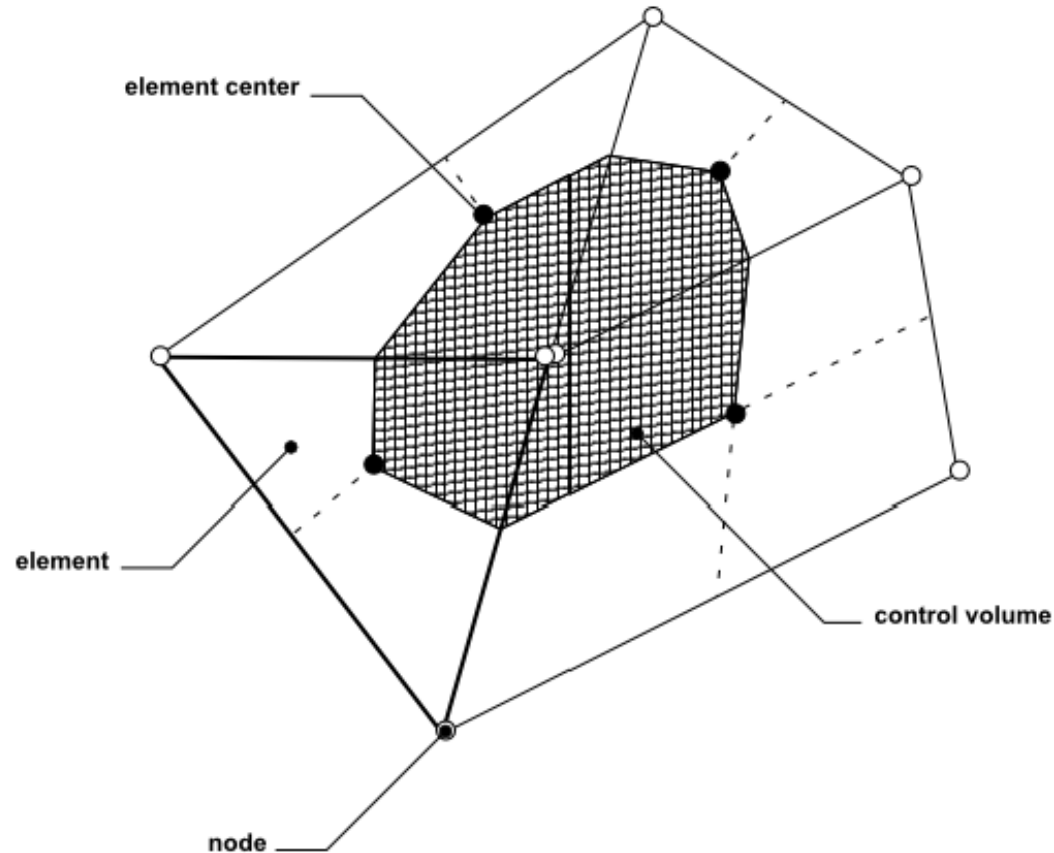
FINITE ELEMENT,

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



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

Control Volume Definition



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



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Diszkretizáció - CFX

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial \mathbf{x}_j} (\rho U_j) = 0$$

$$\frac{\partial}{\partial t} (\rho U_i) + \frac{\partial}{\partial \mathbf{x}_j} (\rho U_j U_i) = - \frac{\partial P}{\partial \mathbf{x}_i} + \frac{\partial}{\partial \mathbf{x}_j} \left(\mu_{eff} \left(\frac{\partial U_i}{\partial \mathbf{x}_j} + \frac{\partial U_j}{\partial \mathbf{x}_i} \right) \right)$$

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

$$\frac{d}{dt} \int_V \rho dV + \int_s \rho U_j dn_j = 0$$

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

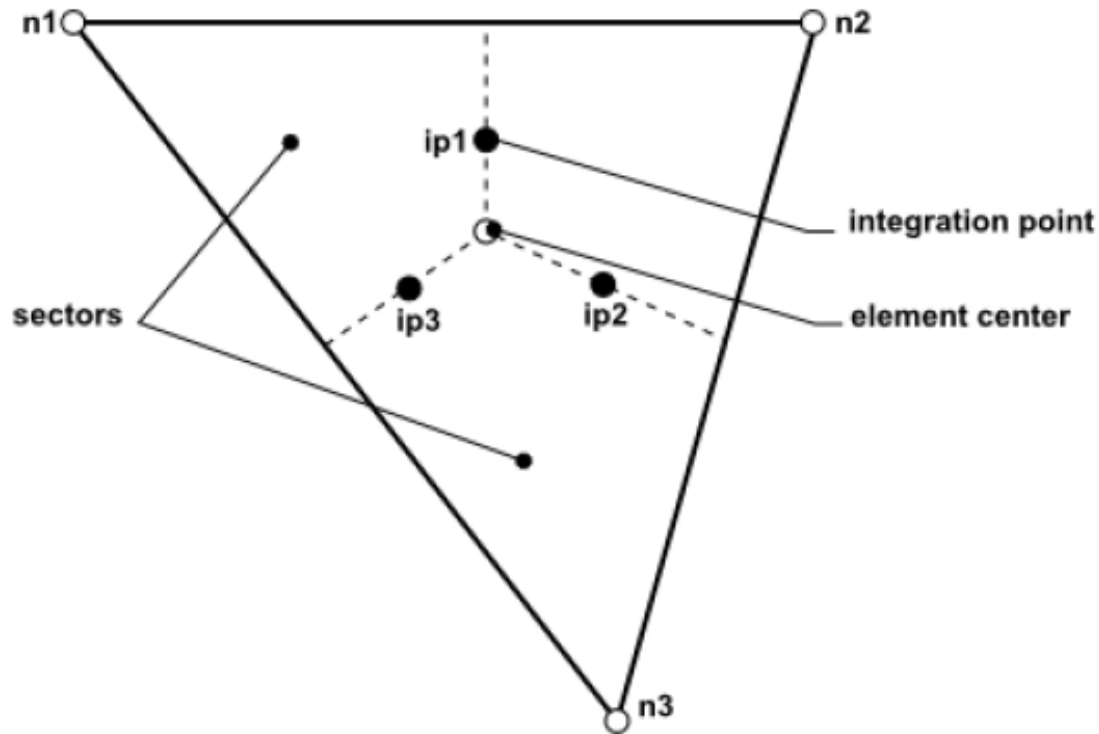
$$\frac{d}{dt} \int_V \rho \varphi dV + \int_s \rho U_j \varphi dn_j = \int_s \Gamma_{eff} \left(\frac{\partial \varphi}{\partial \mathbf{x}_j} \right) dn_j + \int_V S_\varphi dV$$

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



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Diszkretizáció - CFX



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



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Diszkretizáció - CFX

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

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

$$\sum_{ip} \left(\mu_{eff} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \Delta n_j \right)_{ip} + \overline{S_{U_i}} V$$

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

- a következő iterációs időlépés paramétereit. Az algebrai egyenletrendszer megoldása: implicit első v. másodrendű Backward Euler módszerrel

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



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Diszkretizáció - 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



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

- Kollokált struktúrájú adattárolás (a változók ugyan azon a helyen tárolódnak) → **Majumdar** által kidolgozott módszerhez hasonló eljárás a nyomás és a sebességtér szétválasztódása ellen.
- Csatolt megoldó (tömeg- és impulzus-megmaradás egybe számolva)
- Implicit első v. másodrendű Backward Euler módszer
- Lineáris egyenletrendszer megoldó: „Multigrid accelerated Incomplete Lower Upper (ILU) factorization technique”.

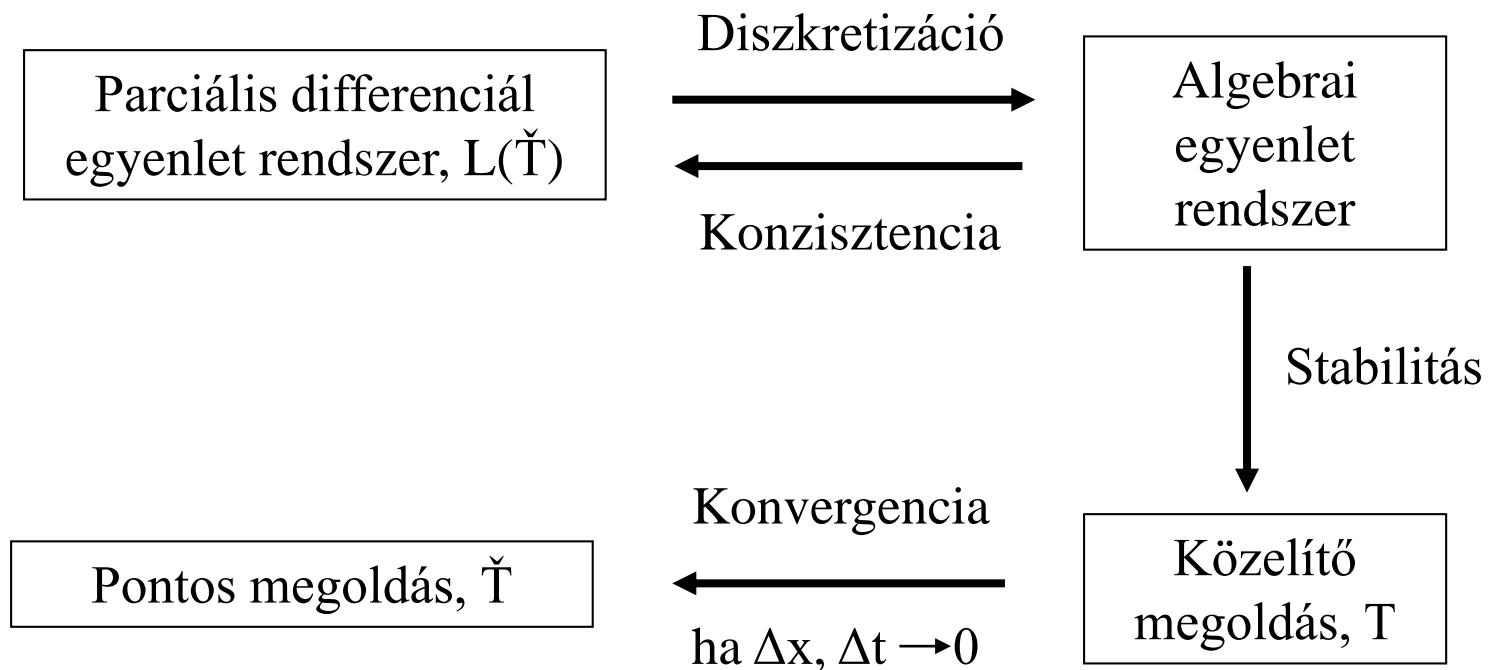
Forrás: ANSYS CFX-Solver Theory Guide, ANSYS Inc., Southpointe, 275 Technology Drive, Canonsburg, PA 15317, October 2012



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Diszkretizáció

Fontosabb matematikai tulajdonságok



- Kezdeti- és peremfeltételek



CFD feladat végrehajtásának főbb lépései és fontosabb szabályai



CFD feladat főbb lépései

Pre-processzálás

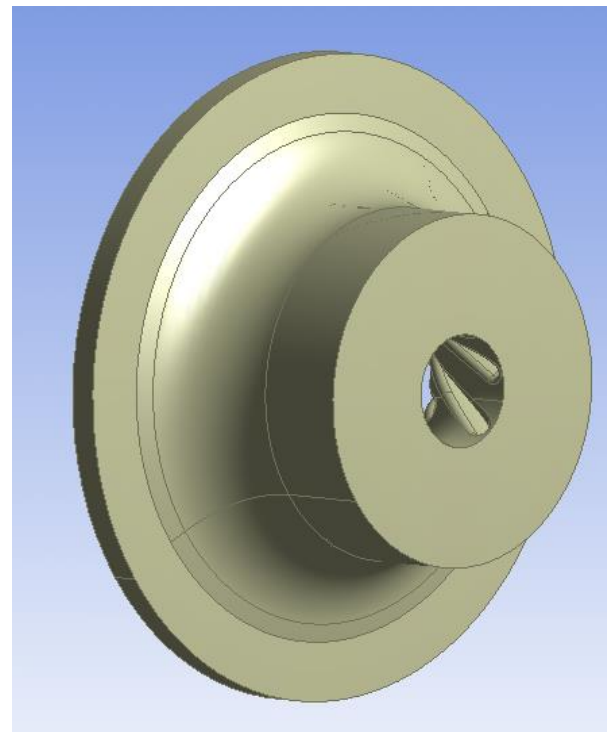
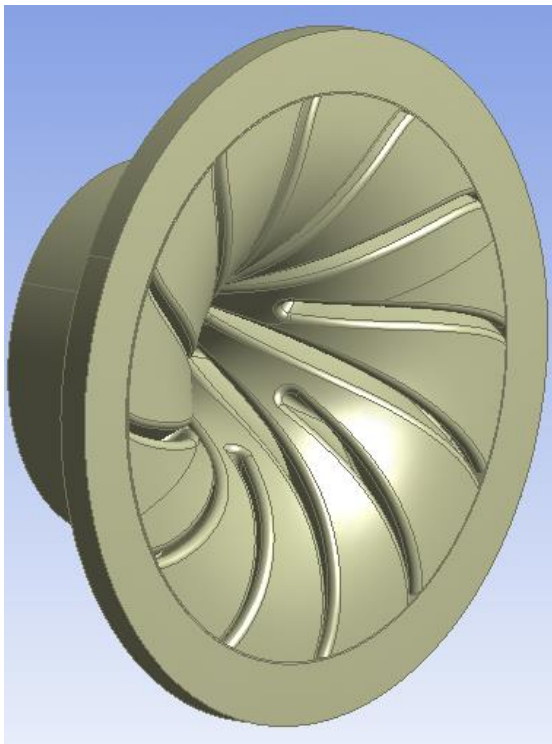
- Célkitűzés, a megoldandó feladat áttekintése és lefordítása a modellezéssel kezelhető formára, idő és ütemterv (esetleg árajánlat) elkészítése,
- Geometriai modellalkotás (áramlási tér),
- A numerikus háló elkészítése,
- Anyagtulajdonságok definiálása,
- Kapcsolódó fizikai modellek és paramétereik beállítása,
- Peremfeltételek megadása és a geometriához rendelése,
- Kezdeti feltételek megadása,
- A megoldó tulajdonságainak beállítása,
- A számítás elindítása és a konvergencia értékelése,
- Az eredmények megtekintése, elemzése és értékelése.
- Validáció (ha van rá lehetőség) és verifikáció.
- Paraméter érzékenységi vizsgálatok elvégzése. Szükség szerint javaslattétel a javító intézkedések meghozására és ellenőrzése.
- Dokumentáció elkészítése.

Post-processzálás

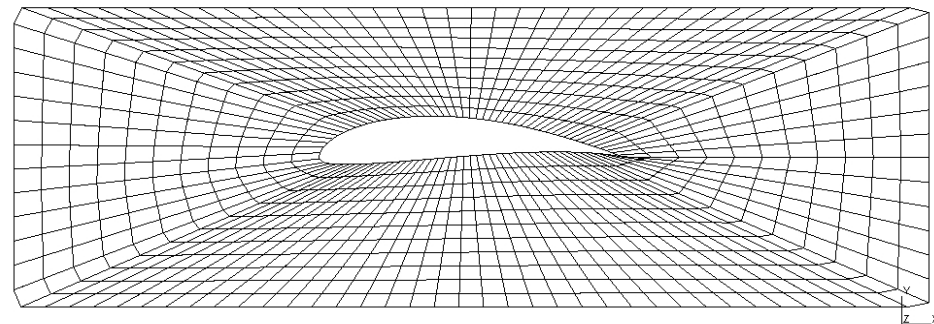
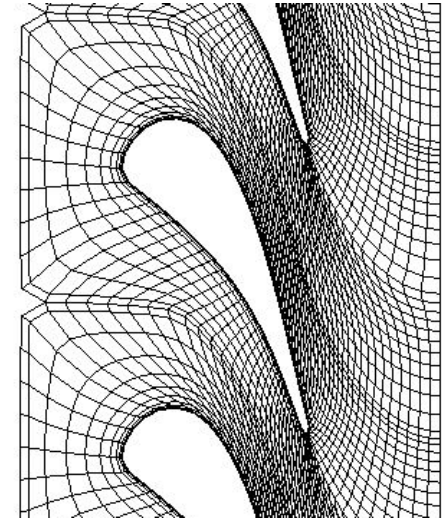
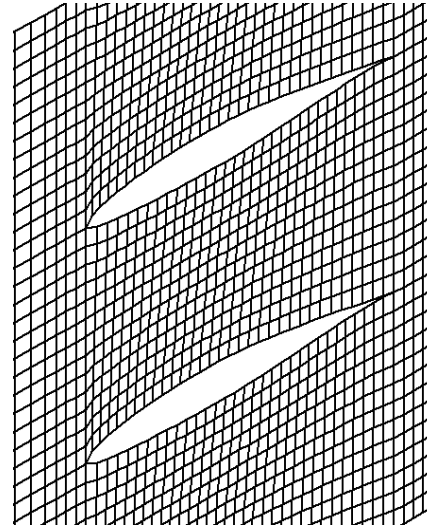
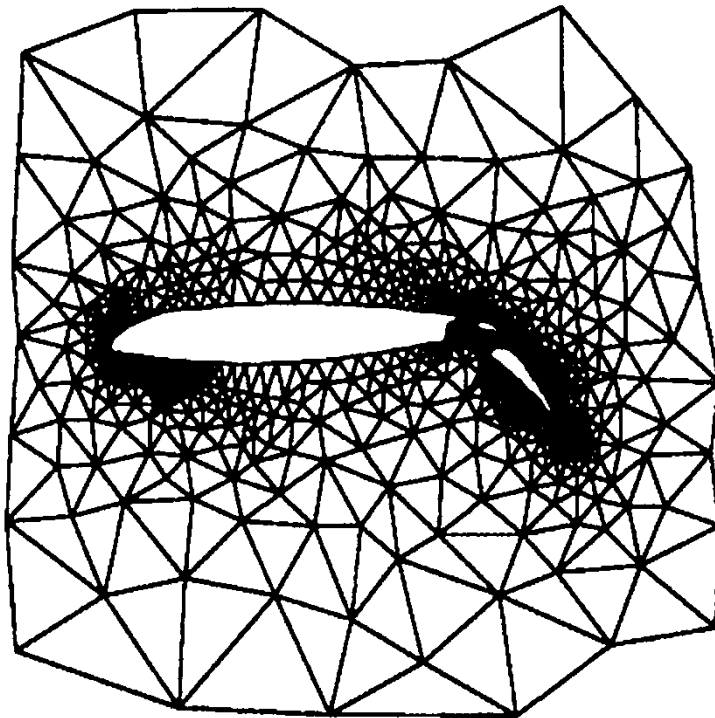


CFD feladat főbb lépései

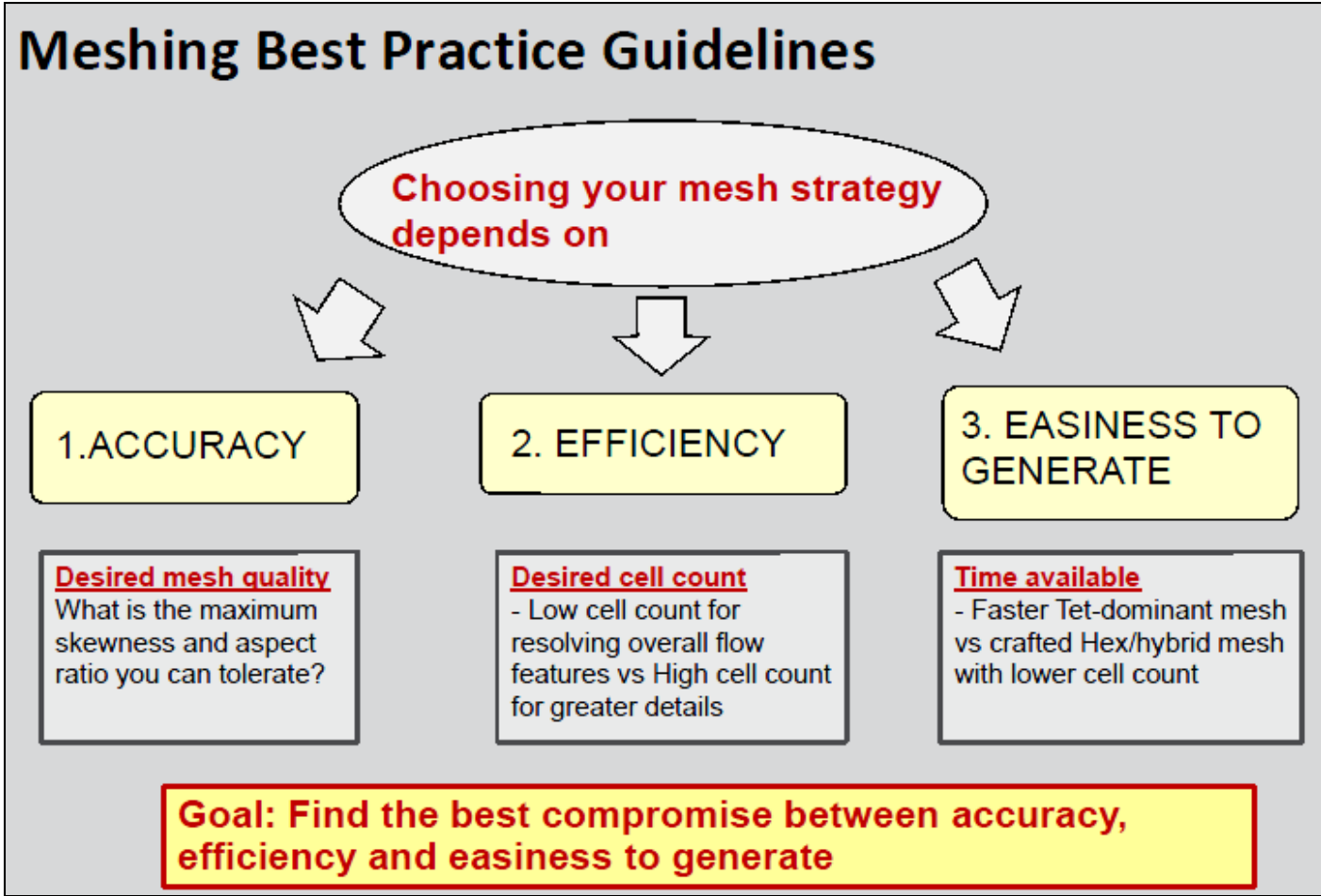
Geometria – áramlástanai tér



CFD feladat főbb lépései - Geometriai diszkretizáció



CFD feladat főbb lépései - Geometriai diszkretizáció



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



CFD feladat főbb lépései - Geometriai diszkretizáció

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:
 - $\leq 1.2 \dots 1.3$
 - $y^+ \approx 1$ for heat transfer and transition modeling

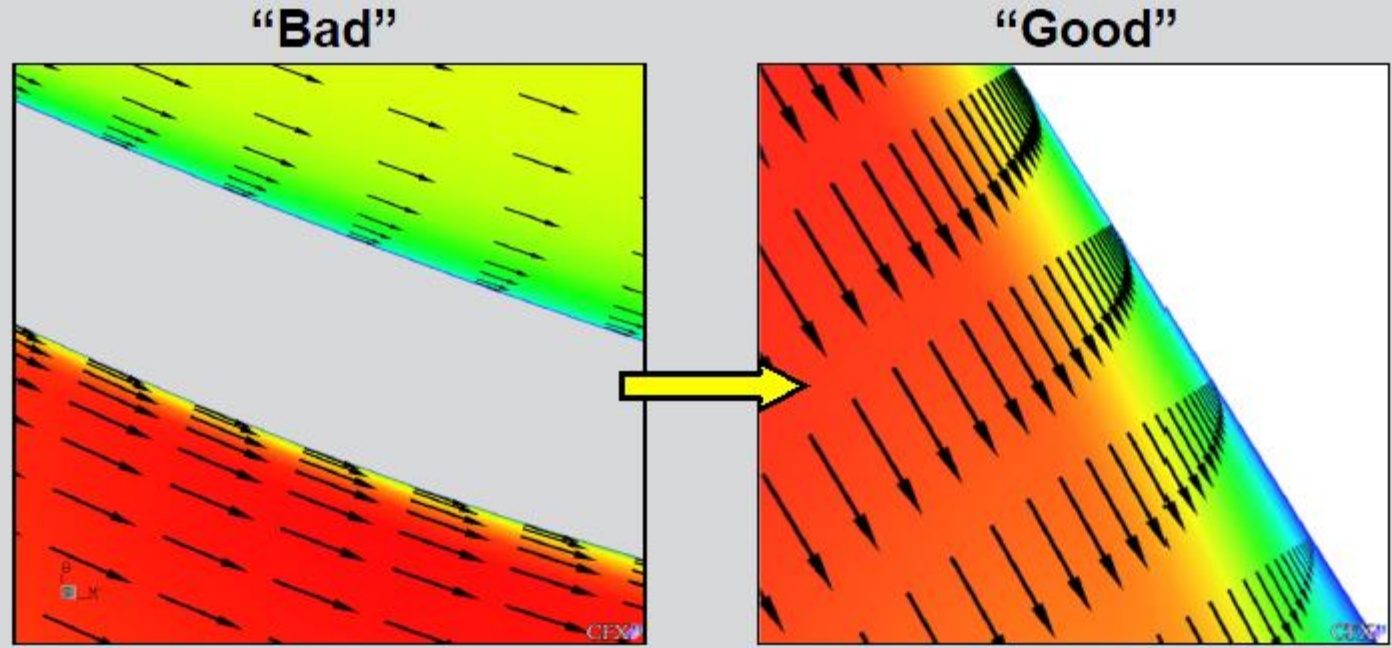
Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)



CFD feladat főbb lépései - Geometriai diszkretizáció

Meshing: Capture Flow Physics

- Example: Velocity profiles at airfoil



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



CFD feladat főbb lépései - Geometriai diszkretizáció

Mesh Quality

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

	Good	Not Good
Cell not too distorted		
Cell not too stretched		
Smooth Cells transition		

Részletesen lásd: Mesh Metrix, Mesh_metrix_in_ANSYS_WB_13_v11.ppt

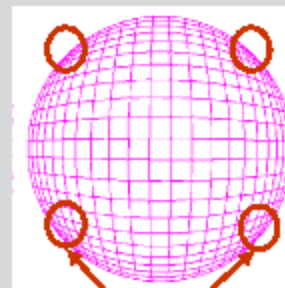
Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)



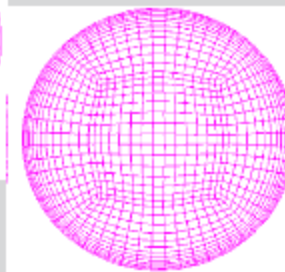
CFD feladat főbb lépései - Geometriai diszkretizáció

Mesh Quality

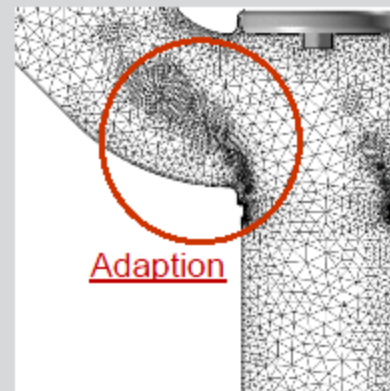
- **Grid generation:**
 - Scalable grids
 - **Skewness < 0.9 (accuracy, convergence)**
 - Aspect ratios < 100
 - Expansion ratios < 1.5 ...2
 - Capture physics based on experience (shear layers, shocks)
 - Angle between grid face & flow vector
- **Grid refinement:**
 - Manual, based on error estimate
 - Automatic adaptive based on 'error sensor'



Bad cells



No Bad cells



Adaption

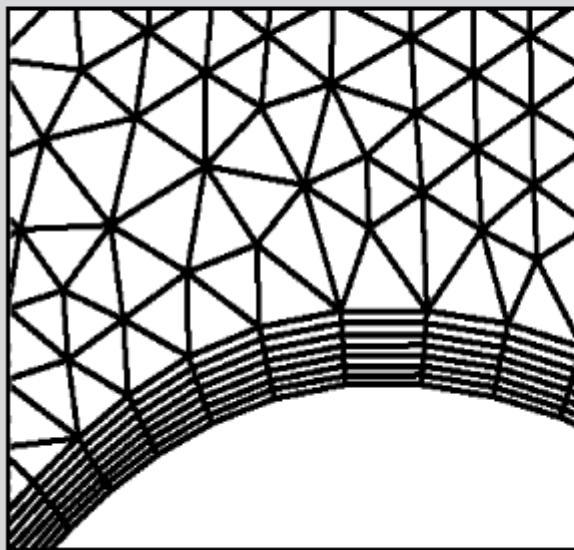
Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)



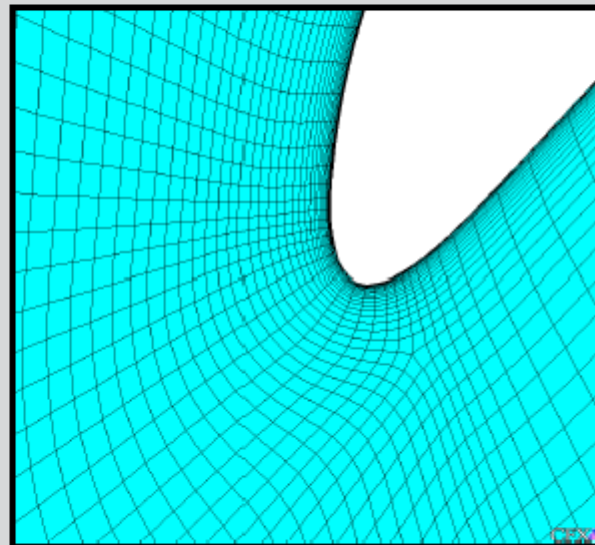
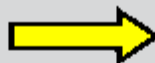
CFD feladat főbb lépései - Geometriai diszkretizáció

Mesh Quality

- Avoid sudden change in mesh density



Not good



Good

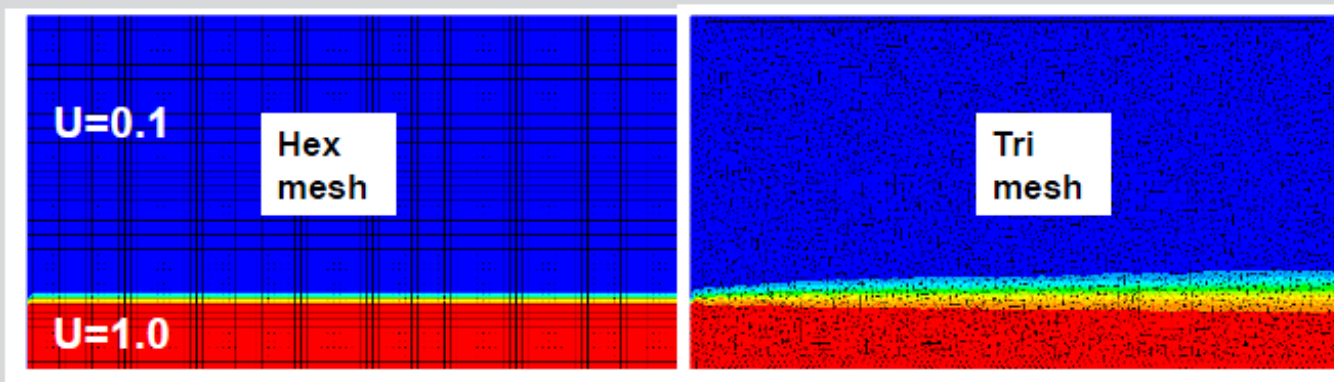
Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)



CFD feladat főbb lépései - Geometriai diszkretizáció

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

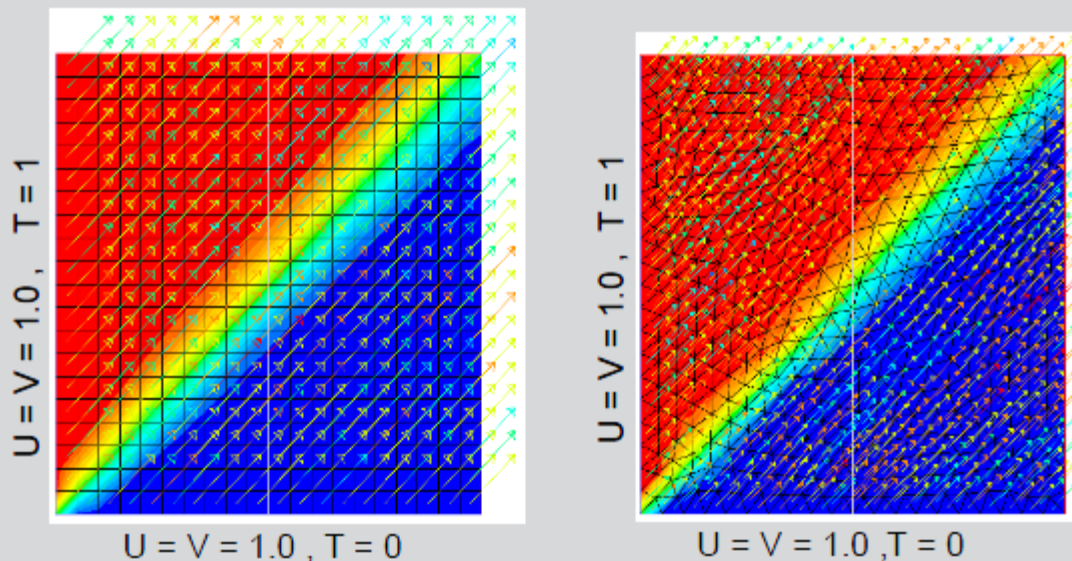
Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)



CFD feladat főbb lépései - Geometriai diszkretizáció

Hex vs Tet Mesh : Accuracy comparison

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



Contours of temperature for inviscid flow

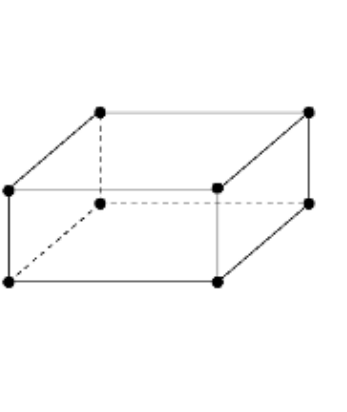
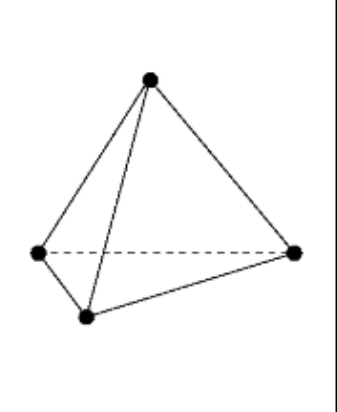
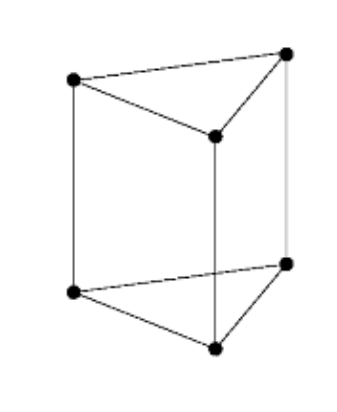
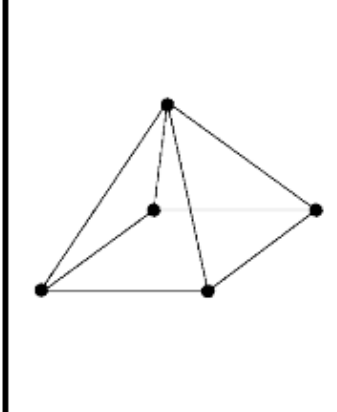
Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)



CFD feladat főbb lépései - Geometriai diszkretizáció

Element Types

- Common 3-D element types:

Hexahedron (hex)	Tetrahedron (tet)	Prism	Pyramid
			

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

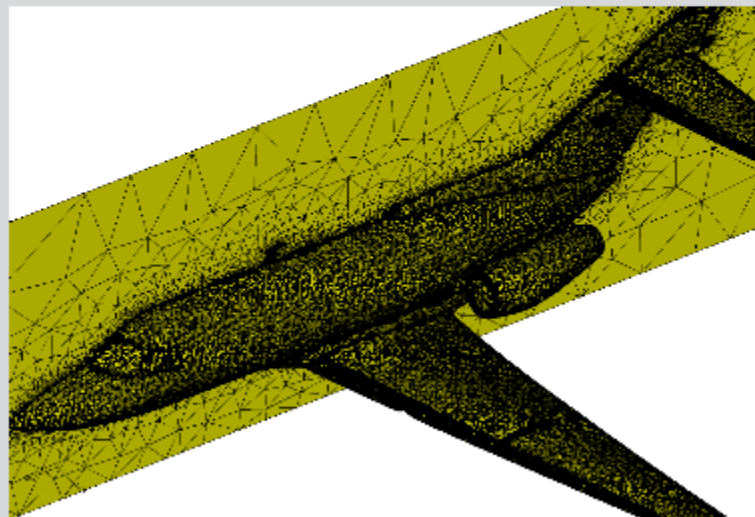
Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)



CFD feladat főbb lépései - Geometriai diszkretizáció

Elements: Tet

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



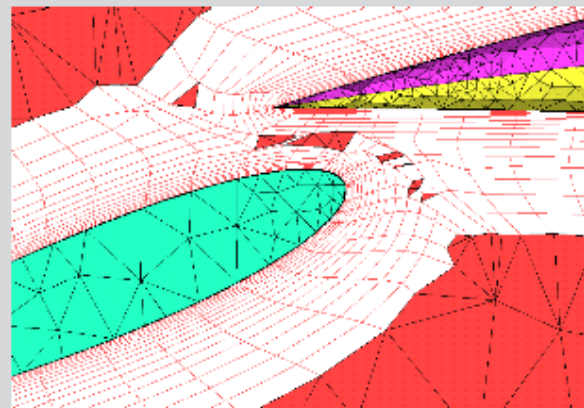
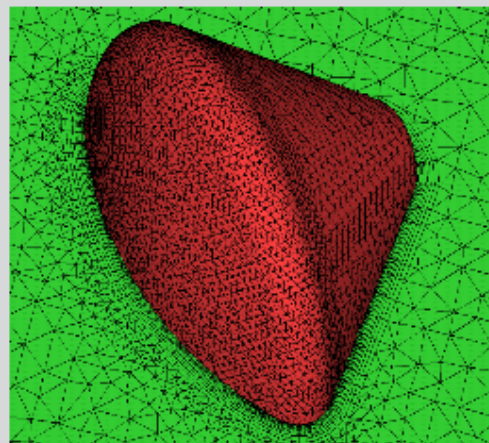
Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)



CFD feladat főbb lépései - Geometriai diszkretizáció

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



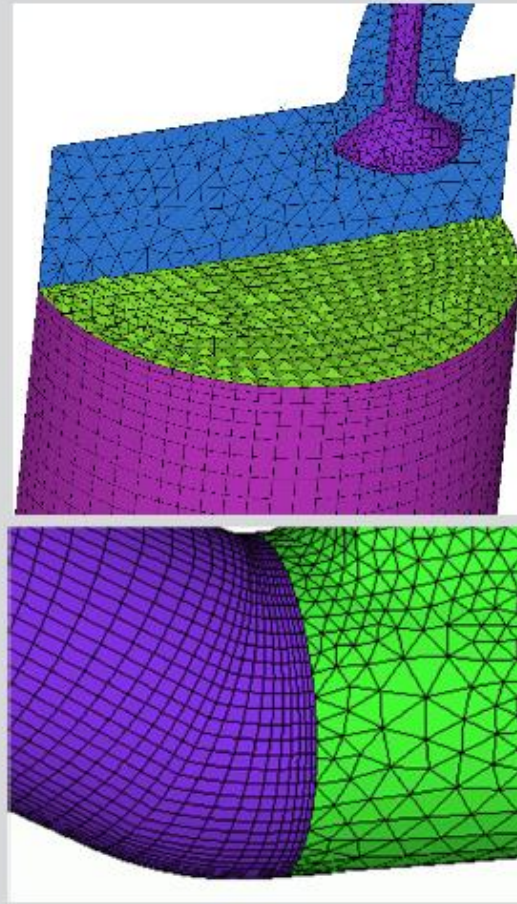
Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)



CFD feladat főbb lépései - Geometriai diszkretizáció

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.)



CFD feladat főbb lépései - Geometriai diszkretizáció

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.)



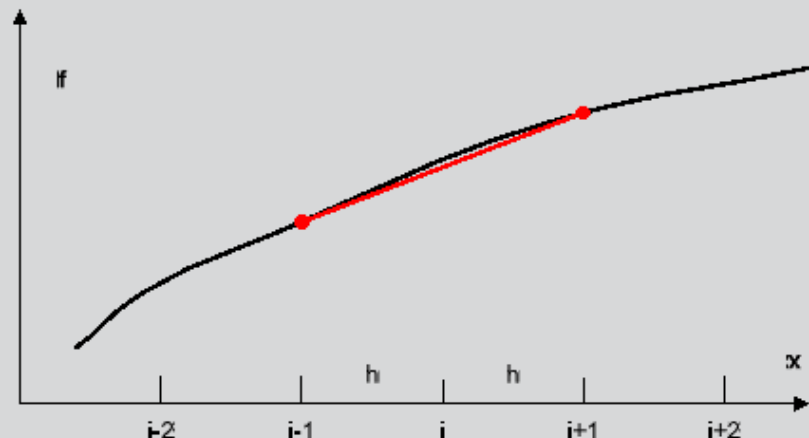
CFD feladat főbb lépései - Geometriai diszkretizáció

Grid Optimization

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

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

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

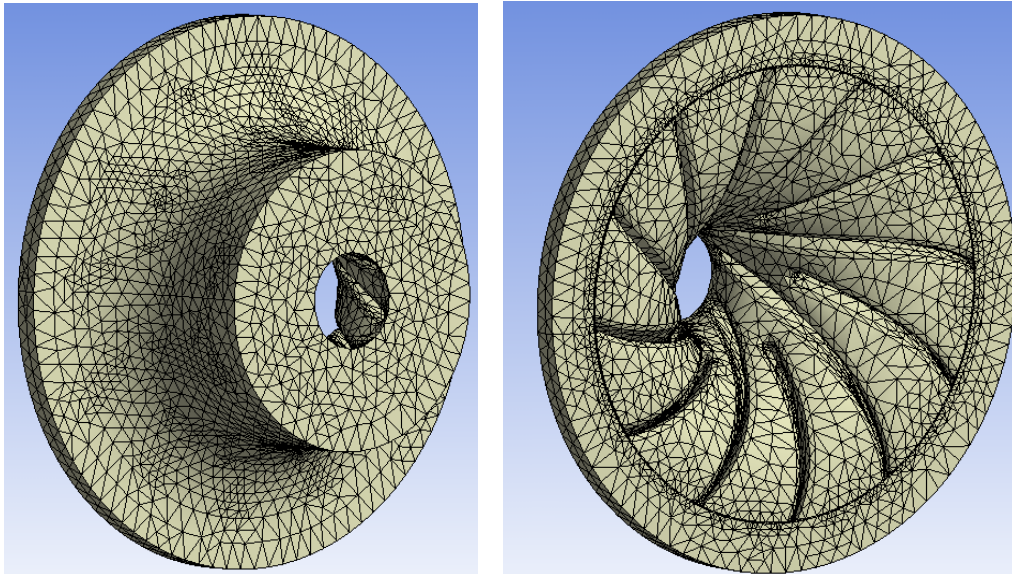


Forrás: Introduction to ANSYS CFX, Lecture 10 - Best Practice Guidelines - CFX-Intro_14.0_L10_BestPractices (2013.09.01.)



CFD feladat főbb lépései

Geometriai diszkretizáció



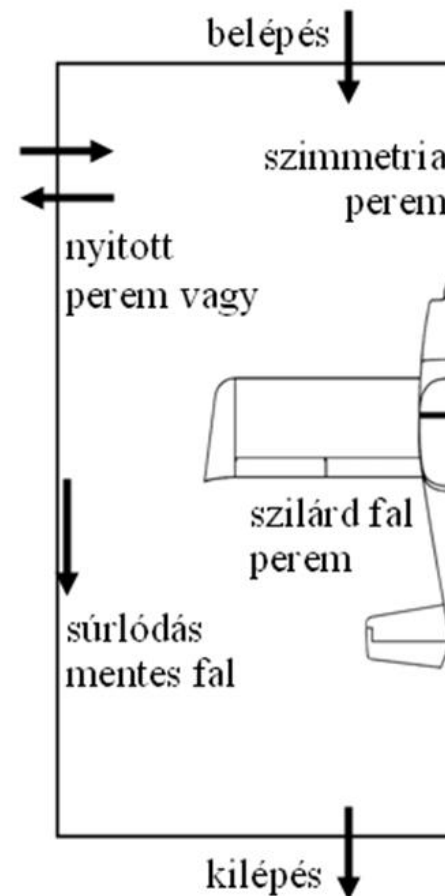
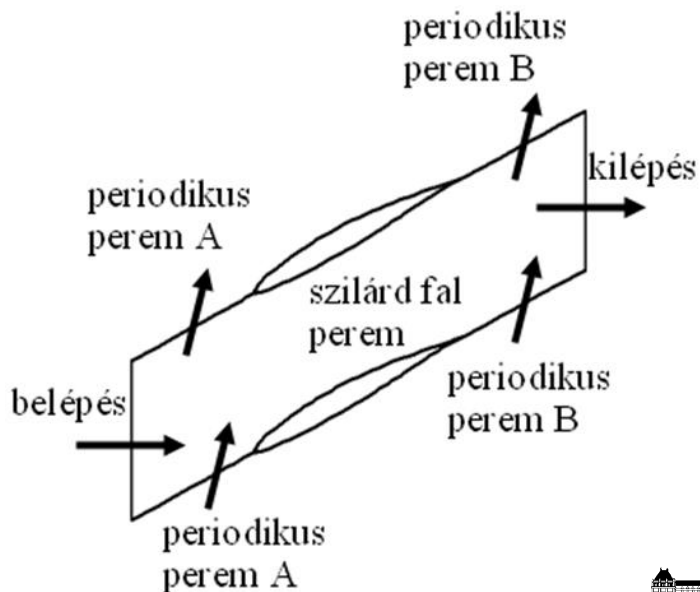
Centrifugál kompresszor-forgórész áramlási terének modellje. (A háló elemek méretét és minőségét, különösen a nagy gradienseket tartalmazó helyeken, addig kell javítani, ameddig nem lesz hatással a végeredményre. Gyakorlati tapasztalataink azt mutatják, hogy legalább 10 cellára szükség van a legszűkebb keresztmetszetekben is összenyomható közegek modellezése esetén.)



CFD feladat főbb lépései

Peremfeltételek definiálása és anyagtulajdonságok megadása

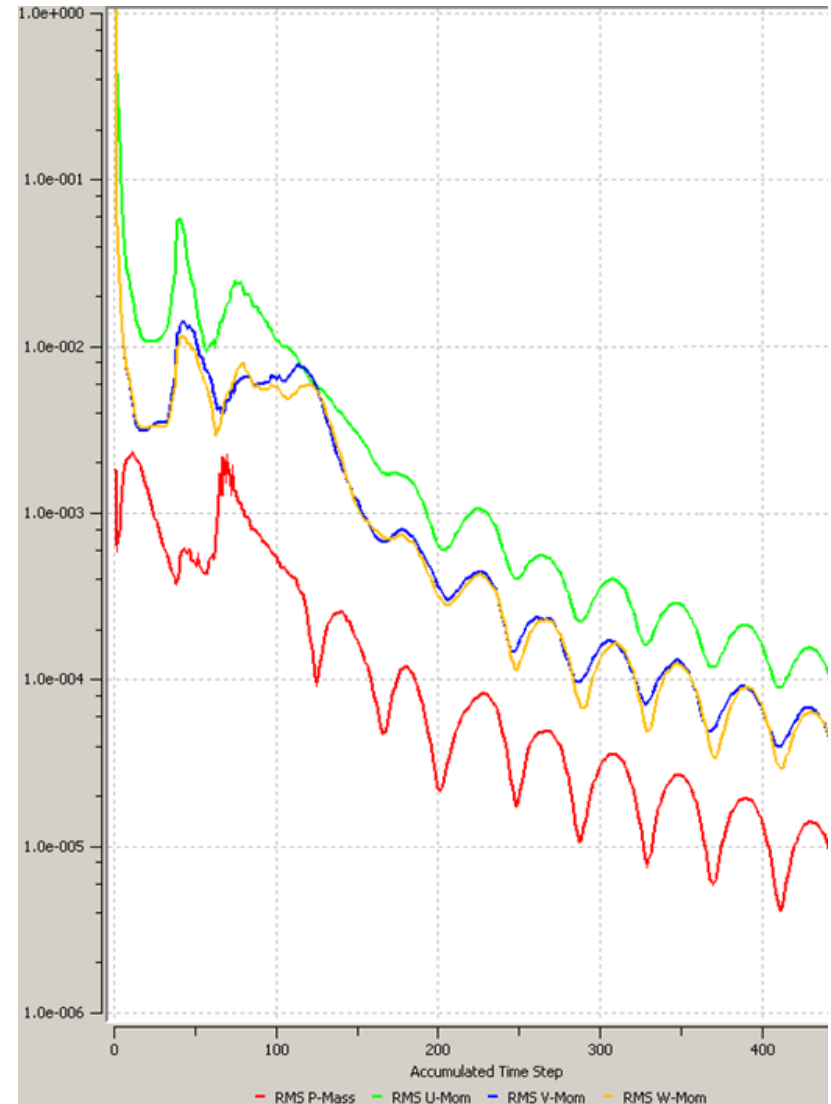
Jellemző peremfeltételek CFD számítások esetén; belépés, kilépés, szilárd fal, nyitott és periodikus perem (megj.: a peremeket olyan távol célszerű felvenni a geometriától, hogy annak zavaró hatása lehetőleg ne terjedjen el a peremig (pl. kilépő peremnél visszaáramlás, áramvonalak ne lépjenek ki, illetve be a nyitott (távoltéri) peremnél), illetve ne legyen hatása a végeredményre a cellaszám minimalizálása és az eredmények hálófüggetlensége mellett.)



CFD feladat főbb lépései

Konvergencia

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



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 $[\Phi]$ 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.)



CFD feladat főbb lépései

Konvergencia

Residuals

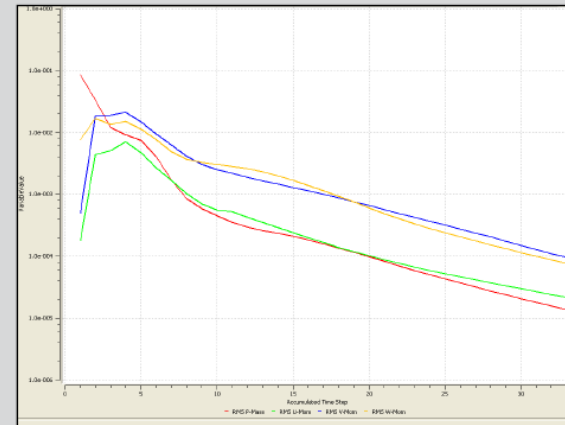
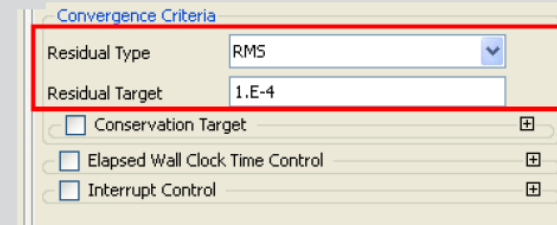
- **Residual Type**

- MAX: Convergence based on maximum residual anywhere
- RMS: Convergence based on average residual from all control volumes

- Root Mean Square =
$$\sqrt{\frac{\sum_i R_i^2}{n}}$$

- **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.)



CFD feladat főbb lépései

Konvergencia

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%

Convergence Criteria

Residual Type: RMS

Residual Target: 1.E-4

Conservation Target

Value: 0.01

Elapsed Wall Clock Time Control

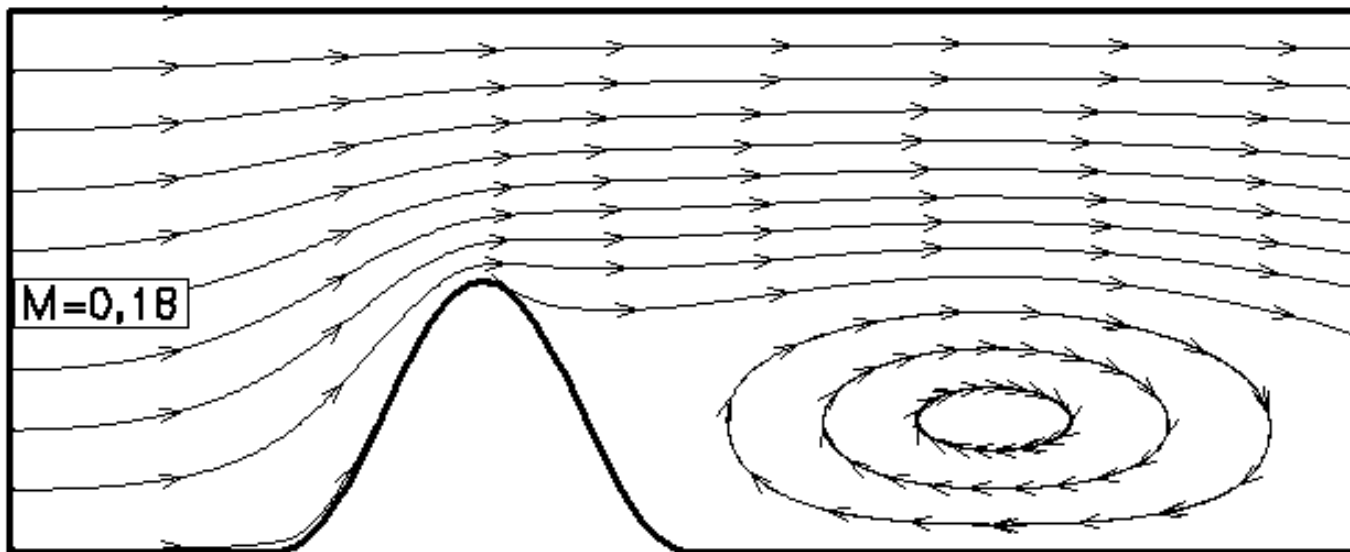
Interrupt Control

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



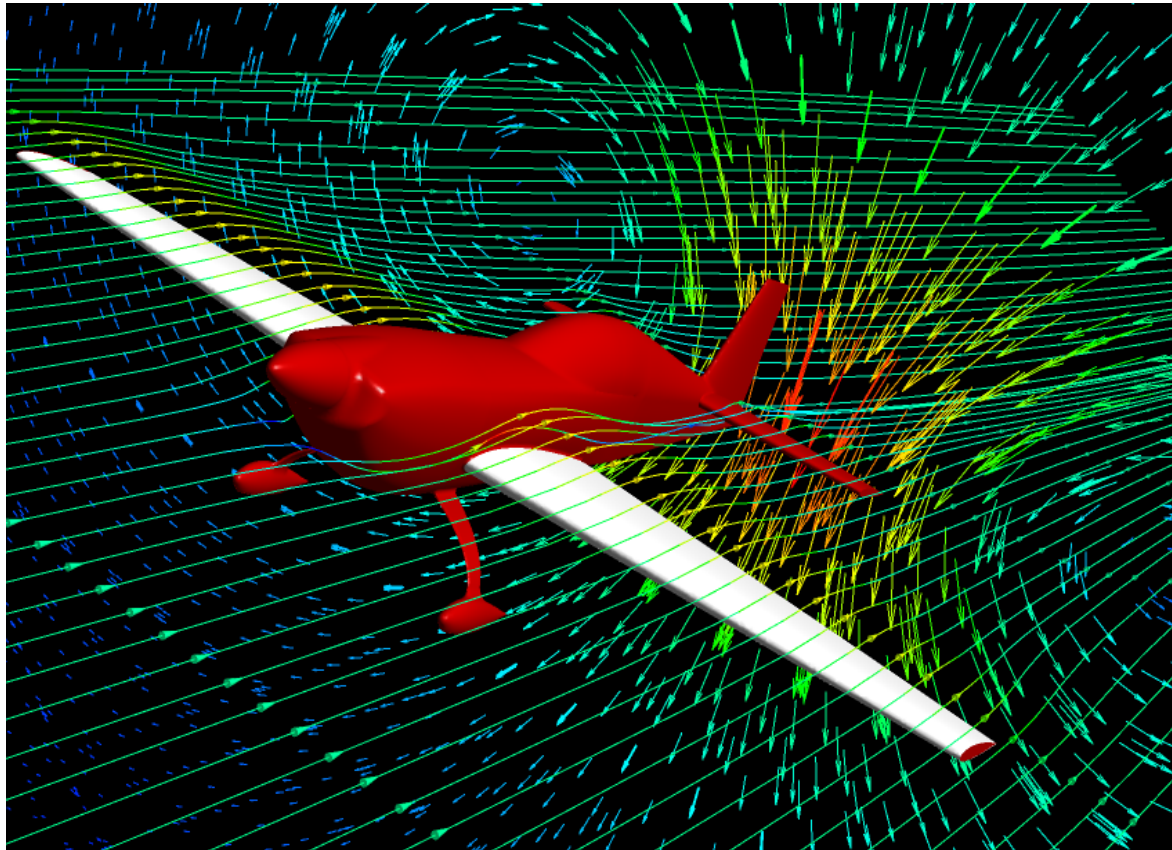
CFD feladat főbb lépései

Eredmények megjelenítése



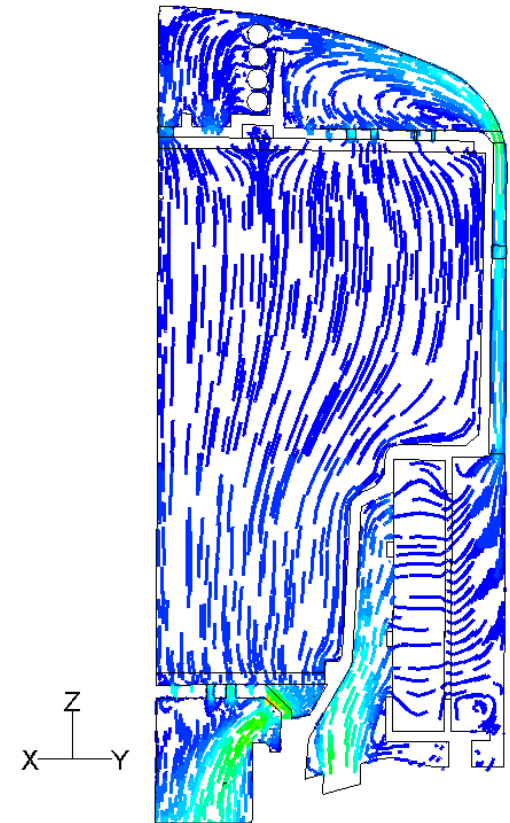
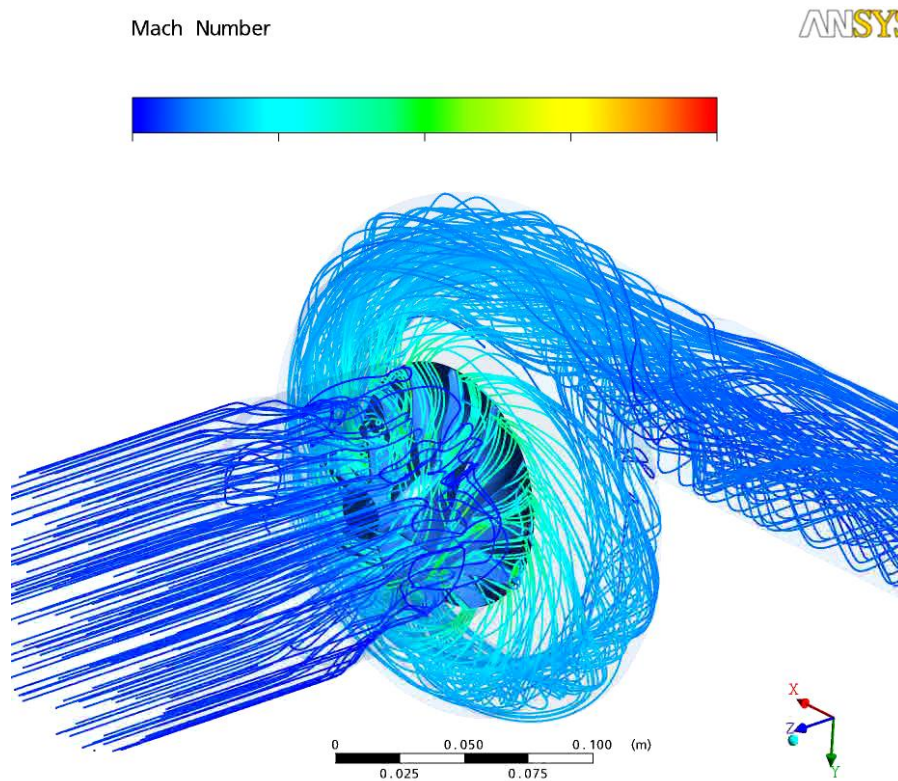
CFD feladat főbb lépései

Eredmények megjelenítése



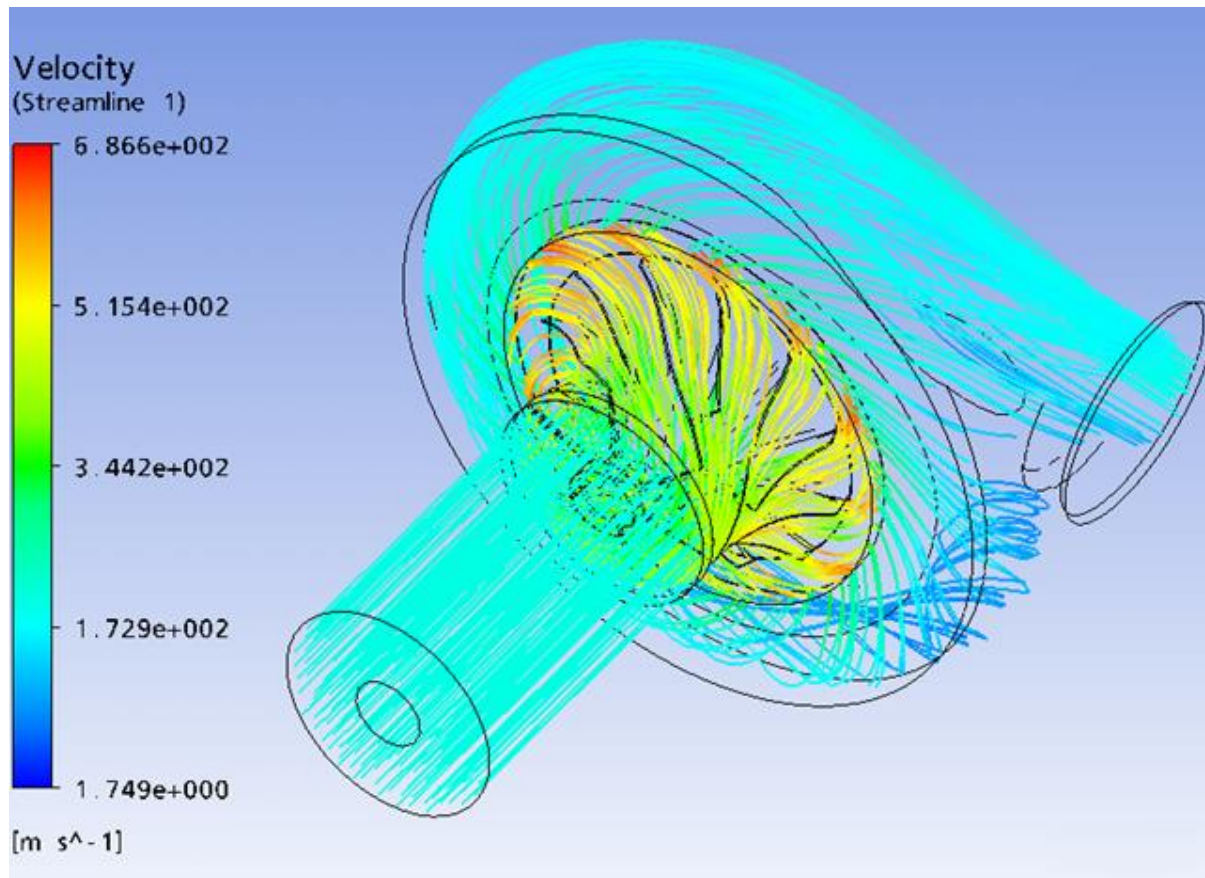
CFD feladat főbb lépései

Eredmények megjelenítése



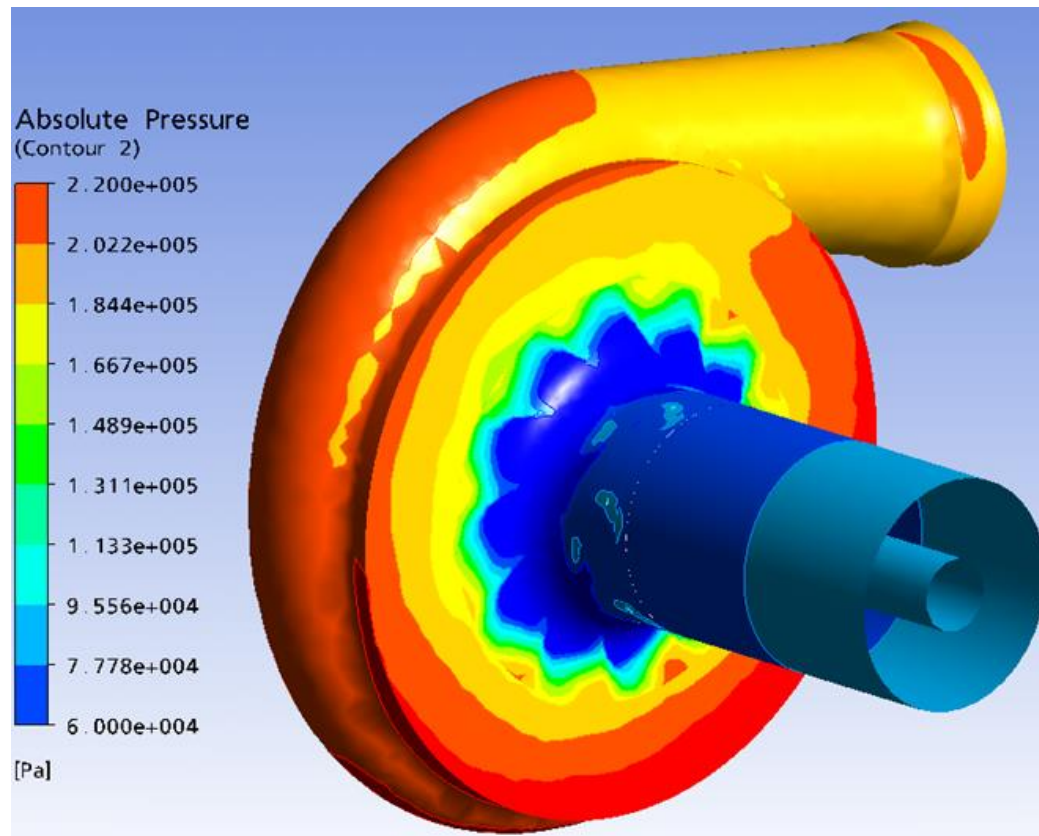
CFD feladat főbb lépései

Eredmények megjelenítése



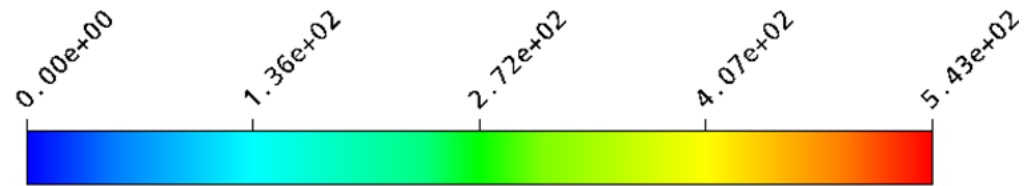
CFD feladat főbb lépései

Eredmények megjelenítése

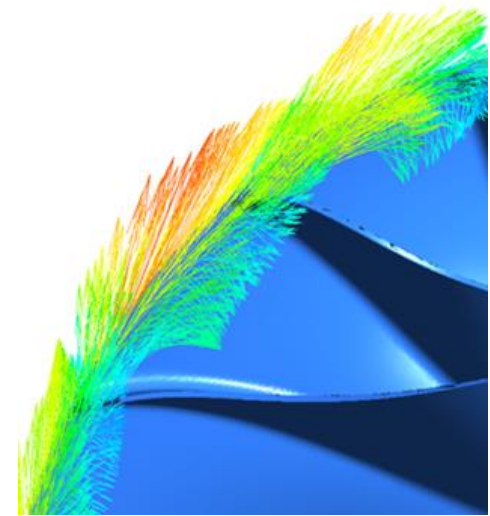
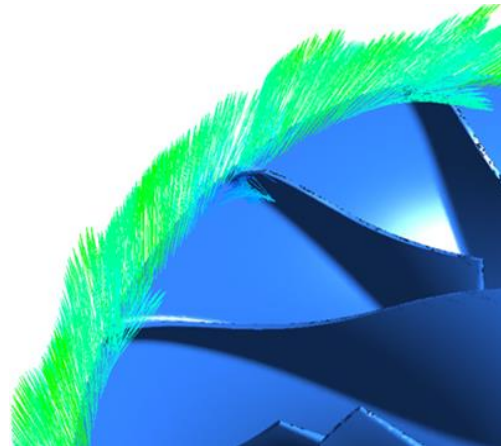
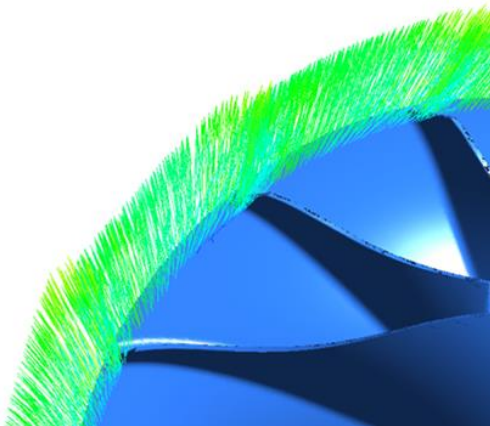


CFD feladat főbb lépései

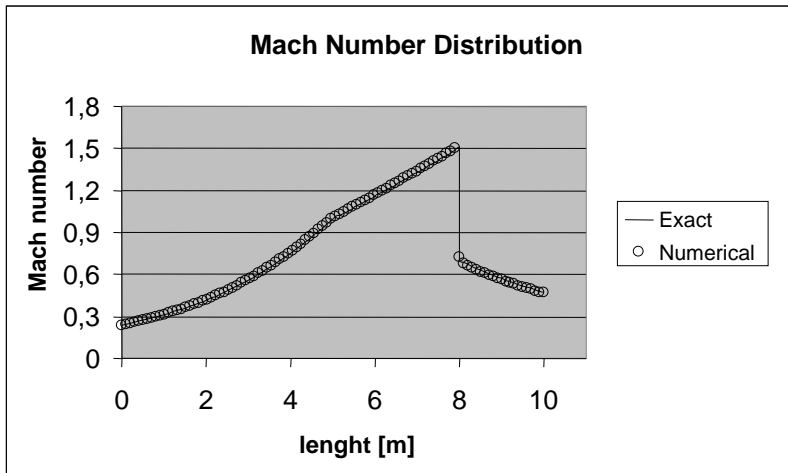
Eredmények megjelenítése



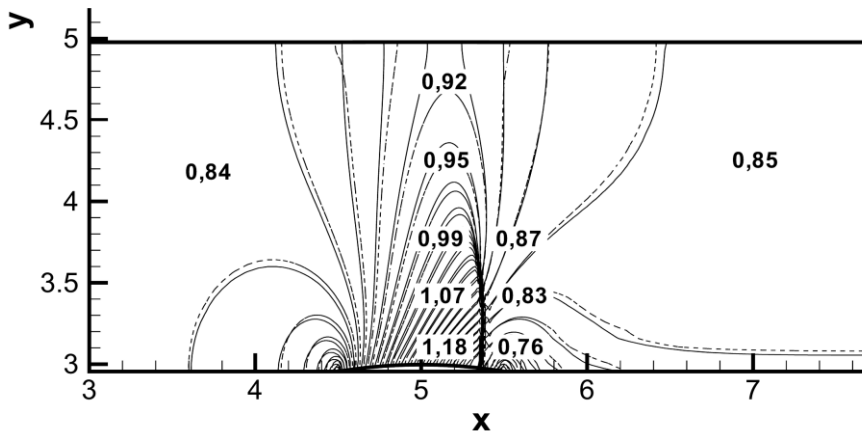
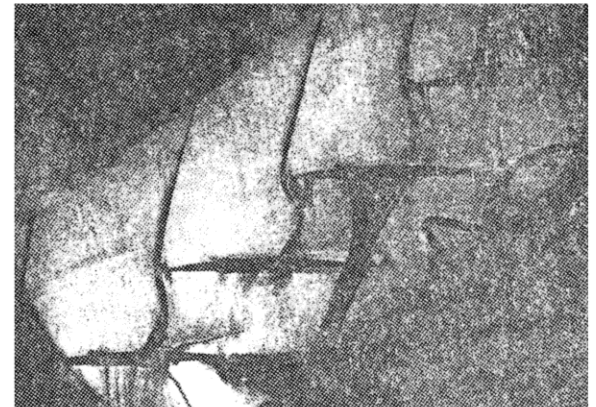
Velocity (Vector 1)
[m s⁻¹]



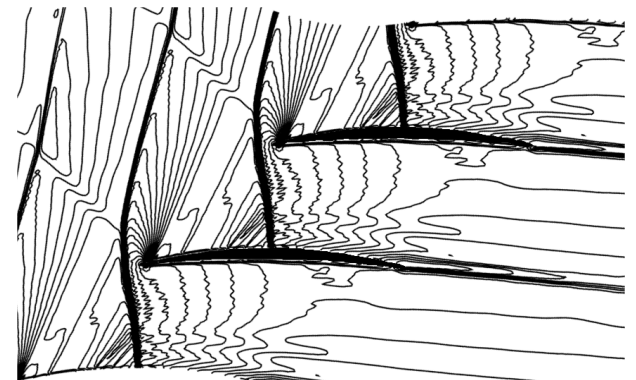
Validáció - (súrlódásmentes áramlás Euler egyenlet)



$M_1=1.1$



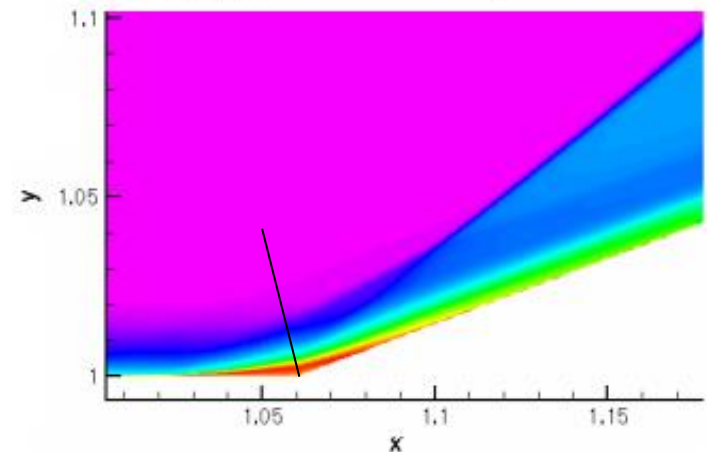
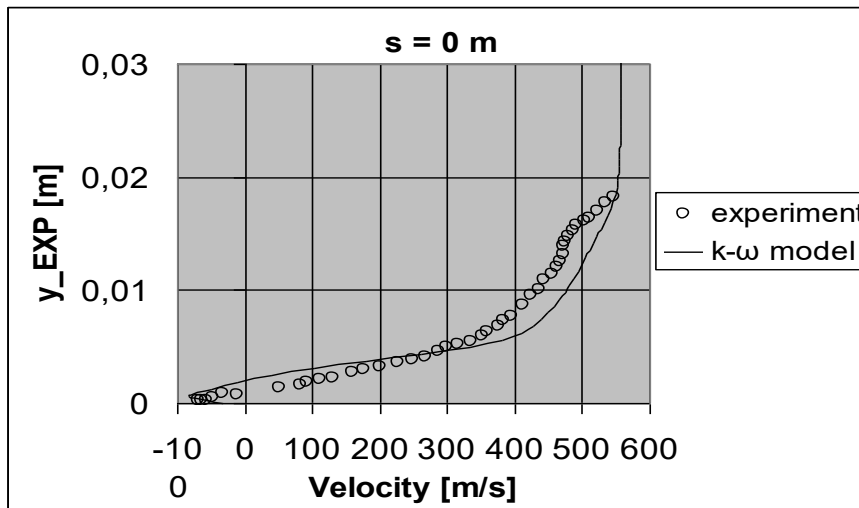
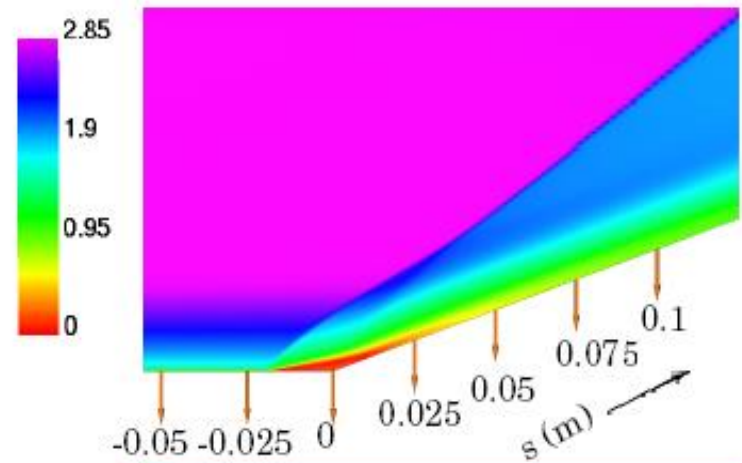
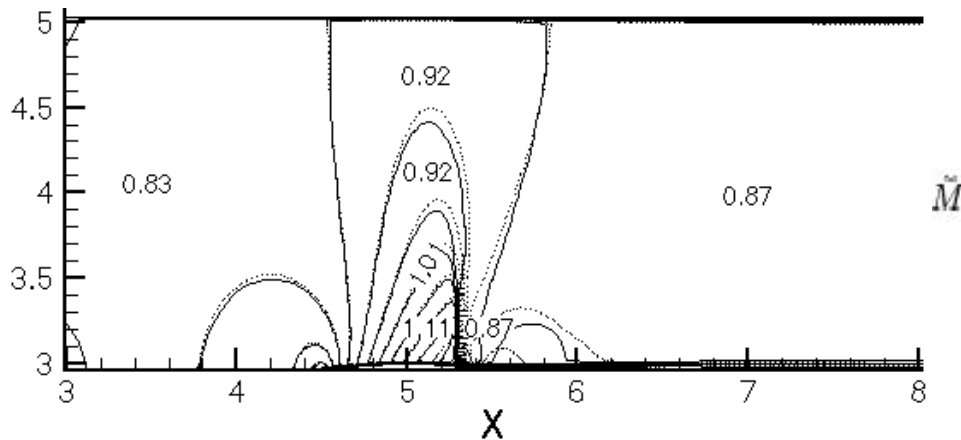
$M_1=1.1$



A mérés forrásanyaga: Starken, H., 1971, "Untersuchung der Strömung in ebenen Überschallverzögerungsgittern," DLR-Forschungsbericht 71-99.



Validáció (súrlódásos áramlás NS egyenlet)



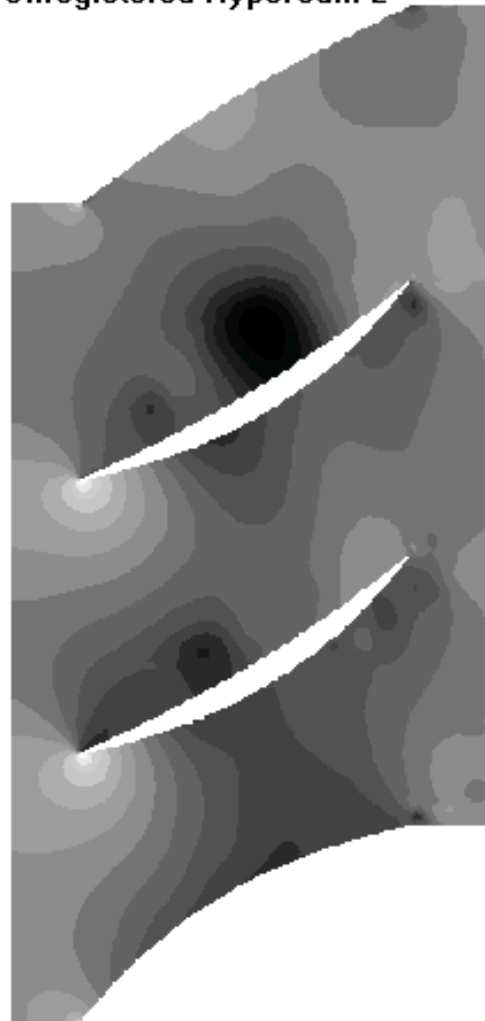
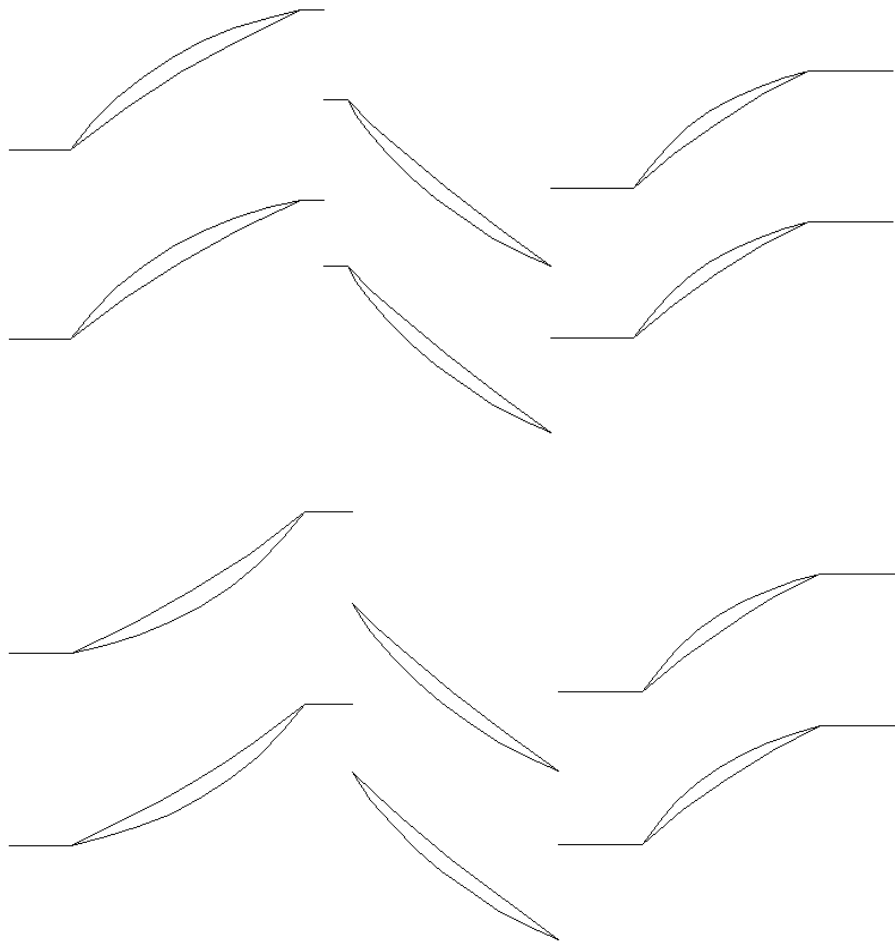
A mérés forrása: Gerolymos, G. A.; Sauret, E. & Vallet, I. (2003). *Oblique-Shock-Wave/Boundary-Layer Interaction using Near-Wall Reynolds-Stress Models*, Université Pierre-et-Marie-Curie, AIAA 2003-3466, 33rd Fluid Dynamics Conference, 23-26 June 2003 Orlando, Florida, USA



DASFLOW Program - Lapátrács numerikus áramlástanai vizsgálata

Mach szám eloszlás

Unregistered HyperCam 2



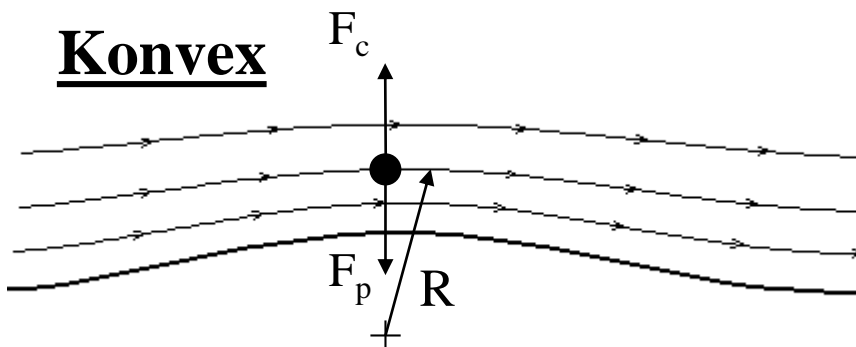
Mellékletek

Konkáv-konvex görbületi hatás



Konkáv - konvex görbületi hatás

Konvex

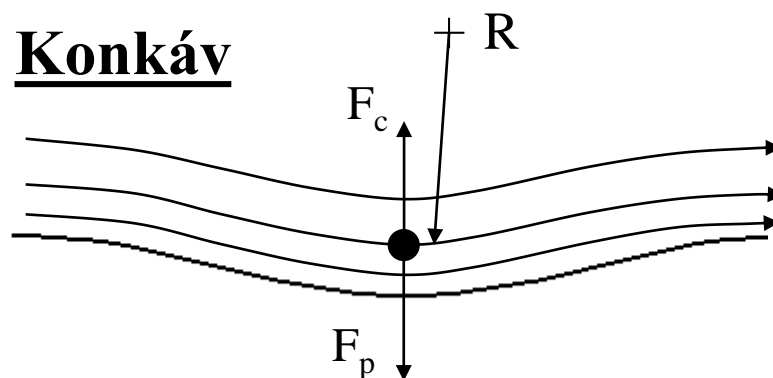


Ha $R \downarrow \Rightarrow mR\omega^2$, $F_c \downarrow \Rightarrow F_p \downarrow$

Ha $F_p \downarrow \Rightarrow p \downarrow \Rightarrow v \uparrow$

$$\frac{p}{\rho} + \frac{1}{2}v^2 \approx \text{állandó}$$

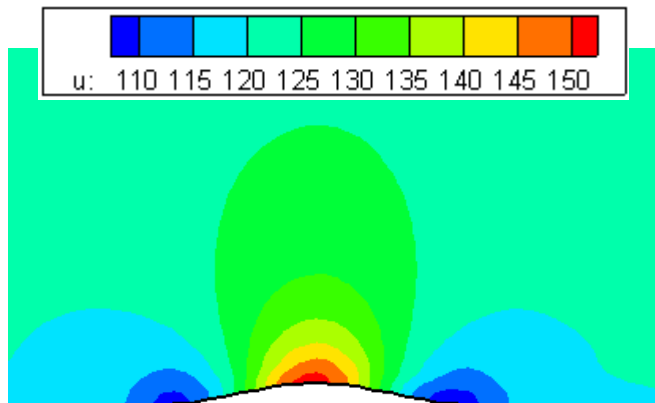
Konkáv



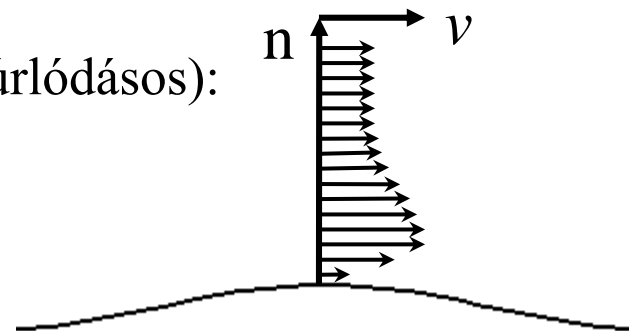
Ha $R \uparrow \Rightarrow mR\omega^2$, $F_c \uparrow \Rightarrow F_p \uparrow$

Ha $F_p \uparrow \Rightarrow p \uparrow \Rightarrow v \downarrow$

$$\frac{p}{\rho} + \frac{1}{2}v^2 \approx \text{állandó}$$



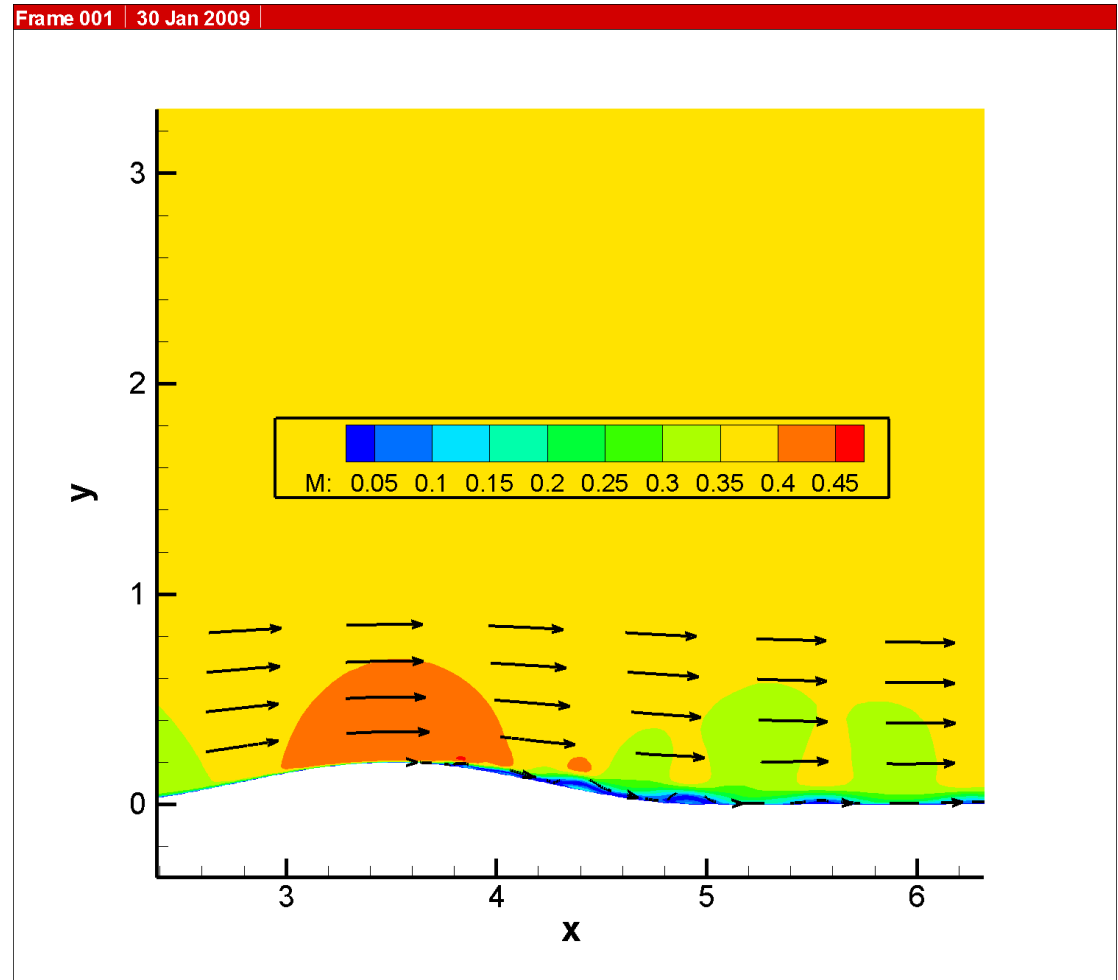
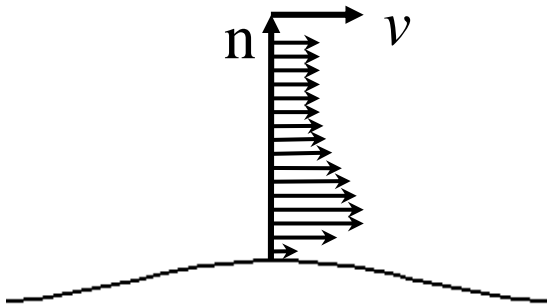
Valóságos (súrlódásos):



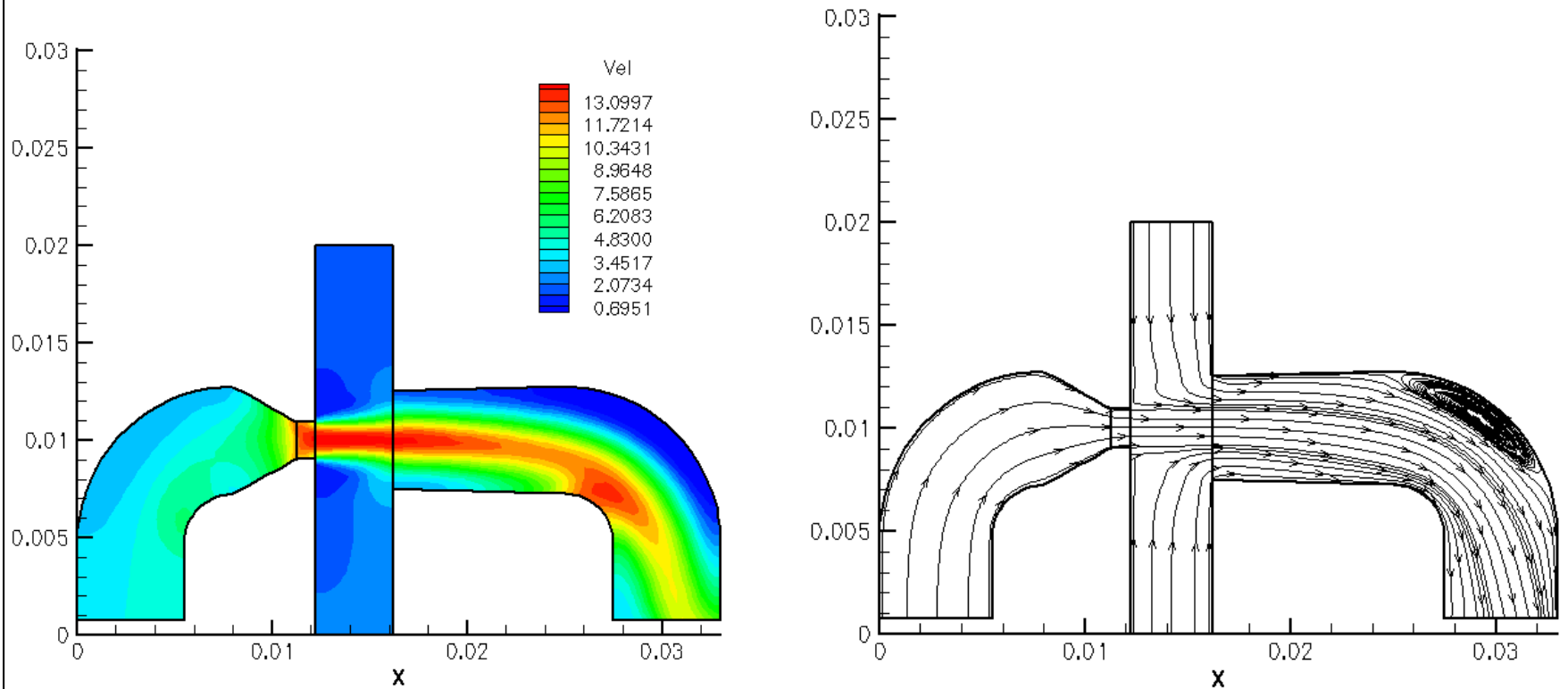
Konkáv - konvex görbületi hatás

Konvex

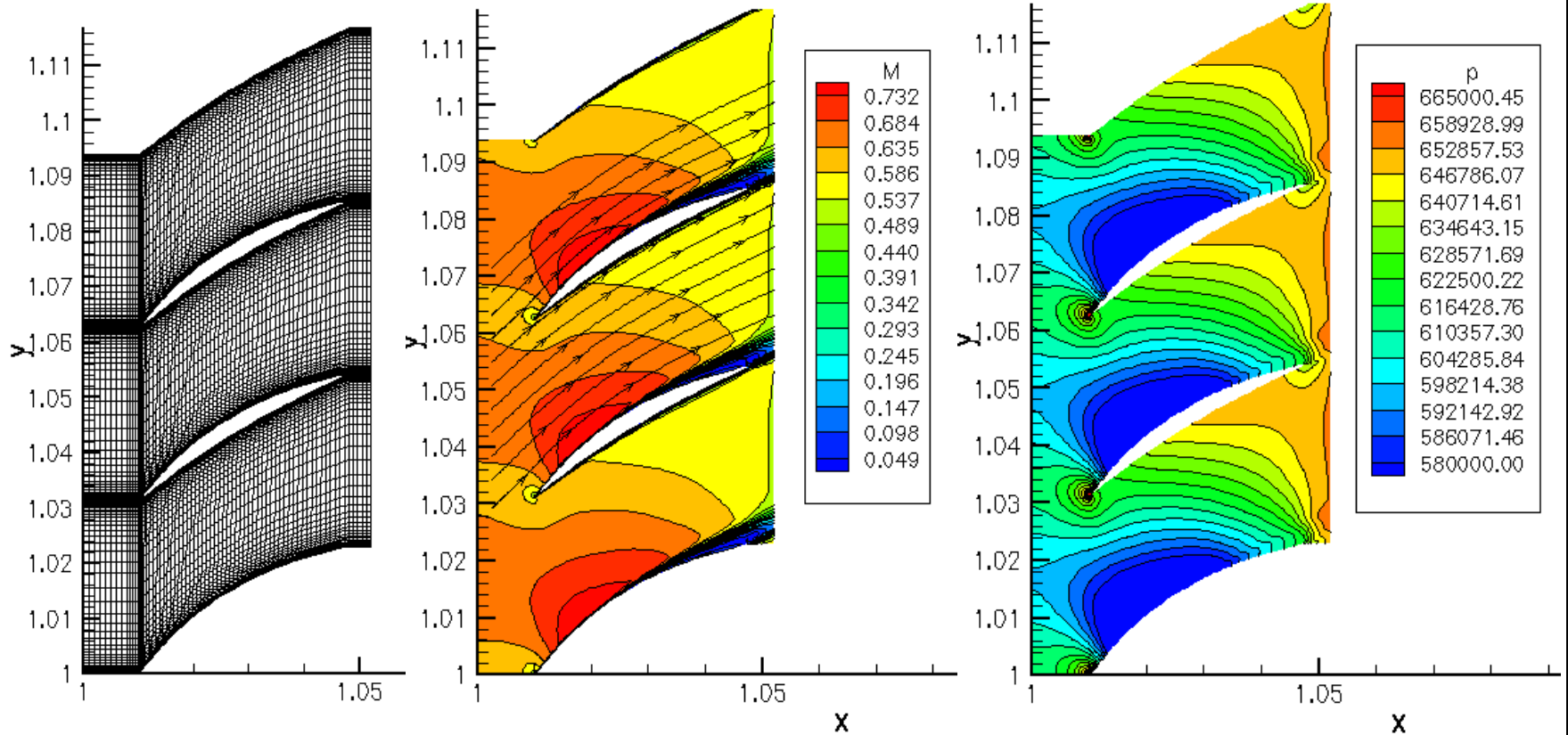
Valóságos (súrlódásos):



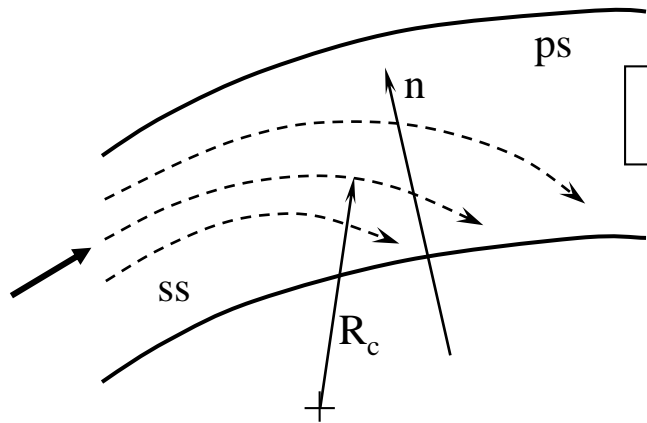
Konkáv - konvex görbületi hatás



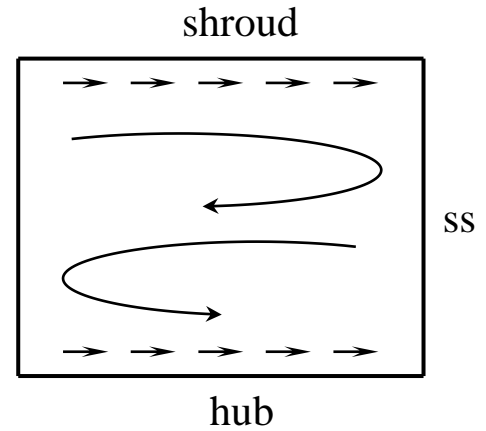
Konkáv - konvex görbületi hatás



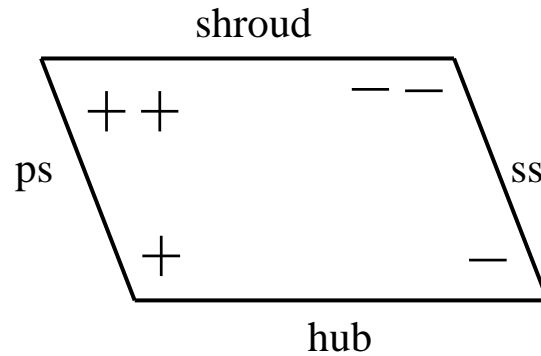
Másodlagos áramlások kialakulása lapátrácsokban



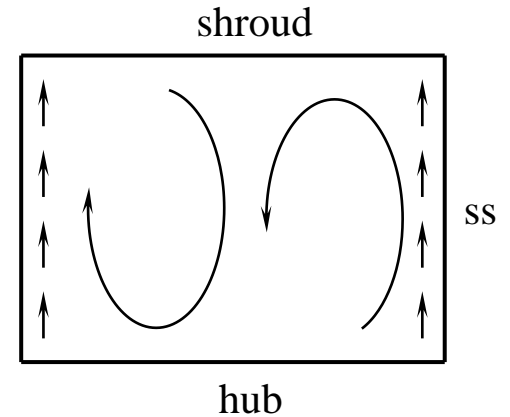
Csatorna örvény



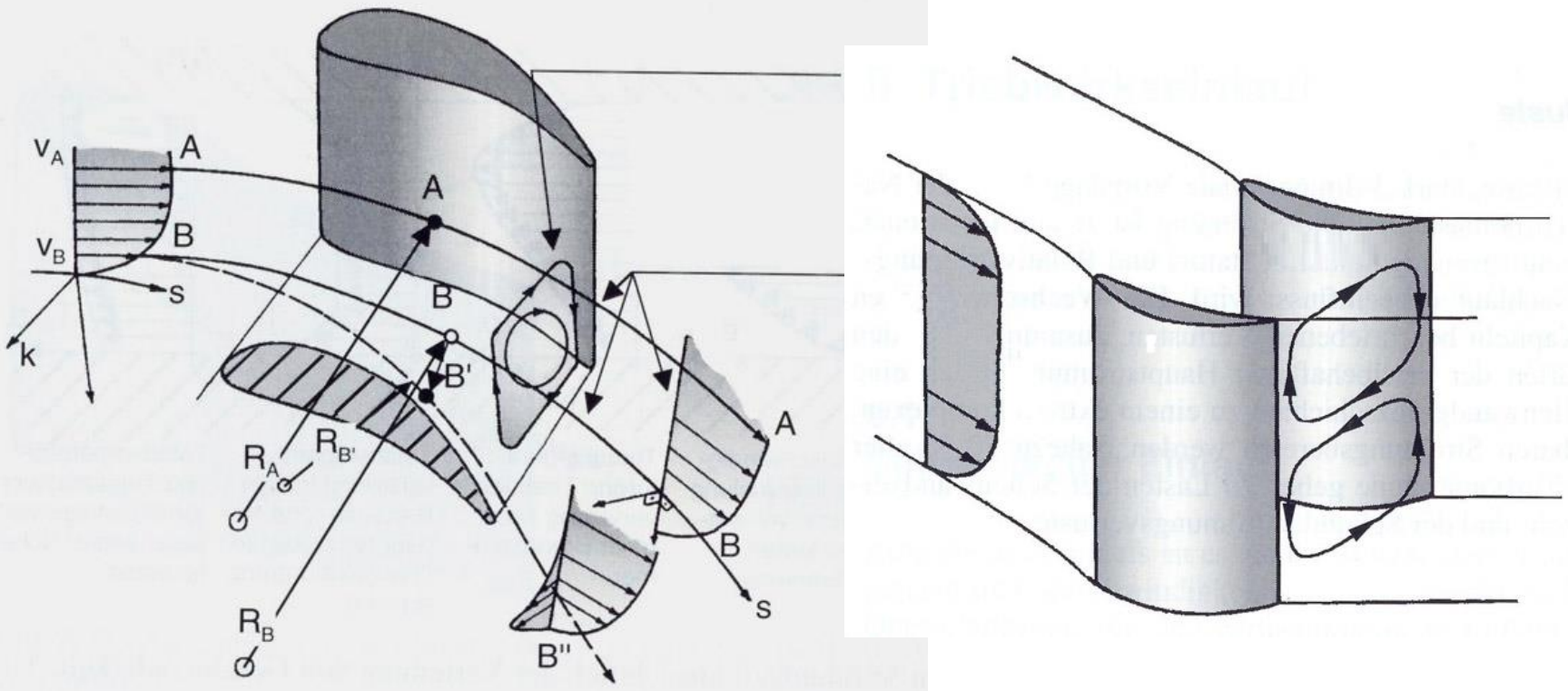
$$\frac{\partial p}{\partial n} = \frac{v^2}{R_c}$$



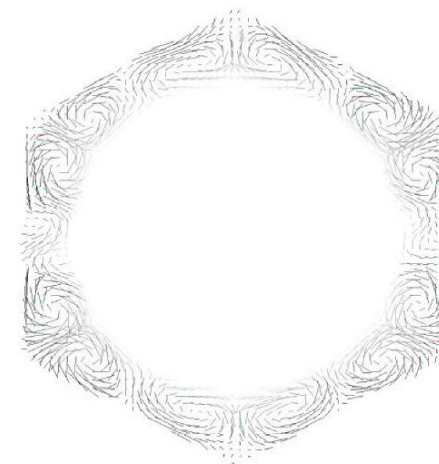
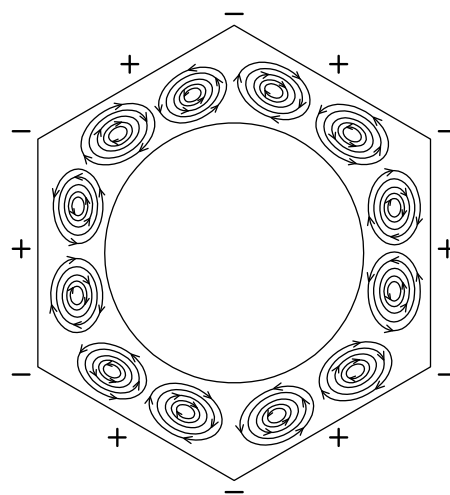
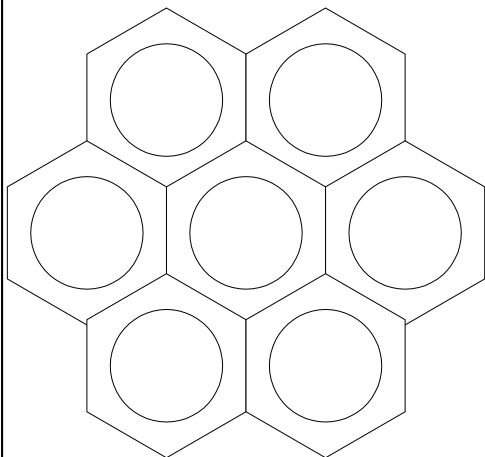
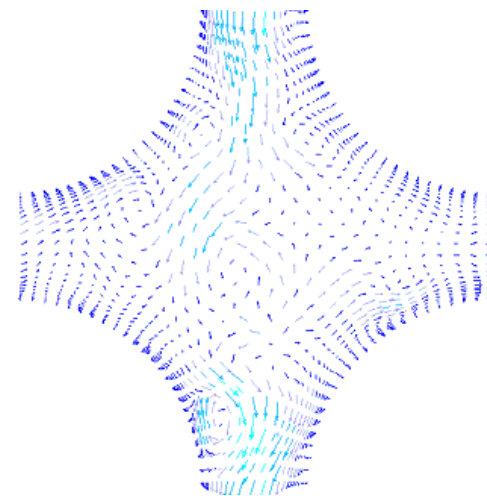
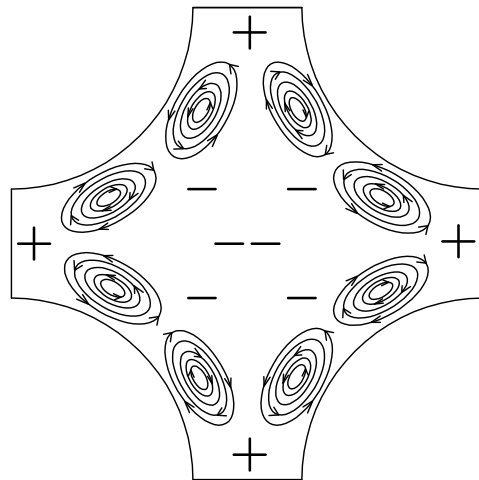
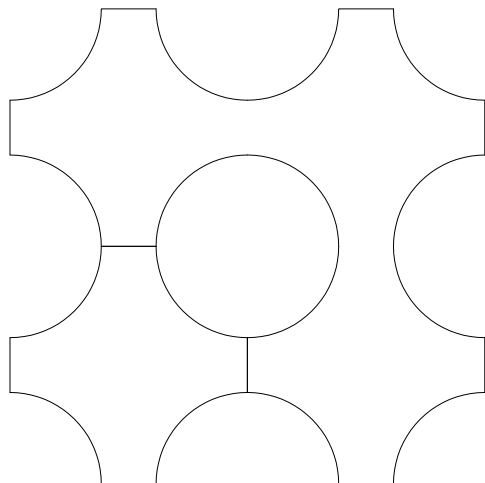
Lapát örvény



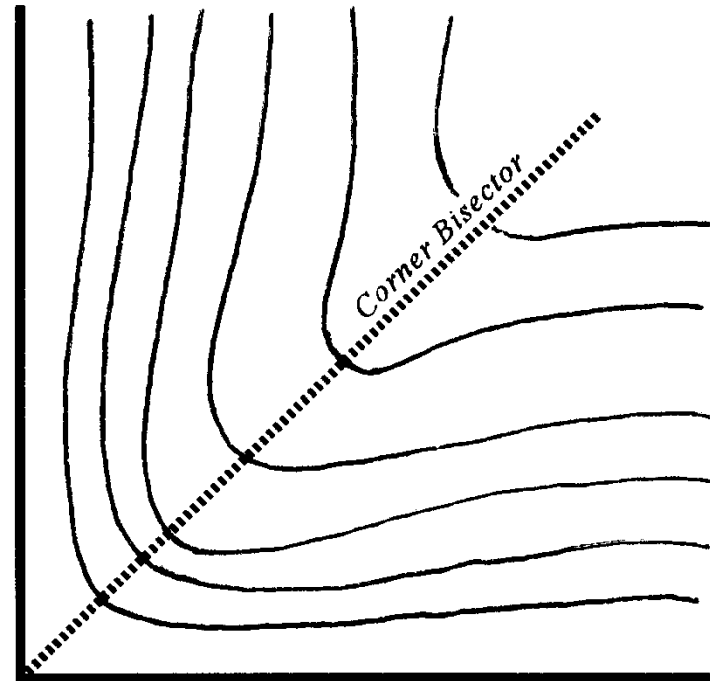
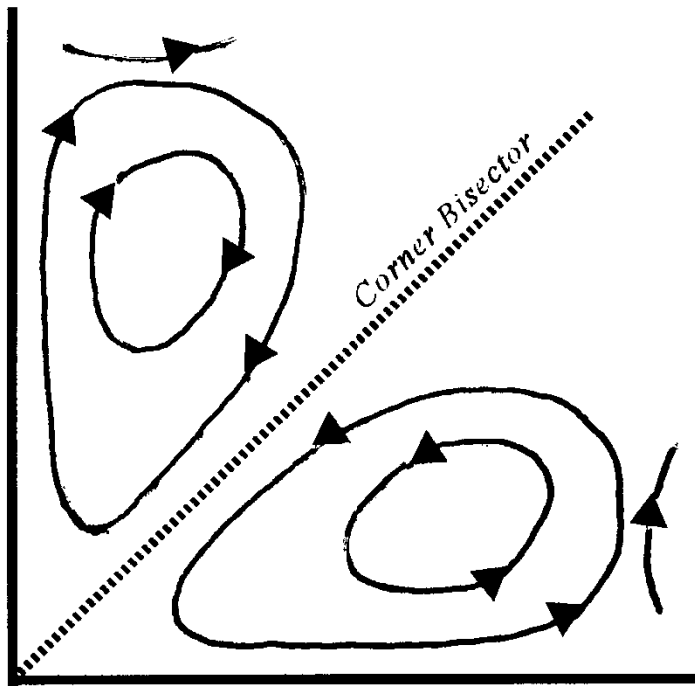
Másodlagos áramlások kialakulása lapátrácsokban (csatornaörvények)



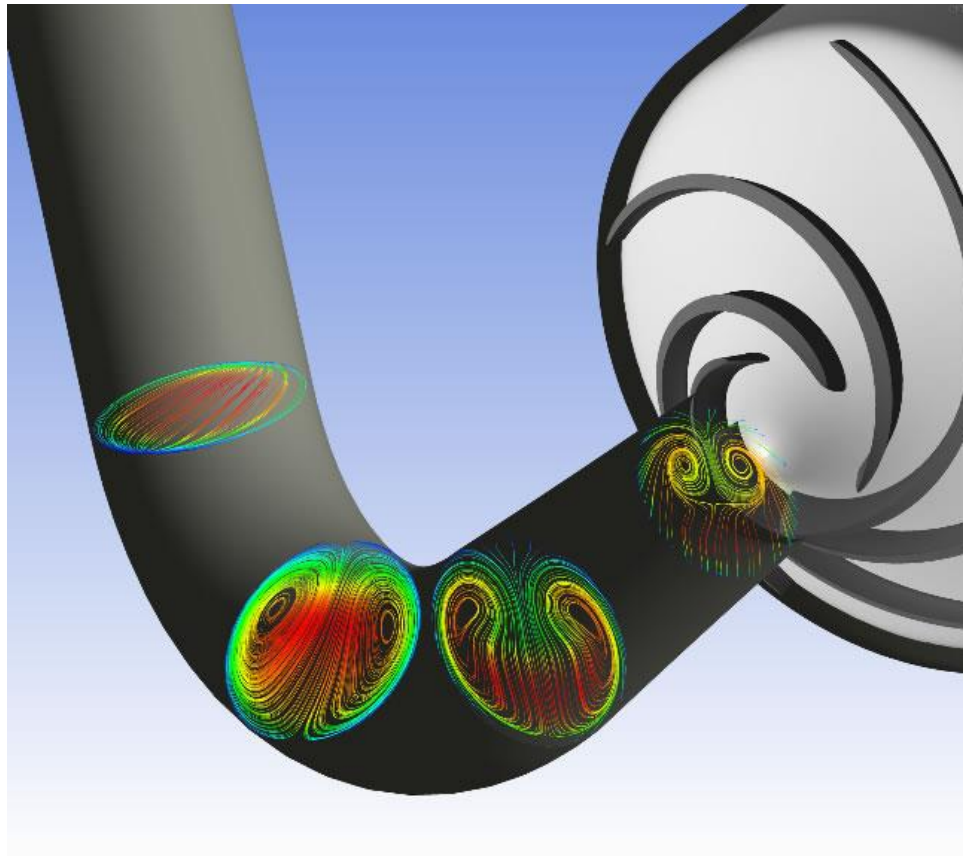
Másodlagos áramlások kialakulása csőkötegekben



Másodlagos áramlások kialakulása téglalap keresztmetszetű csatornában



Másodlagos áramlások kialakulása csőkönyvekben

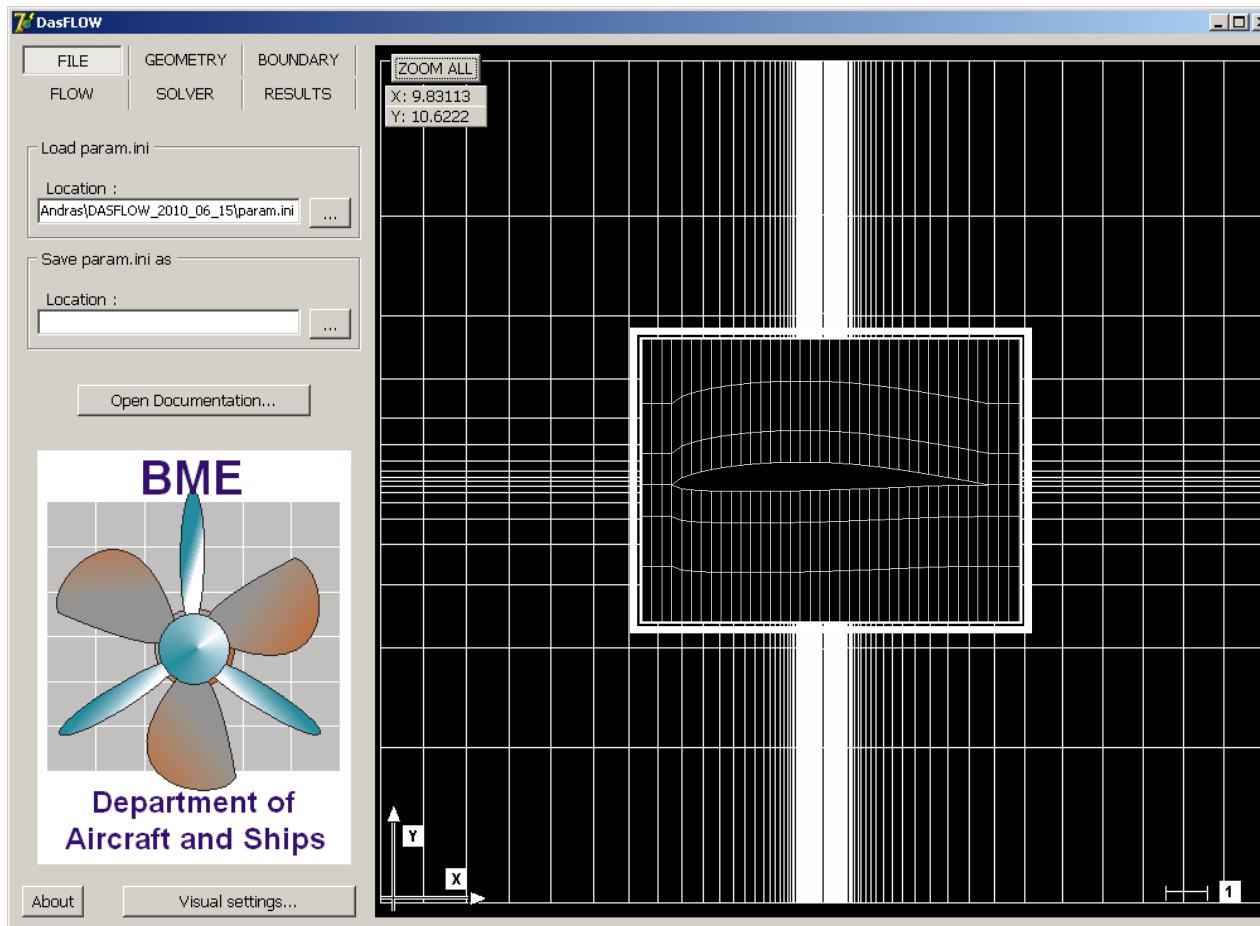


Mellékletek

DASFLOW Software és ipari alkalmazásai



DASFLOW – Tanszéki fejlesztésű CFD és inverz tervezésre alkalmas program



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Reynolds és Favre átlagolás - DASFLOW

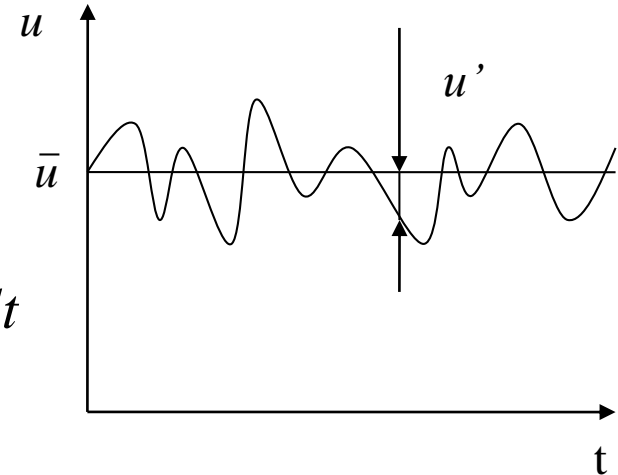
$$u = \bar{u} + u' \quad v = \bar{v} + v'$$

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

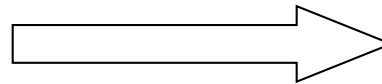
$$\bar{u} = \frac{1}{\Delta t} \int_{t_0}^{t_0+\Delta t} u dt$$

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

$$\tilde{u} = \frac{1}{\bar{\rho}} \frac{1}{\Delta t} \int_{t_0}^{t_0+\Delta t} (\rho u) dt$$



Reynolds Átlagolás



Favre Átlagolás

Nagy sebességű,
összenyomható áramlás esetén.

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

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



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

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

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

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

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

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

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

$$\begin{aligned} & \frac{\partial}{\partial t} \left[\bar{\rho} \left(\tilde{e} + \frac{1}{2} \sum_{i=1}^3 \tilde{u}_i \tilde{u}_i \right) + \frac{1}{2} \sum_{i=1}^3 \overline{\rho u_i'' u_i''} \right] + \sum_{j=1}^3 \frac{\partial}{\partial x_j} \left[\bar{\rho} \tilde{u}_j \left(\tilde{h} + \frac{1}{2} \sum_{i=1}^3 \tilde{u}_i \tilde{u}_i \right) + \tilde{u}_j \frac{1}{2} \sum_{i=1}^3 \overline{\rho u_i'' u_i''} \right] = \\ & = \sum_{j=1}^3 \frac{\partial}{\partial x_j} \left[-\bar{q}_j - \overline{\rho u_j'' h''} + \sum_{i=1}^3 \overline{\tau_{ji} u_i''} - \frac{1}{2} \sum_{i=1}^3 \overline{\rho u_j'' u_i'' u_i''} \right] + \sum_{j=1}^3 \frac{\partial}{\partial x_j} \left[\sum_{i=1}^3 \left[\tilde{u}_i (\bar{\tau}_{ji} - \overline{\rho u_i'' u_j''}) \right] \right] \end{aligned}$$



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Reynolds és Favre átlagolás - DASFLOW

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

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

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

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

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

$$H(x) = \begin{cases} 0 & \text{if } x \leq 0 \\ 1 & \text{if } x > 0 \end{cases} \quad S_{ij} = \frac{1}{2} \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right) \quad F(M_t) = [M_t^2 - M_{t0}^2] H(M_t - M_{t0})$$

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



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Reynolds és Favre átlagolás – Konzervatív összevont forma - DASFLOW

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

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



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Konzervatív forma – RARS - DASFLOW

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

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

$$\hat{\Lambda} = \begin{bmatrix} \vec{V}\hat{n} \\ \vec{V}\hat{n} \\ \vec{V}\hat{n} + \hat{c} \\ \vec{V}\hat{n} - \hat{c} \end{bmatrix}$$

$$\Delta W_n = \begin{bmatrix} \Delta\rho - \frac{\Delta p}{\hat{c}^2} \\ \bar{s} \Delta\vec{V} \\ \bar{n} \Delta\vec{V} + \frac{\Delta p}{\hat{\rho}\hat{c}} \\ -\bar{n} \Delta\vec{V} + \frac{\Delta p}{\hat{\rho}\hat{c}} \end{bmatrix}$$

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

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

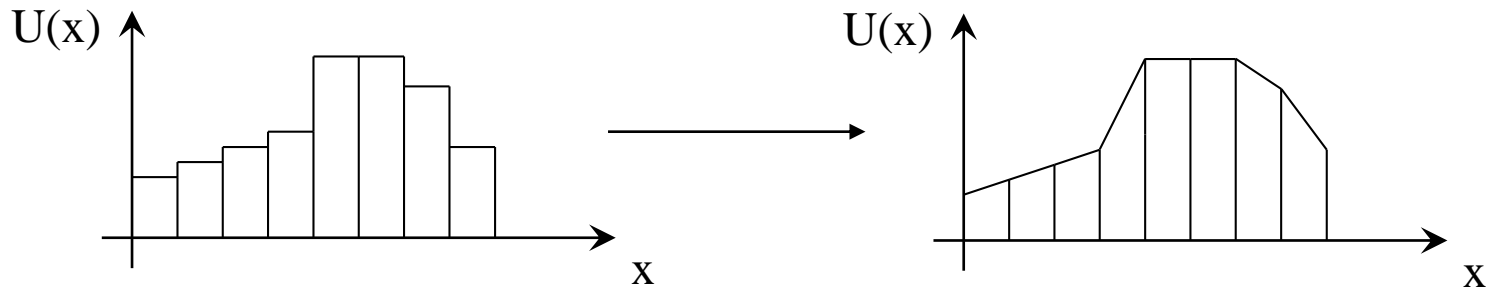
$$\hat{r}_n^3 = \left[\frac{\hat{\rho}}{2\hat{c}}, \frac{\hat{\rho}}{2\hat{c}}(\hat{u} + \hat{c}n_x), \frac{\hat{\rho}}{2\hat{c}}(\hat{v} + \hat{c}n_y), \frac{\hat{\rho}}{2\hat{c}} \left(\alpha + \frac{\hat{c}^2}{\beta} + \hat{c}\hat{V}_n \right) \right]^T$$

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



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Konzervatív forma – MUSCL - DASFLOW



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

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

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

$$\kappa = 1/3$$

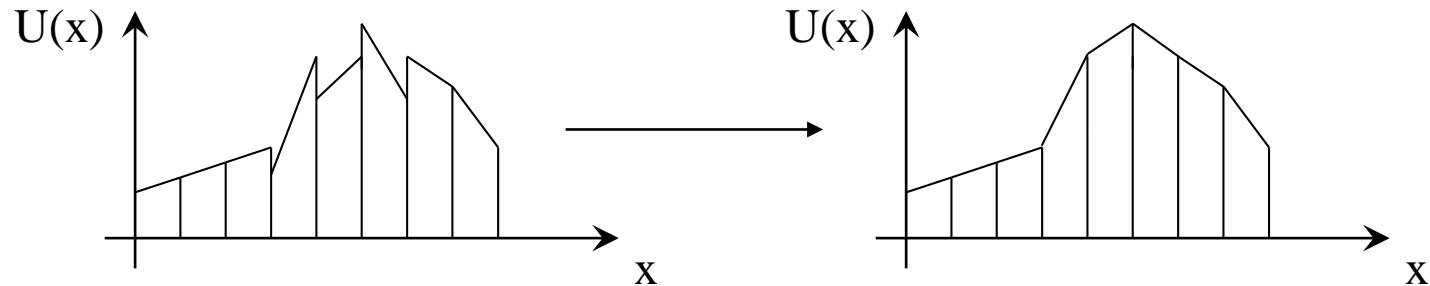
$$\kappa = [-1, 1]$$

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



Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Konzervatív forma – MUSCL (limiterek) - DASFLOW



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

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

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

$$\kappa = 1/3$$

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



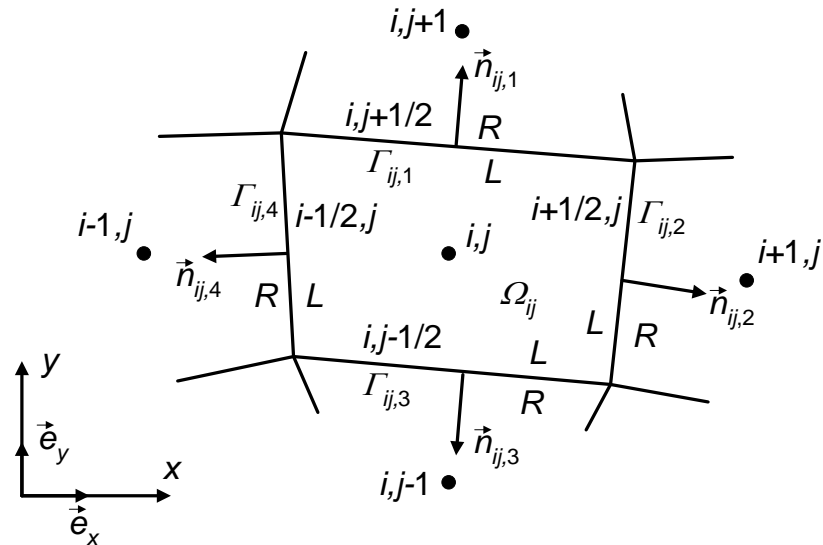
Áramlásmodellezés – Kontinuum-mechanika alapján

RANS egyenletek – Diszkretizáció - DASFLOW

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

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

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

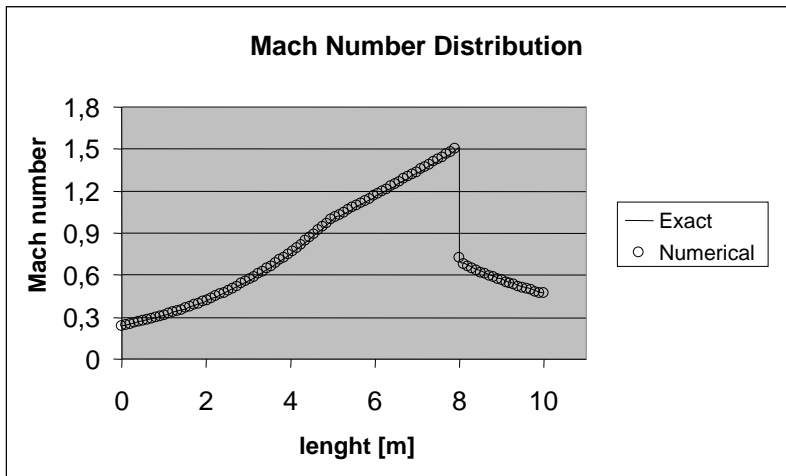


$$\frac{\partial}{\partial t} U_{ij} = -\frac{1}{A_{ij}} \left(\sum_{k=1}^4 ([H_n]_{ij,k} \Gamma_{ij,k}) - \sum_{k=1}^4 ([H_{vn}]_{ij,k} \Gamma_{ij,k}) \right) + [S(U)]_{ij} = \mathfrak{R}$$

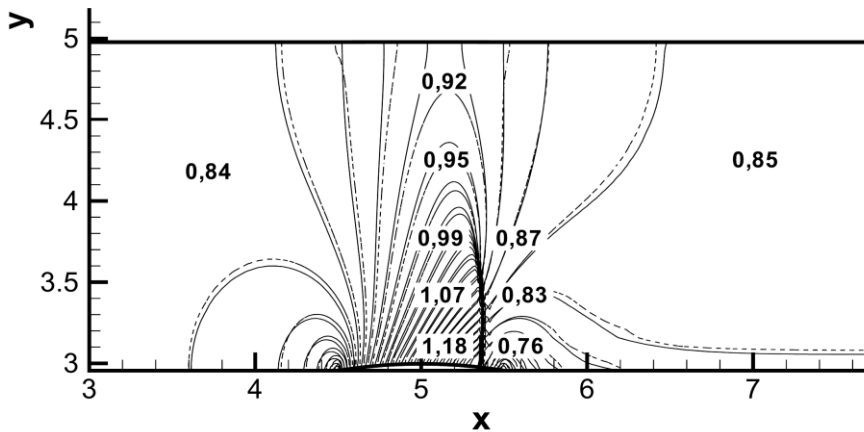
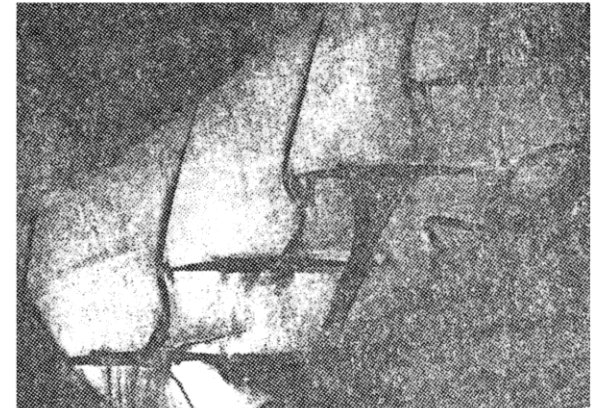
$$\left. \begin{aligned} U^0 &= U^n \\ U^k &= U^0 + \alpha_k \Delta t \mathfrak{R}(U^{k-1}) \quad k = 1, m \\ U^{n+1} &= U^m \end{aligned} \right\} \begin{aligned} &\bullet \text{ Runge-Kutta módszer} \\ &m=4 \end{aligned}$$



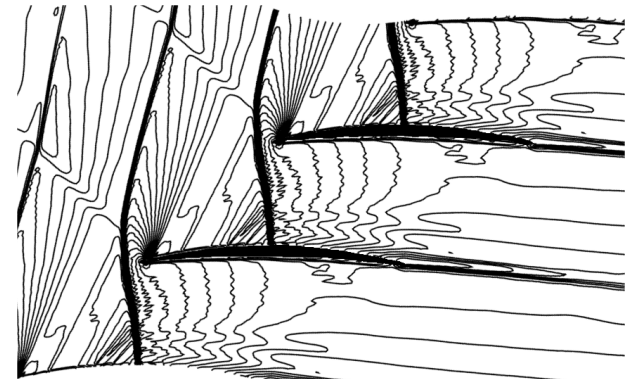
DASFLOW Program – Egyenletek – Validáció (súrlódásmentes)



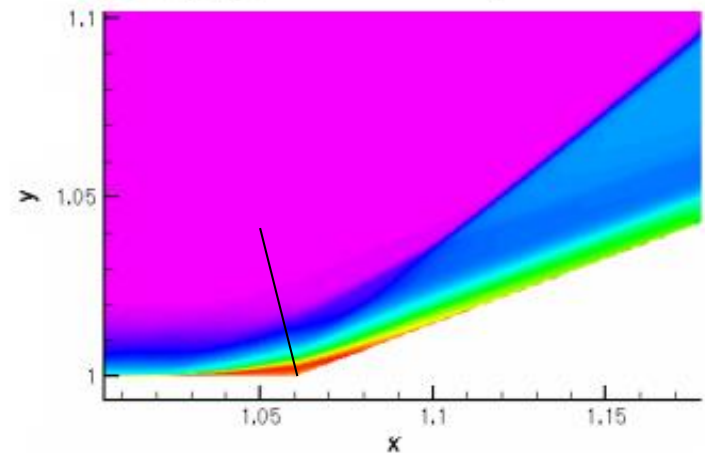
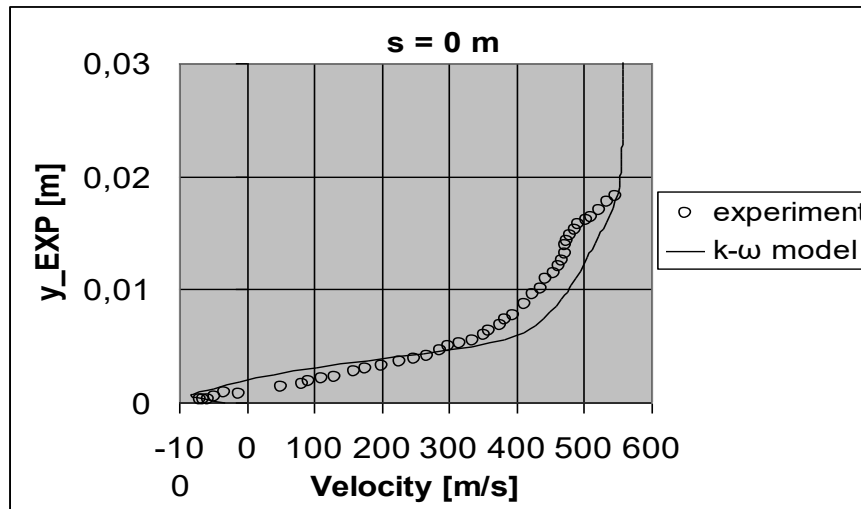
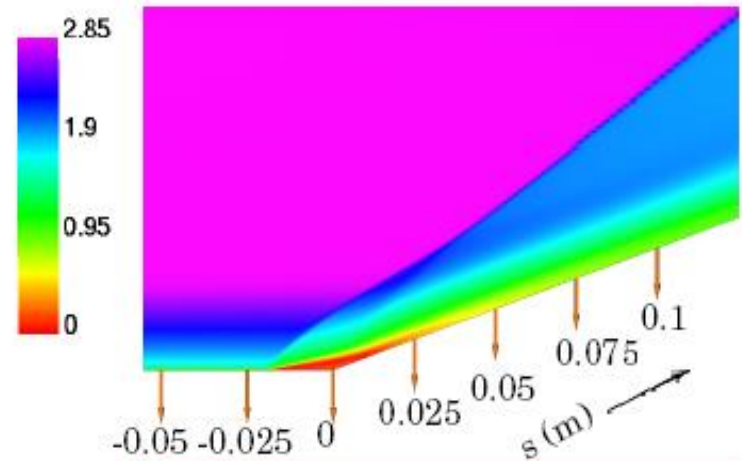
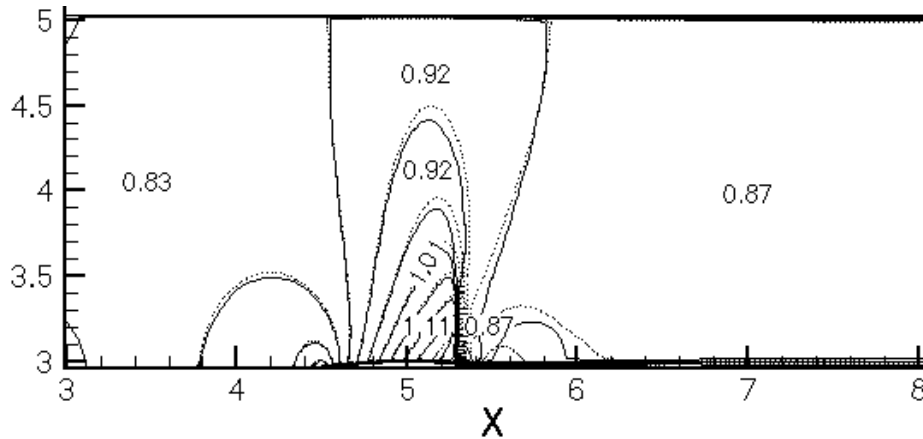
$M_1=1.1$



$M_1=1.1$

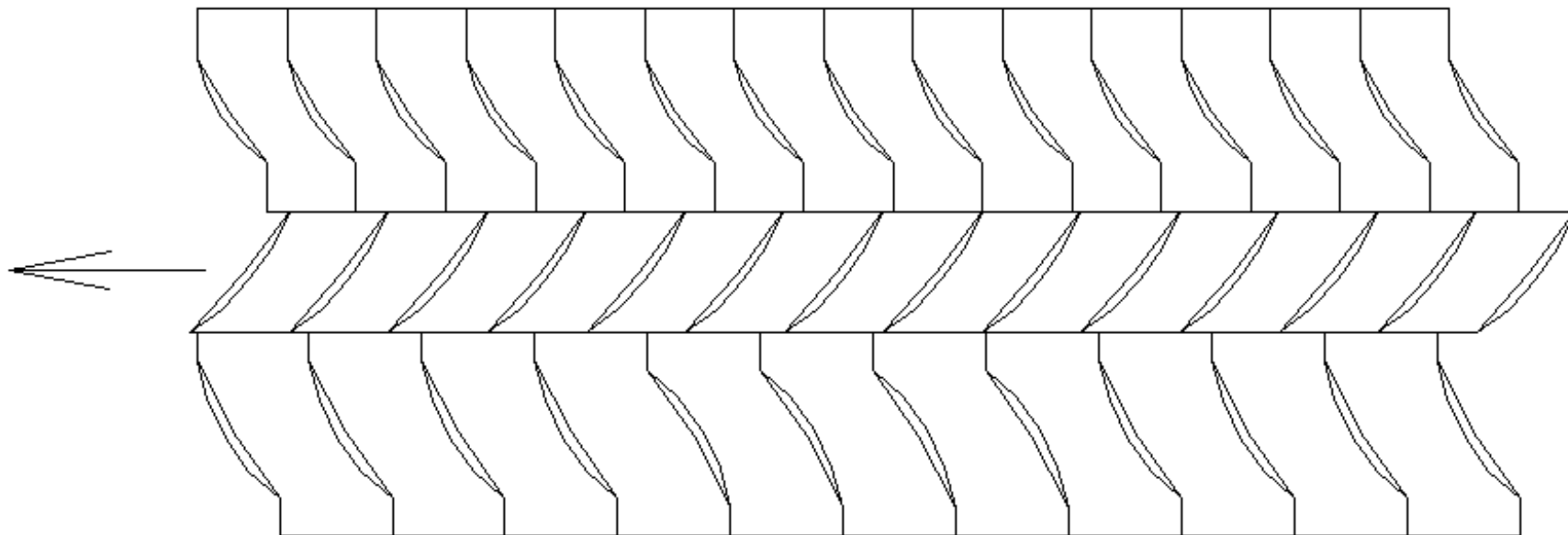


DASFLOW Program – Egyenletek – Validáció (súrlódásos)



DASFLOW Program - Lapátrács numerikus áramlástanai vizsgálata

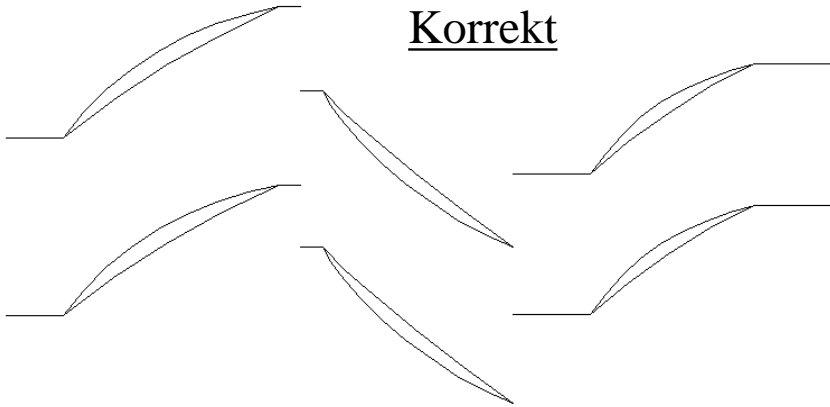
Probléma leírása



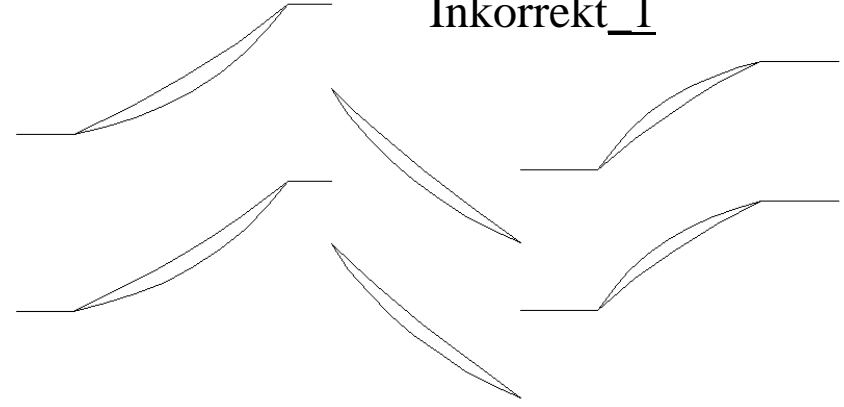
DASFLOW Program - Lapátrács numerikus áramlástanai vizsgálata

Probléma leírása

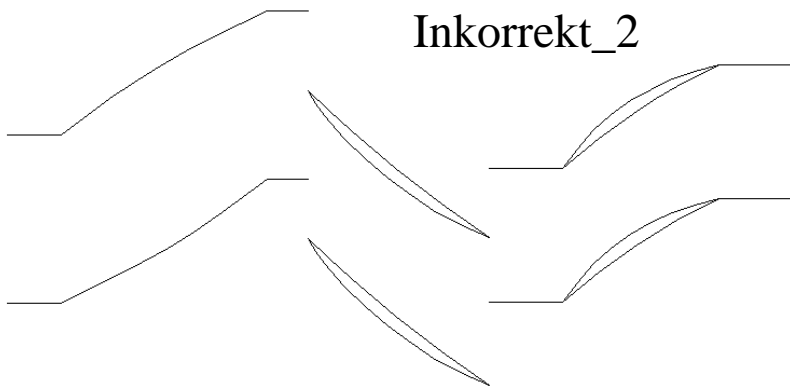
Korrekt



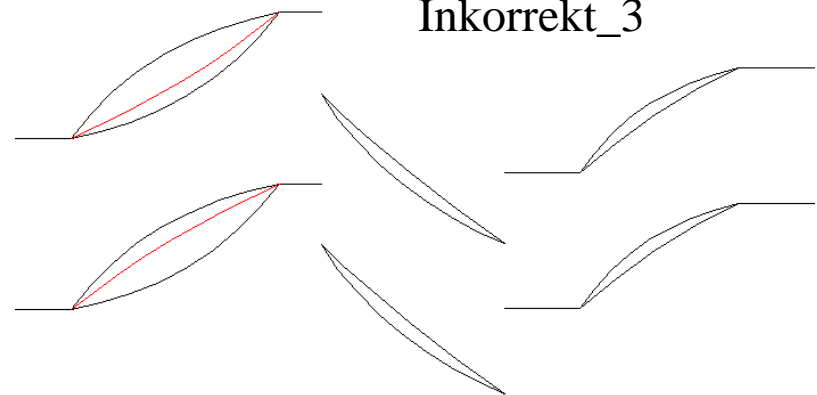
Inkorrekt_1



Inkorrekt_2

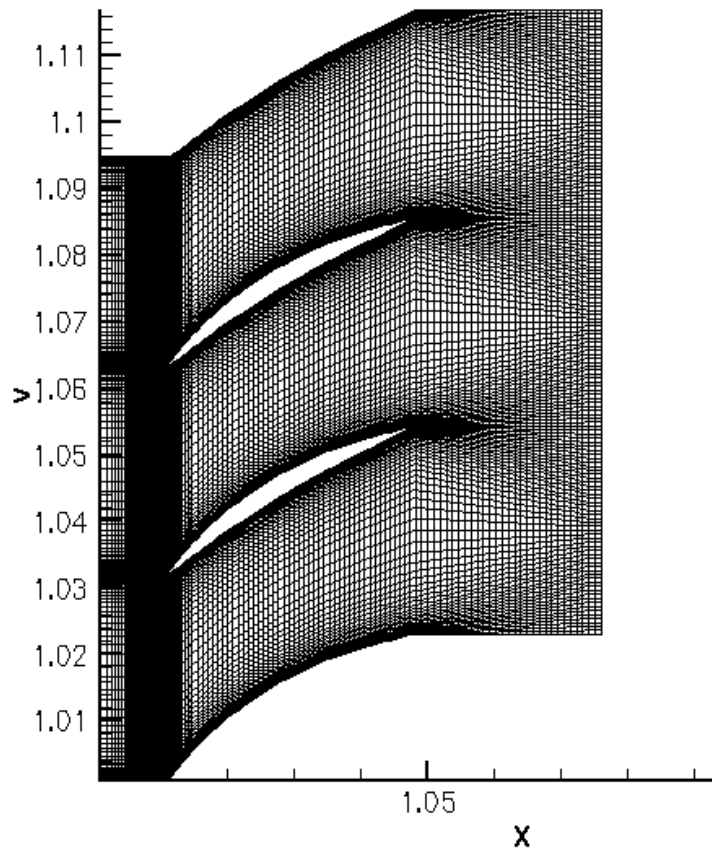


Inkorrekt_3

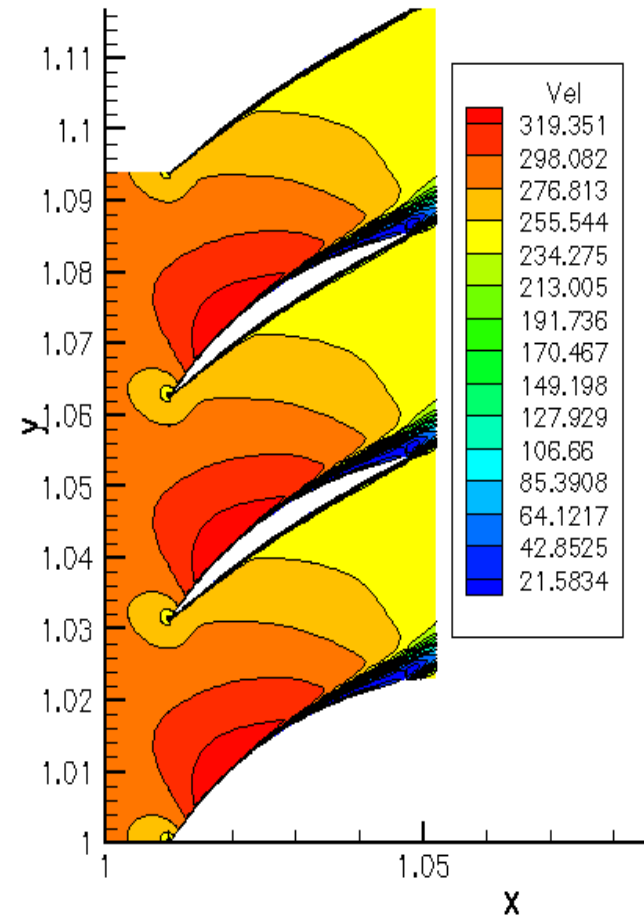


DASFLOW Program - Lapátrács numerikus áramlástanai vizsgálata

Korrekt



$p_{in}^{to} = 796127 \text{ Pa}$, $T_{in}^{to} = 530.7 \text{ K}$
 $\alpha_{in} = 43^\circ$

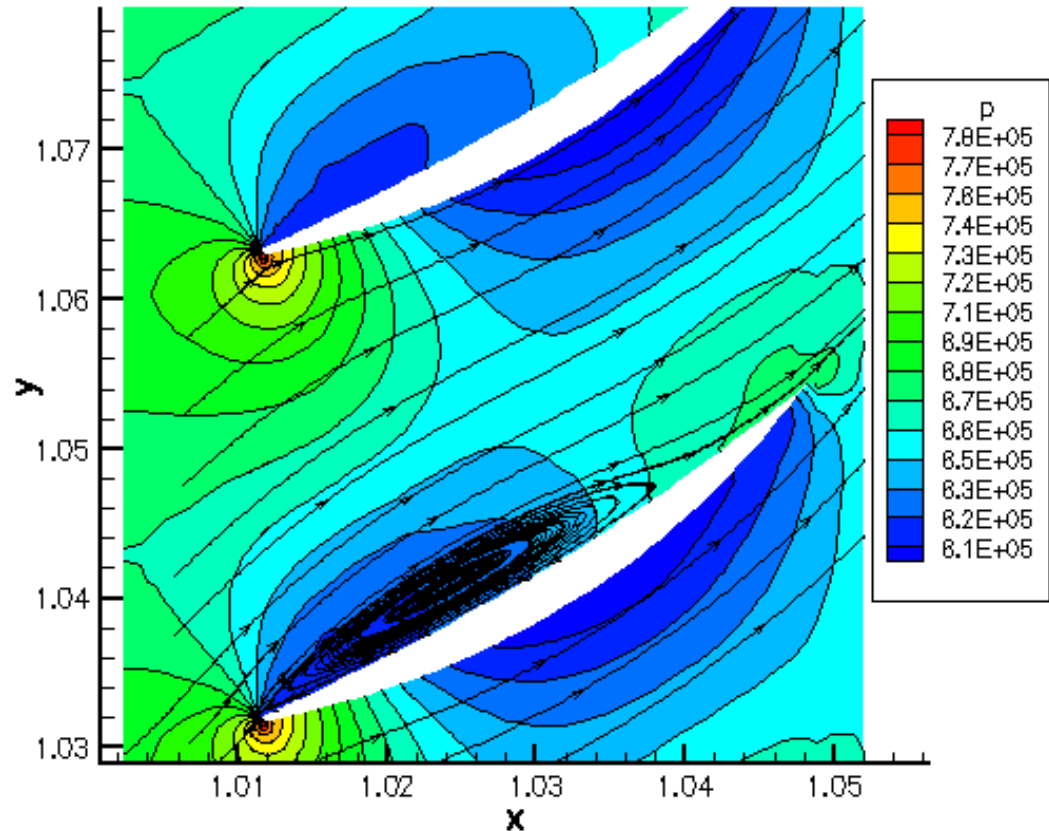
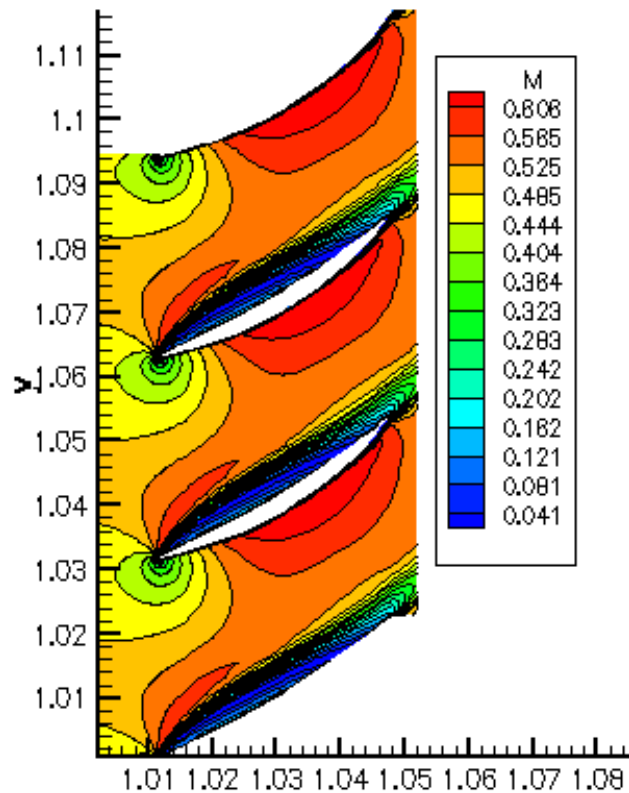


$p_{out} = 654600. \text{ Pa}$, $p_{out_ave}^{to} = 780525. \text{ Pa}$,
 $T_{out_ave}^{to} = 530.5 \text{ K}$, $\alpha_{r_out_ave} = -43.7^\circ$



DASFLOW Program - Lapátrács numerikus áramlástanai vizsgálata

Inkorrekt_1

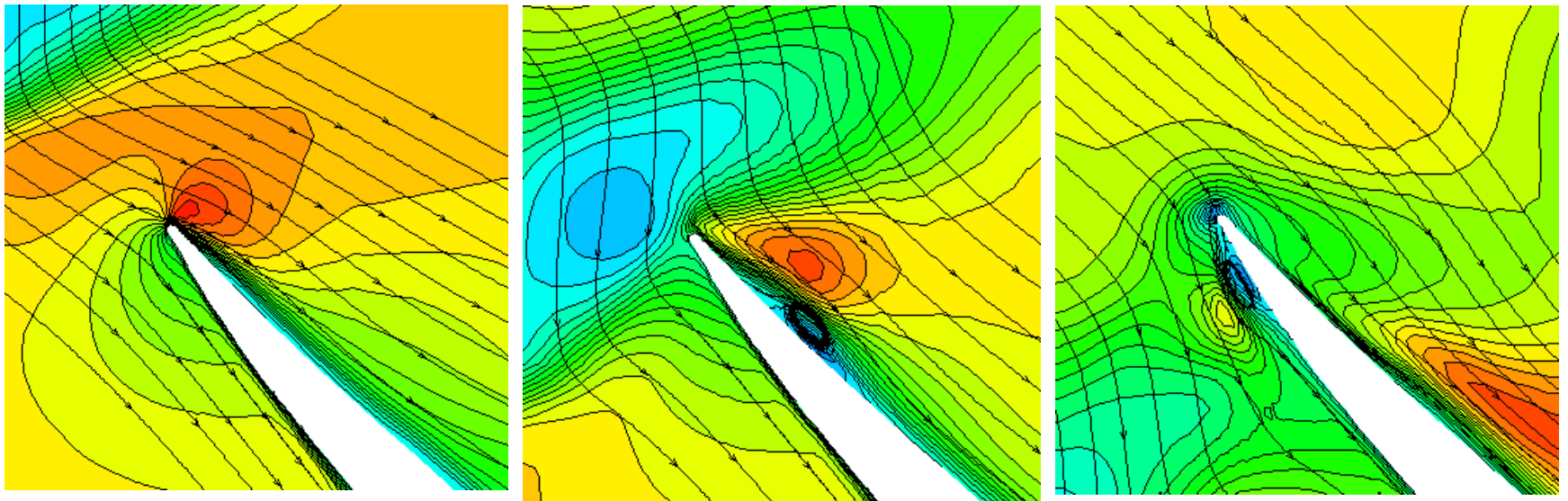


$p_{in}^{to} = 796127 \text{ Pa}$, $T_{in}^{to} = 530.7 \text{ K}$
 $\alpha_{in} = 43^\circ$

$p_{out} = 654600. \text{ Pa}$, $p_{out_ave}^{to} = 780525. \text{ Pa}$,
 $T_{out_ave}^{to} = 530.5 \text{ K}$, $\alpha_{r_out_ave} = -43.7^\circ$

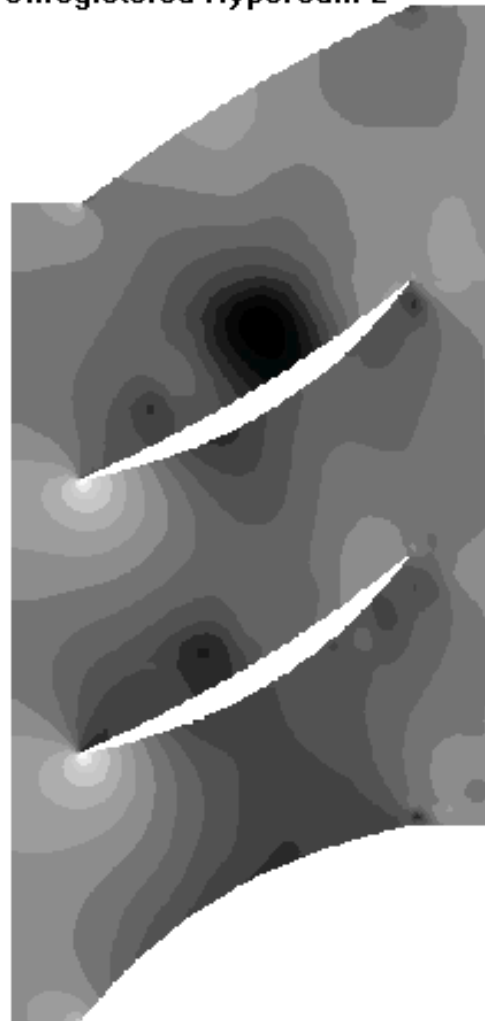
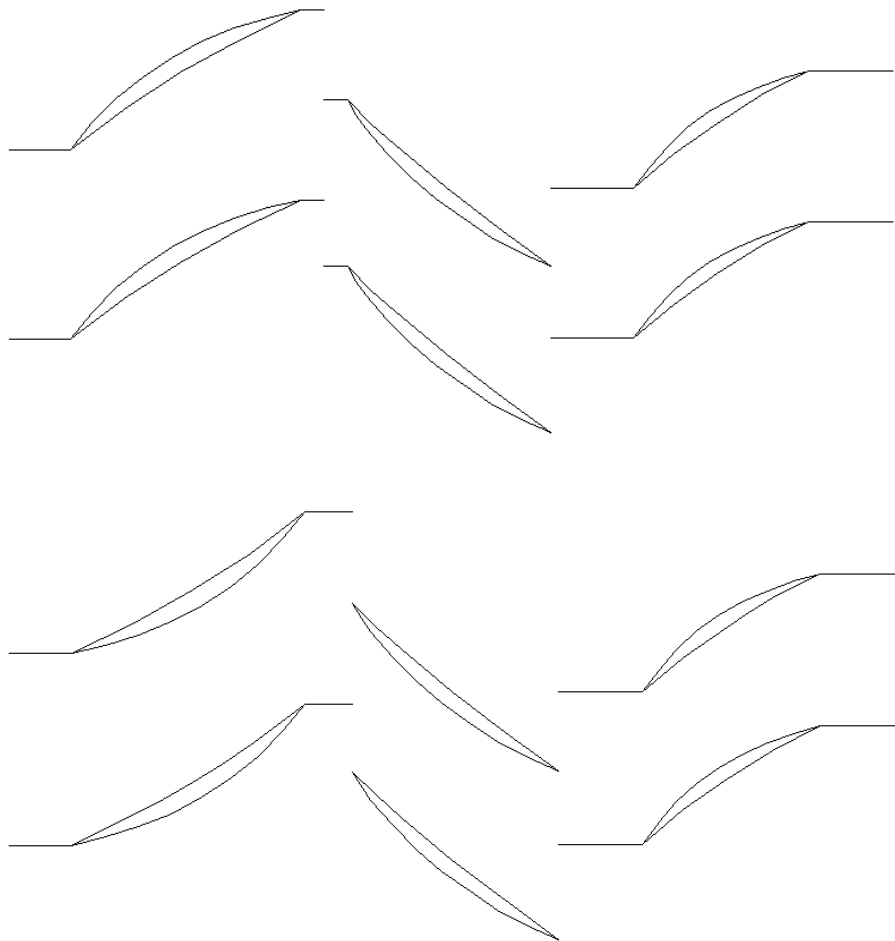


DASFLOW Program - Lapátrács numerikus áramlástanai vizsgálata



DASFLOW Program - Lapátrács numerikus áramlástanai vizsgálata

Unregistered HyperCam 2



Tüzelőanyag sugárszivattyú direkt numerikus optimalizálása - Ipari alkalmazás

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

$$\frac{\partial U^i}{\partial t} + \frac{\partial F^i}{\partial x} + \frac{\partial G^i}{\partial y} = 0$$
$$U^i = (0, u, v)^T \quad F^i = (u, u^2 + \frac{P}{\rho}, vu)^T$$
$$G^i = (v, uv, v^2 + \frac{P}{\rho})^T$$

Chorin módszere szerint:

$$\frac{\partial U^i}{\partial t} + \frac{\partial F^i}{\partial x} + \frac{\partial G^i}{\partial y} = 0$$
$$U^i = (P, u, v)^T \quad F^i = (u\beta^2, u^2 + P, vu)^T$$
$$G^i = (v\beta^2, uv, v^2 + P)^T$$

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



Tüzelőanyag sugárszivattyú direkt numerikus optimalizálása - Ipari alkalmazás

• Integrál egyenletek

$$\left\{ \begin{array}{l} \frac{\partial}{\partial t} \iint_{\Omega} U d\Omega + \int_{\Gamma} H_n d\Gamma = 0 \\ H_n = \vec{H}\vec{n} = \begin{pmatrix} \beta^2 V_n \\ uV_n + Pn_x \\ vV_n + Pn_y \end{pmatrix} \end{array} \right.$$

$$\frac{\partial}{\partial t} U_j = -\frac{1}{\Omega_j} \sum_{k=1}^{N_f} [H_n]_{j,k} \Gamma_{j,k} + \frac{1}{\Omega_j} D$$

$$U^0 = U^n$$

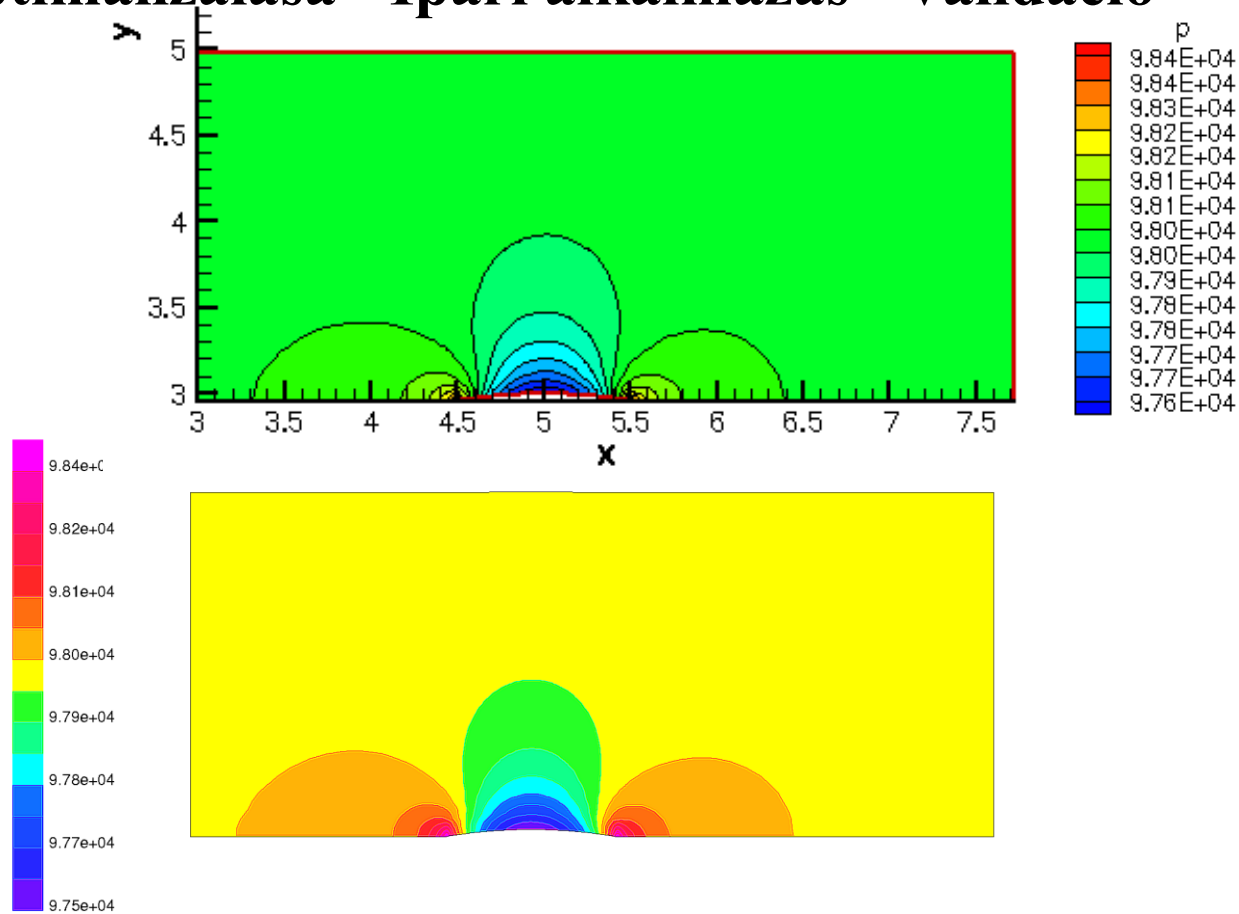
$$U^k = U^0 + \alpha_k \Delta t \mathcal{R}(U^{k-1}) \quad k = 1, m$$

$$U^{n+1} = U^m$$

• Runge-Kutta módszer



Tüzelőanyag sugárszivattyú direkt numerikus optimalizálása - Ipari alkalmazás - Validáció

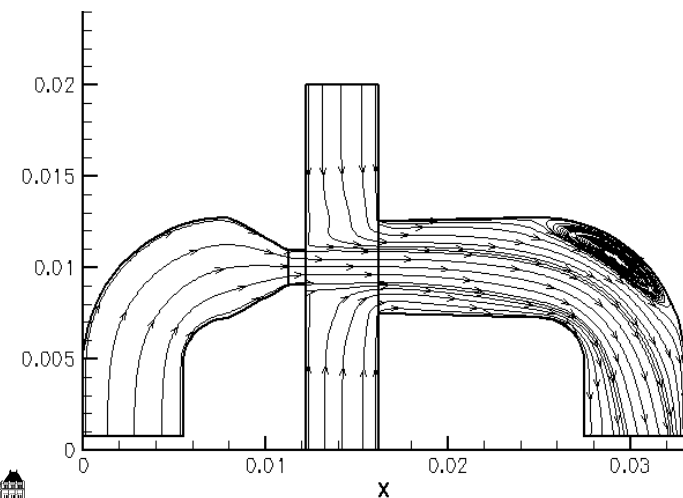
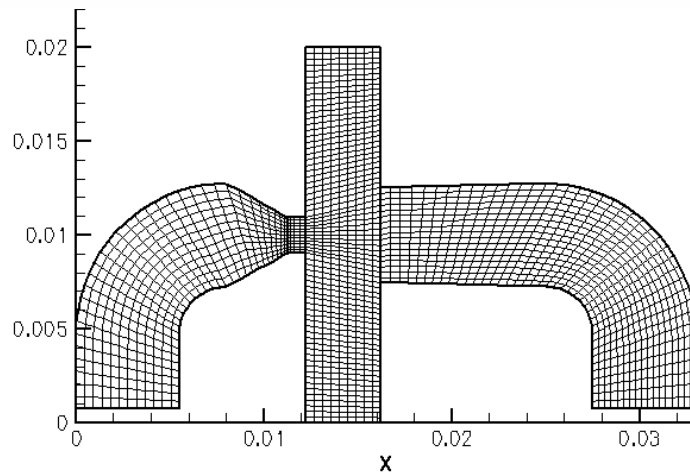
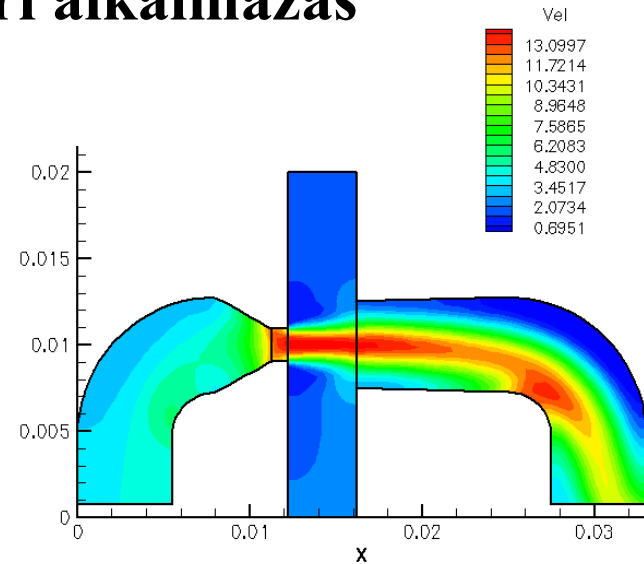
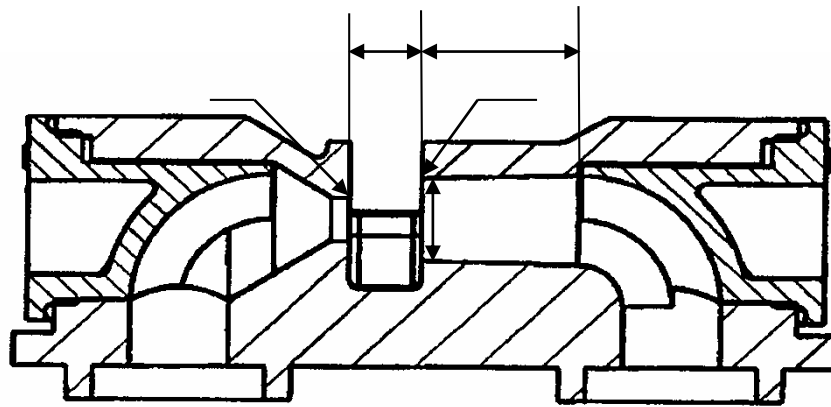


Contours of Static Pressure (pascal)

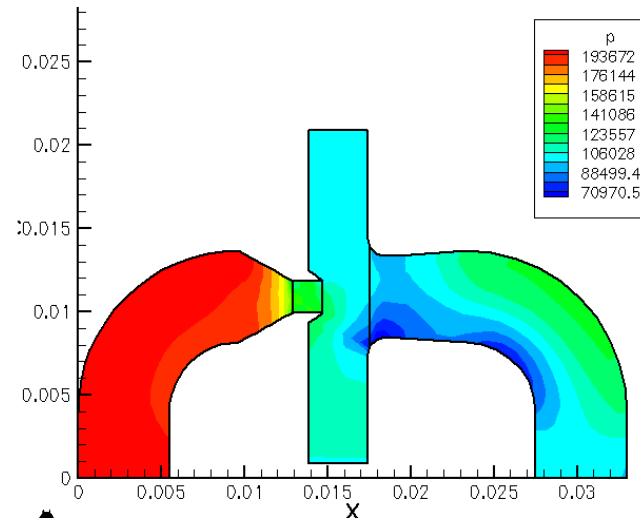
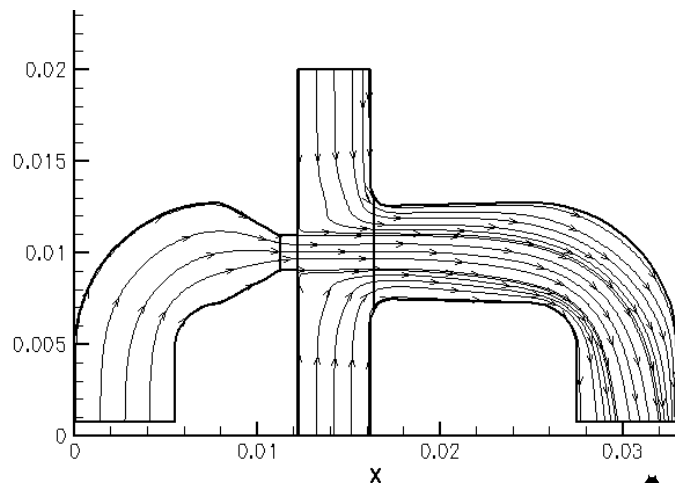
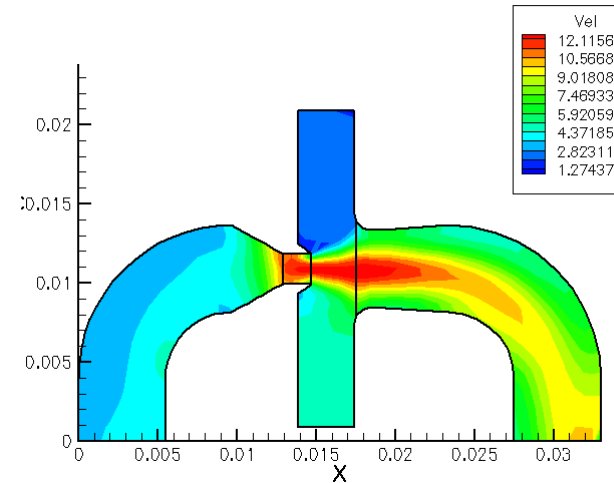
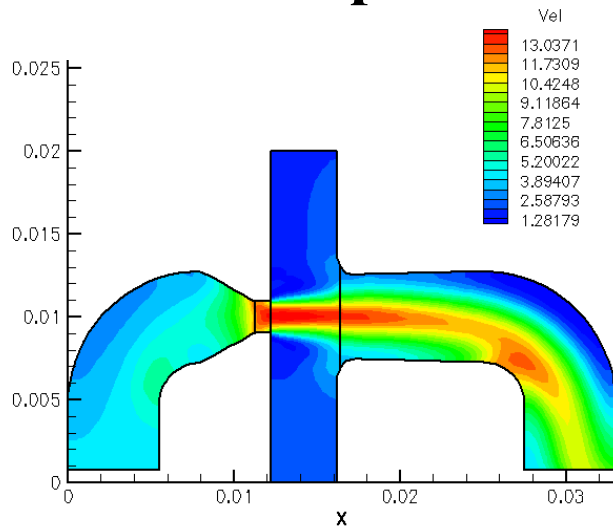
Sep 13, 2003
FLUENT 5.2 (2d, segregated)



Tüzelőanyag sugárszivattyú direkt numerikus optimalizálása - Ipari alkalmazás



Tüzelőanyag sugárszivattyú direkt numerikus optimalizálása - Ipari alkalmazás



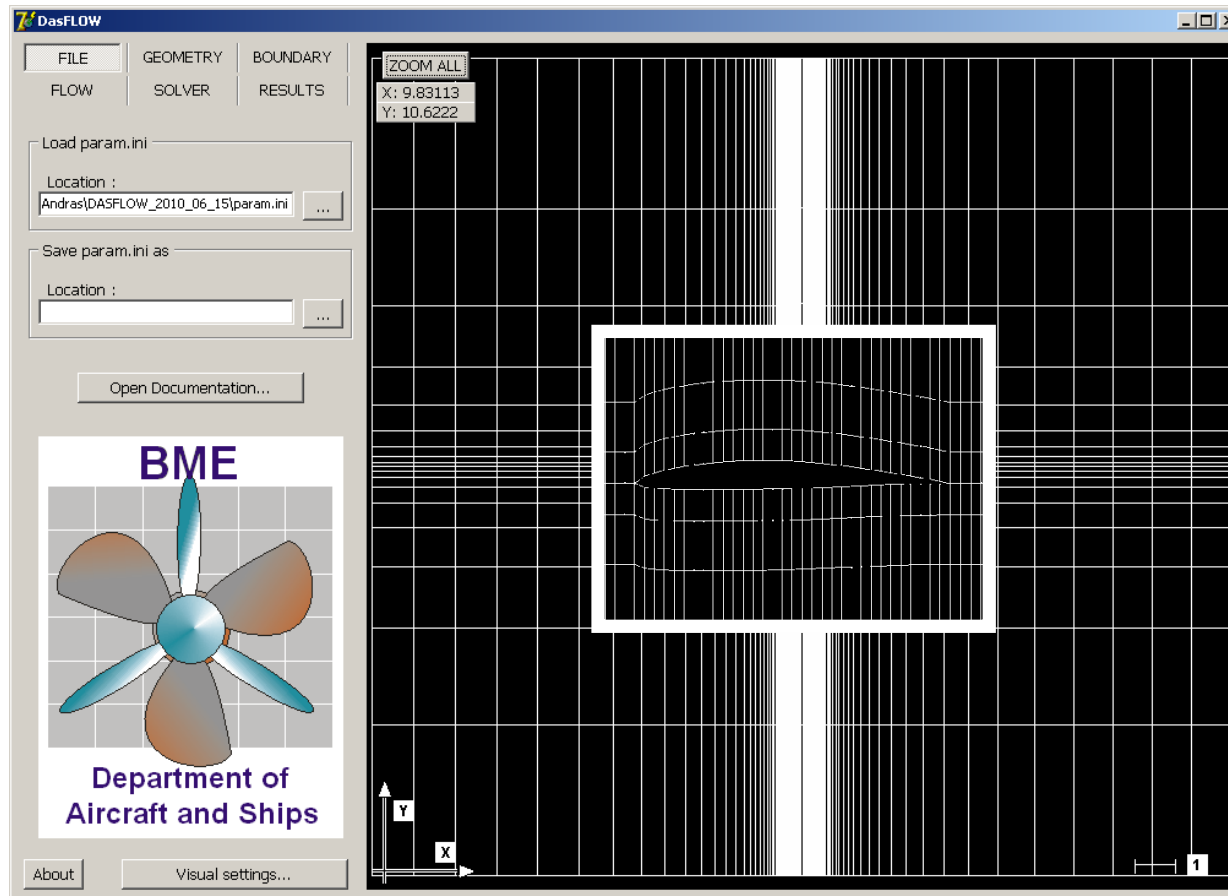
Tüzelőanyag sugárszivattyú direkt numerikus optimalizálása - Ipari alkalmazás

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
<i>Sz.k.</i>	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

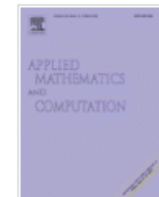
More options... ▼



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

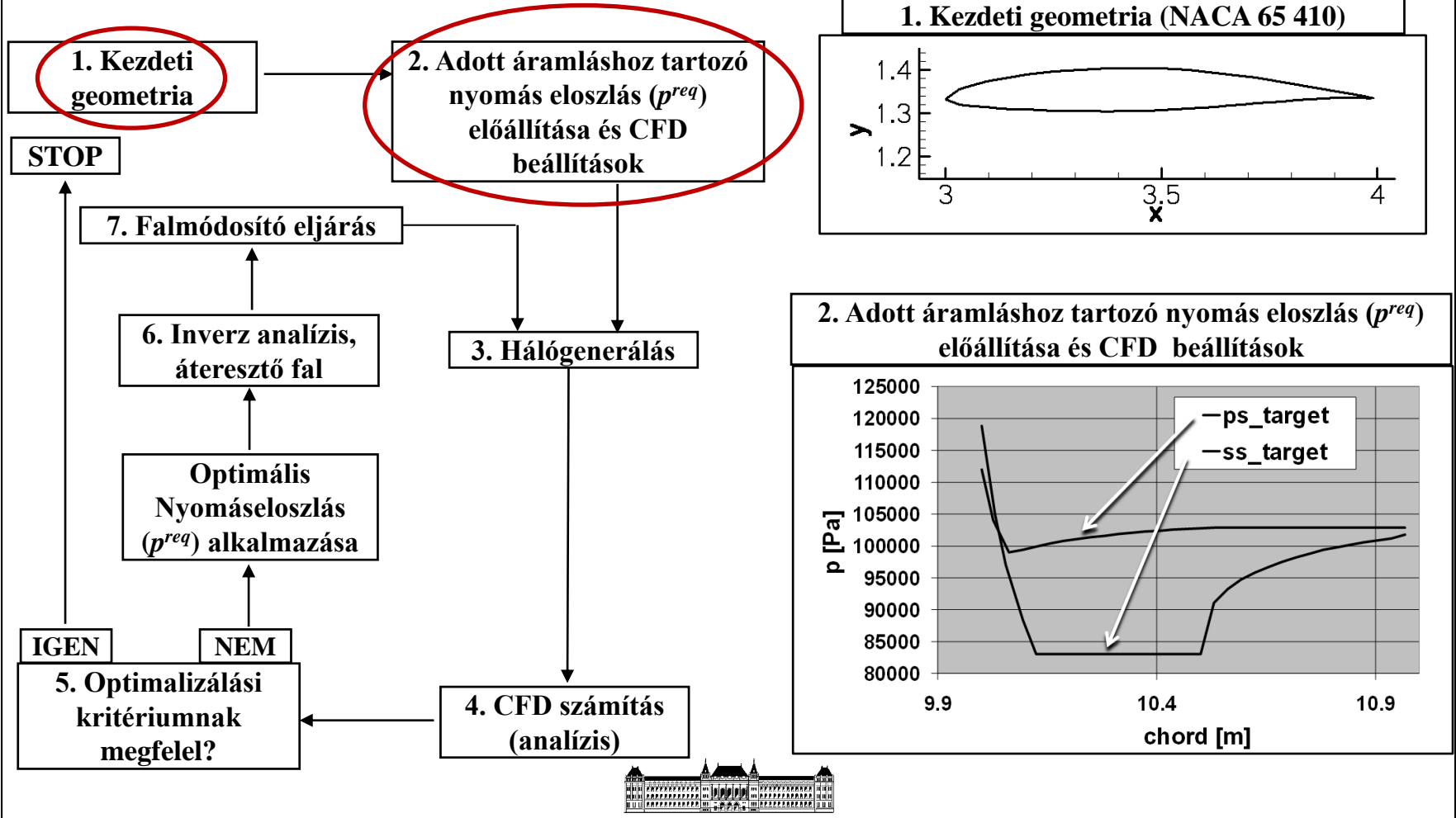
^b Knorr-Bremse R&D Center Budapest, Major u. 69, H-1119 Budapest, Hungary



Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

Szárnyprofilra, NACA 65-4101

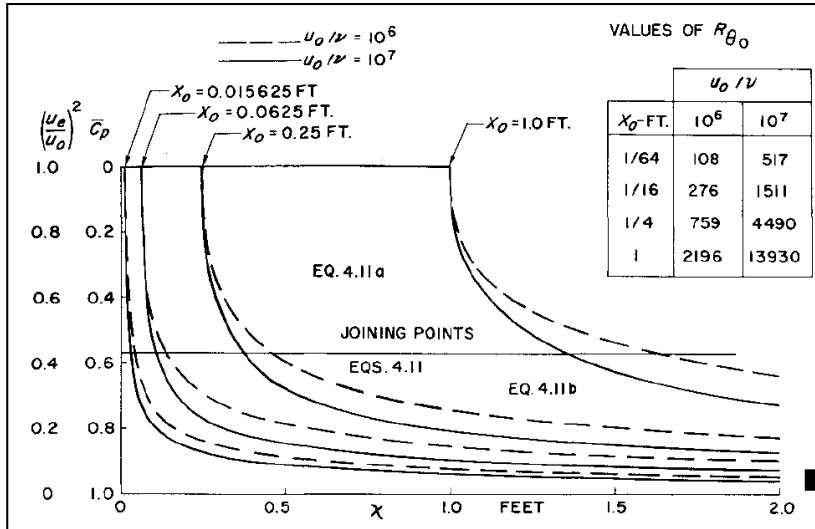
A számítás folyamata:



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

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

Pressure coefficient:

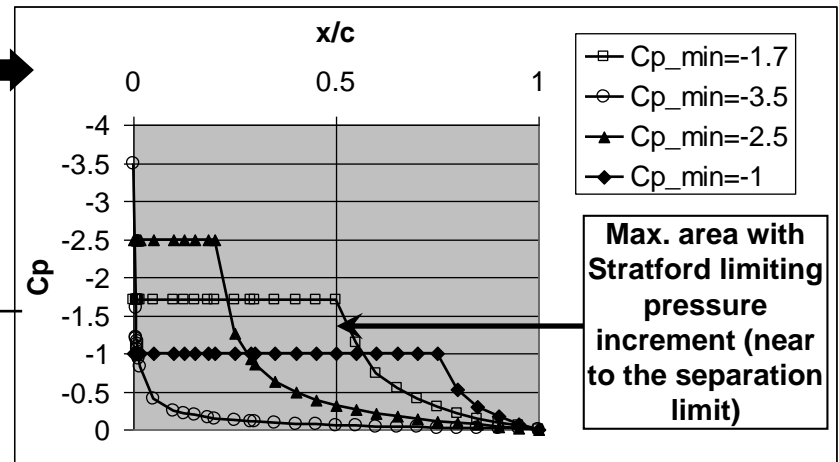
$$C_p = \frac{p - p_\infty}{1/2 \rho_\infty V_\infty^2} \equiv \frac{p - p_\infty}{0.7 p_\infty M_\infty^2}$$

Canonical pressure distribution:

$$\bar{C}_p = \frac{p - p_o}{1/2 \rho_o V_o^2}$$

o : Location of the start of the positive pressure gradient

p^{req}



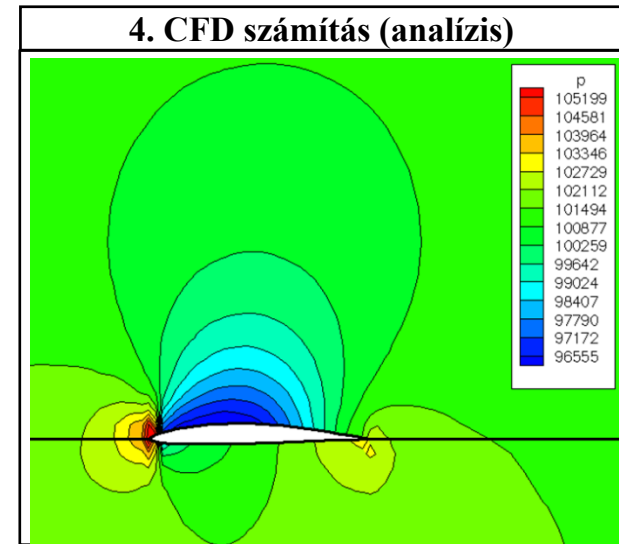
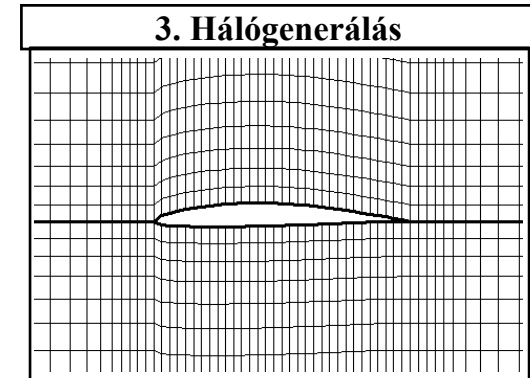
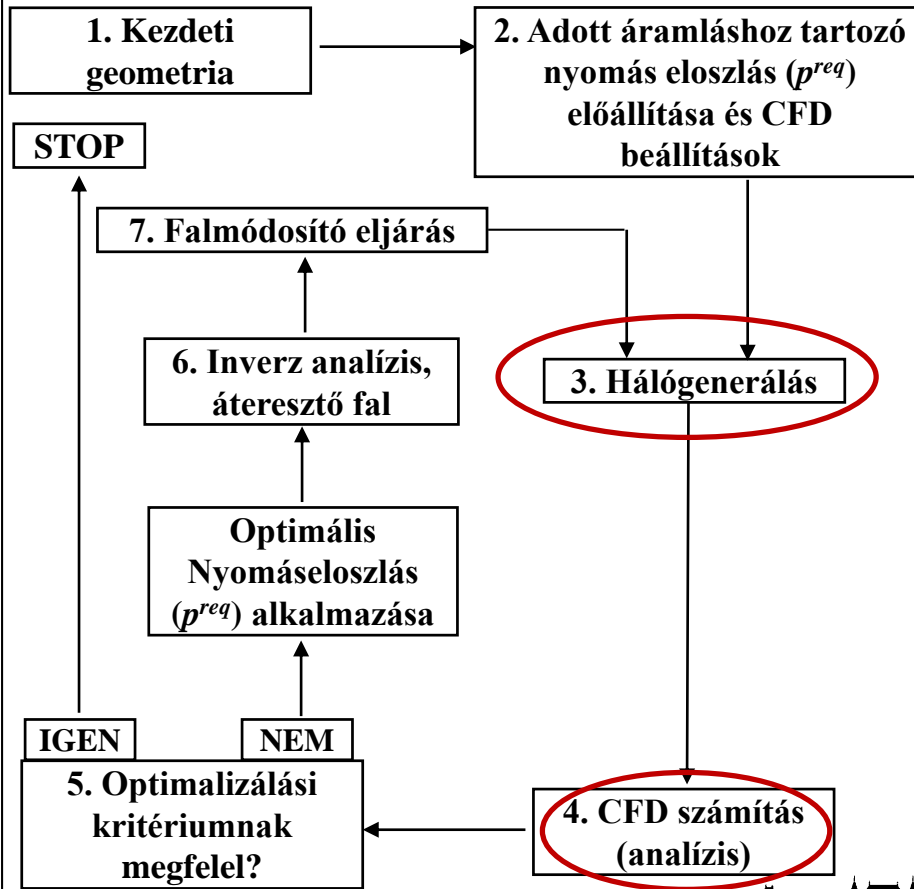
p^{req} is the output of the optimization



Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

Szárnyprofilra, NACA 65-4101

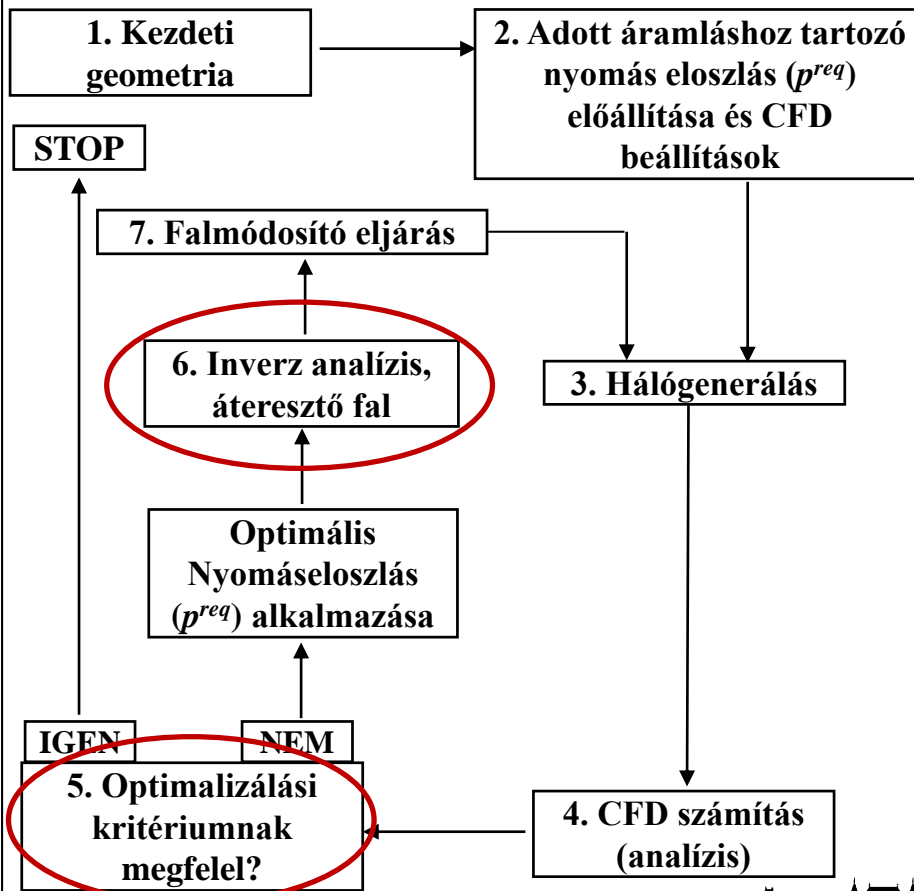
A számítás folyamata:



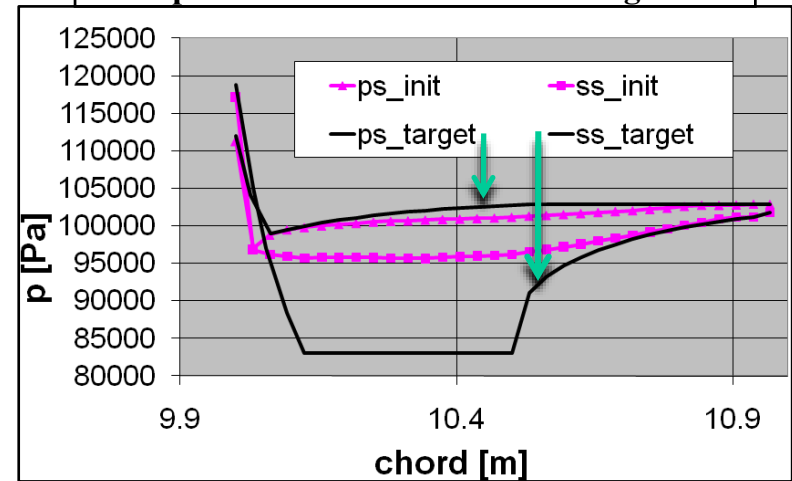
Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

Szárnyprofilra, NACA 65-4101

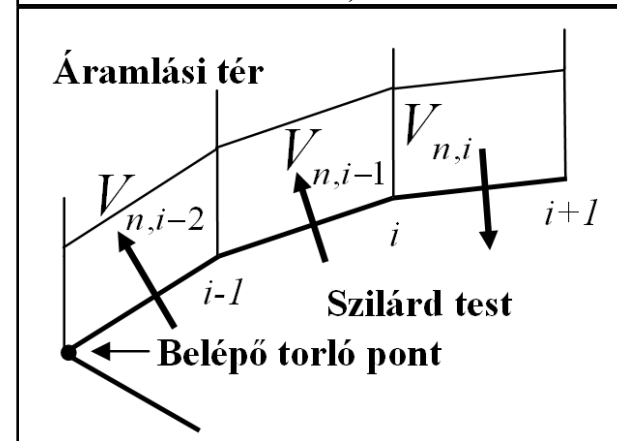
A számítás folyamata:



5. Optimalizálási kritériumnak megfelel?



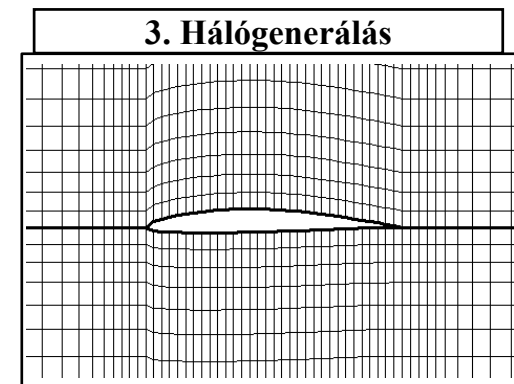
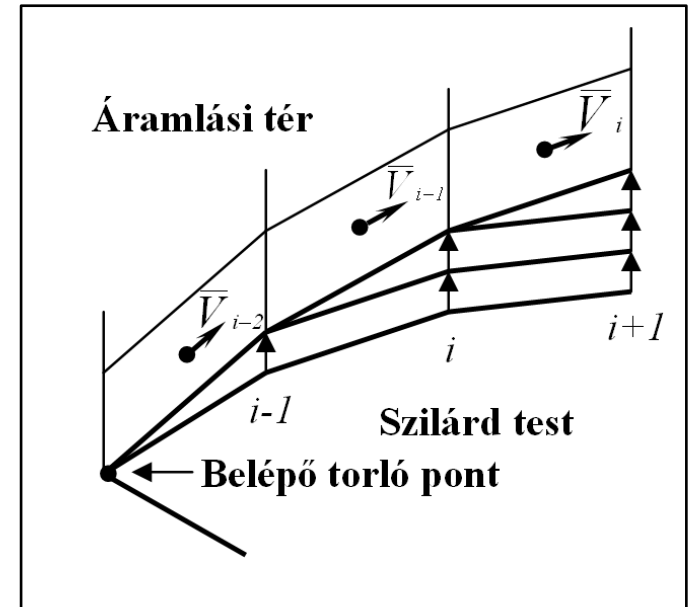
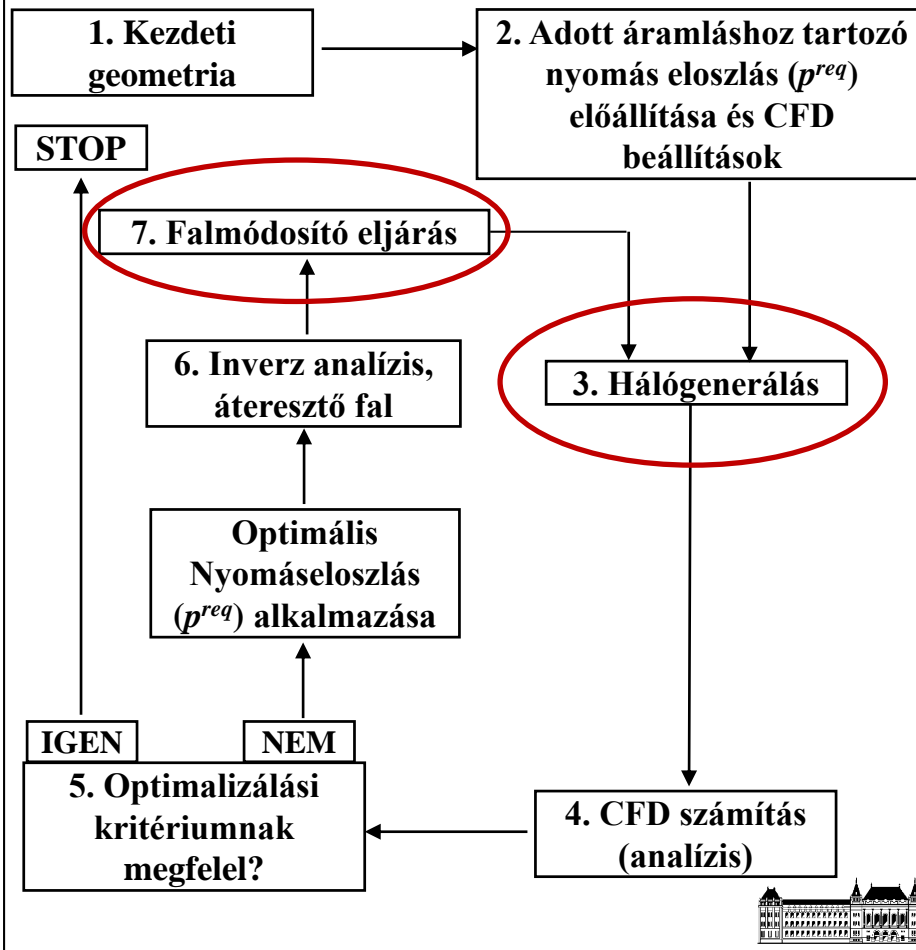
6. Inverz analízis, áteresztő fal



Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

Szárnyprofilra, NACA 65-4101

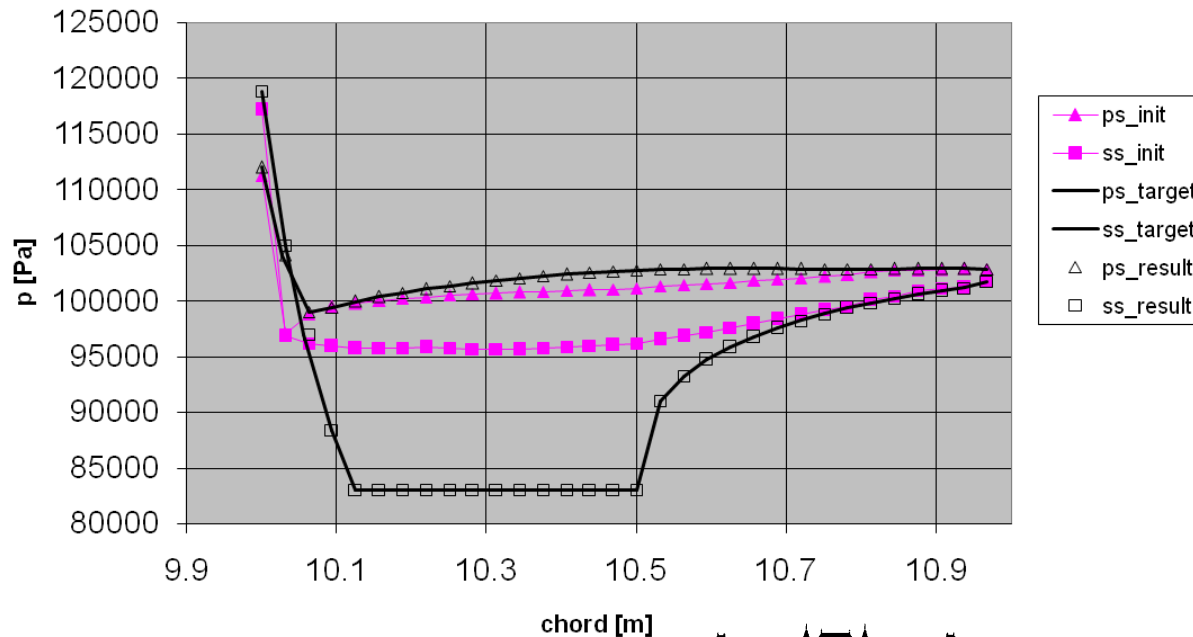
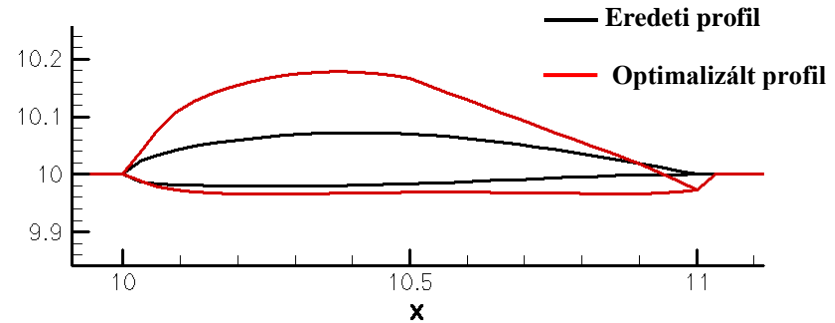
A számítás folyamata:



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

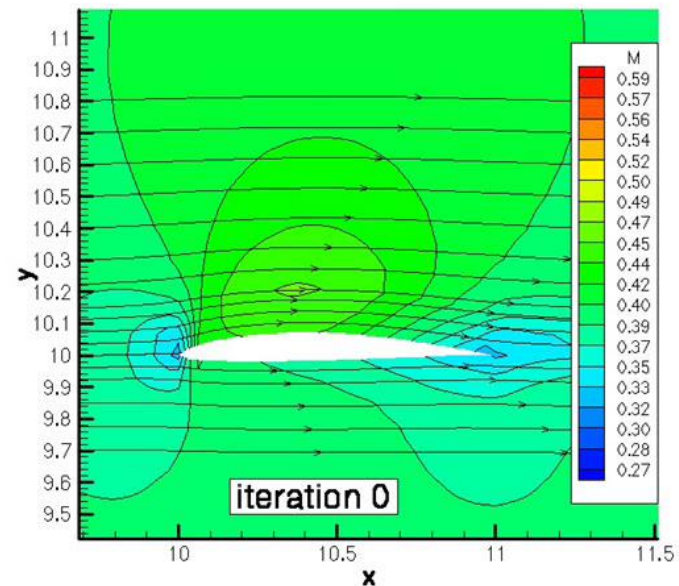
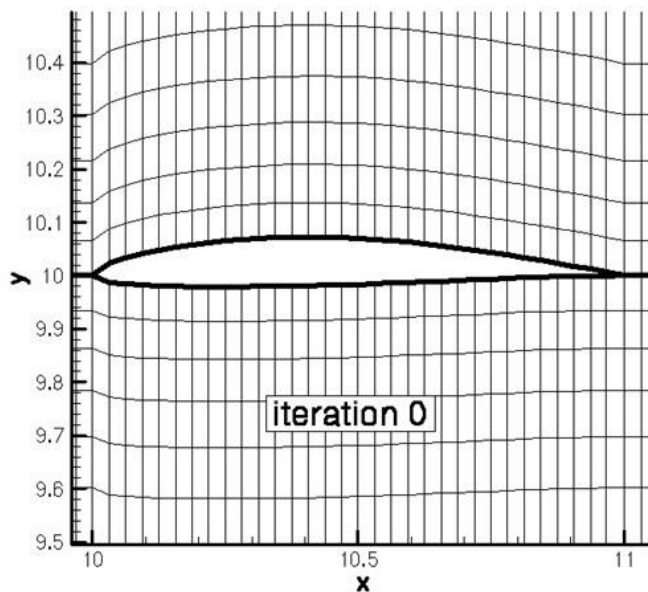
sűrűség NKÉ: $1e-5.6$



Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

Szárnyprofilra, NACA 65-4101

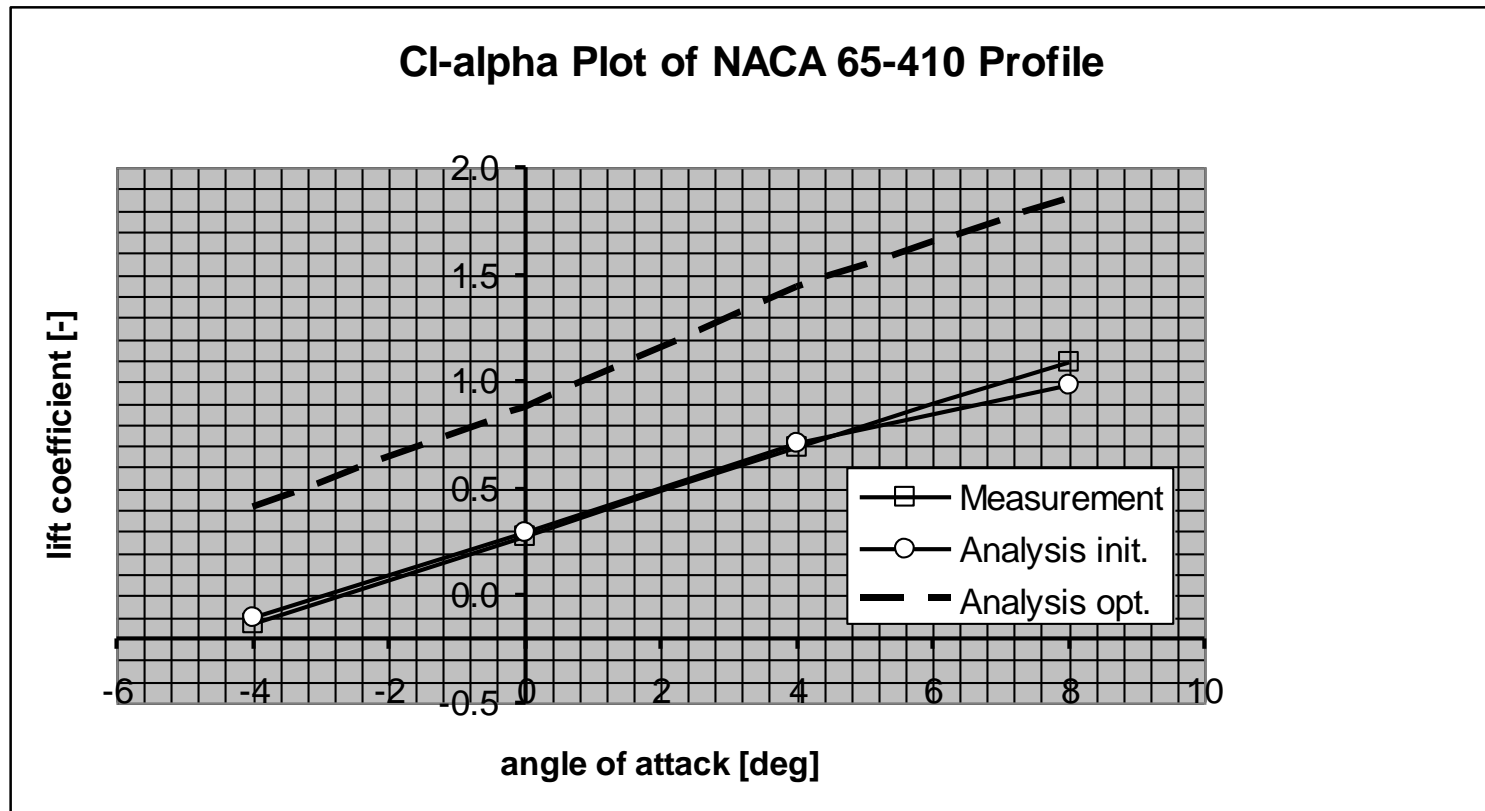
Eredmények 10 inverz iterációt követően:



Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

Szárnyprofilra, NACA 65-4101

Eredmények a kezdeti profil esetén és 10 inverz iterációt követően:

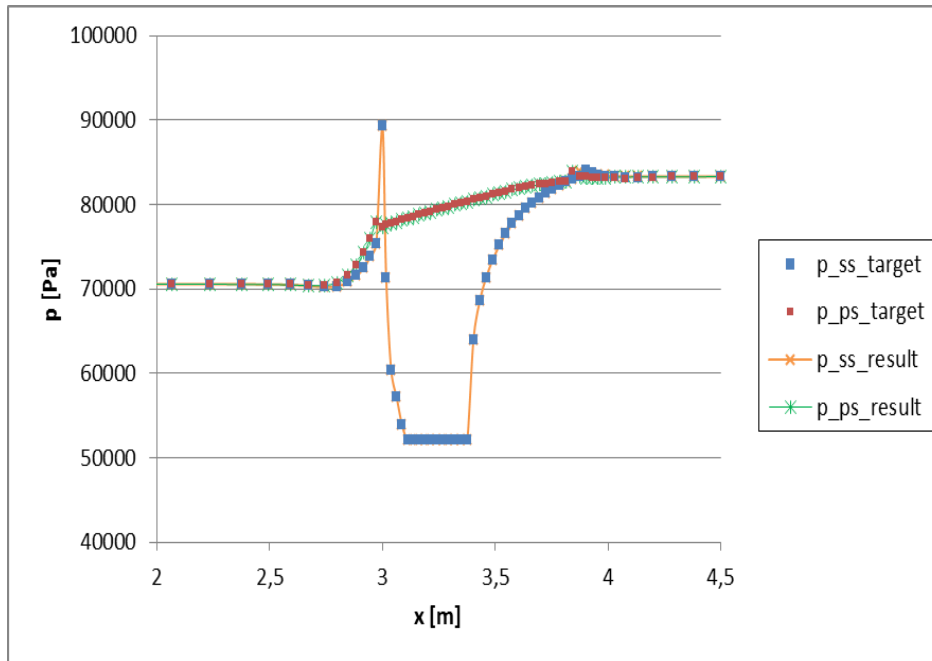


Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

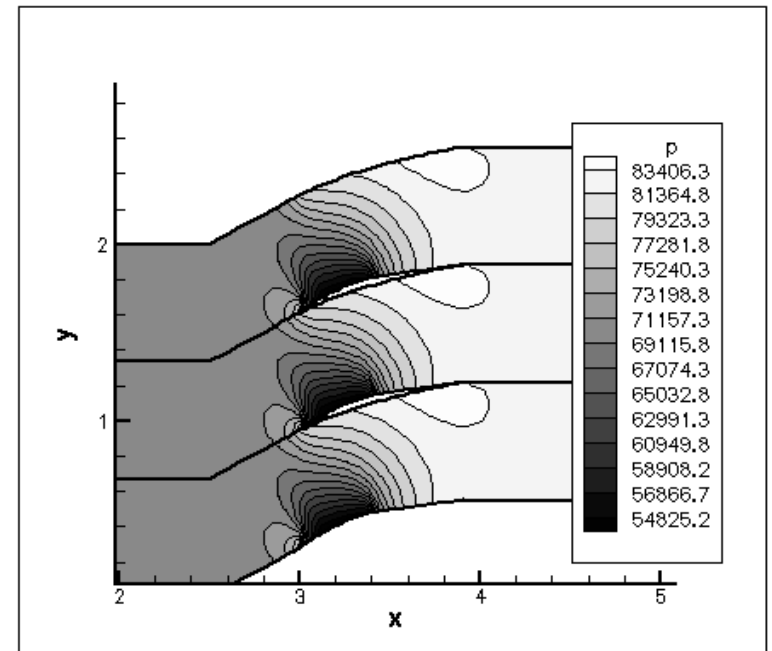
Fokozati kompresszió viszony nagyságát befolyásoló tényezők: $\pi_{ks} = f(U(n), C_a, \beta_1 - \beta_2)$

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 nyomáeloszlás



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

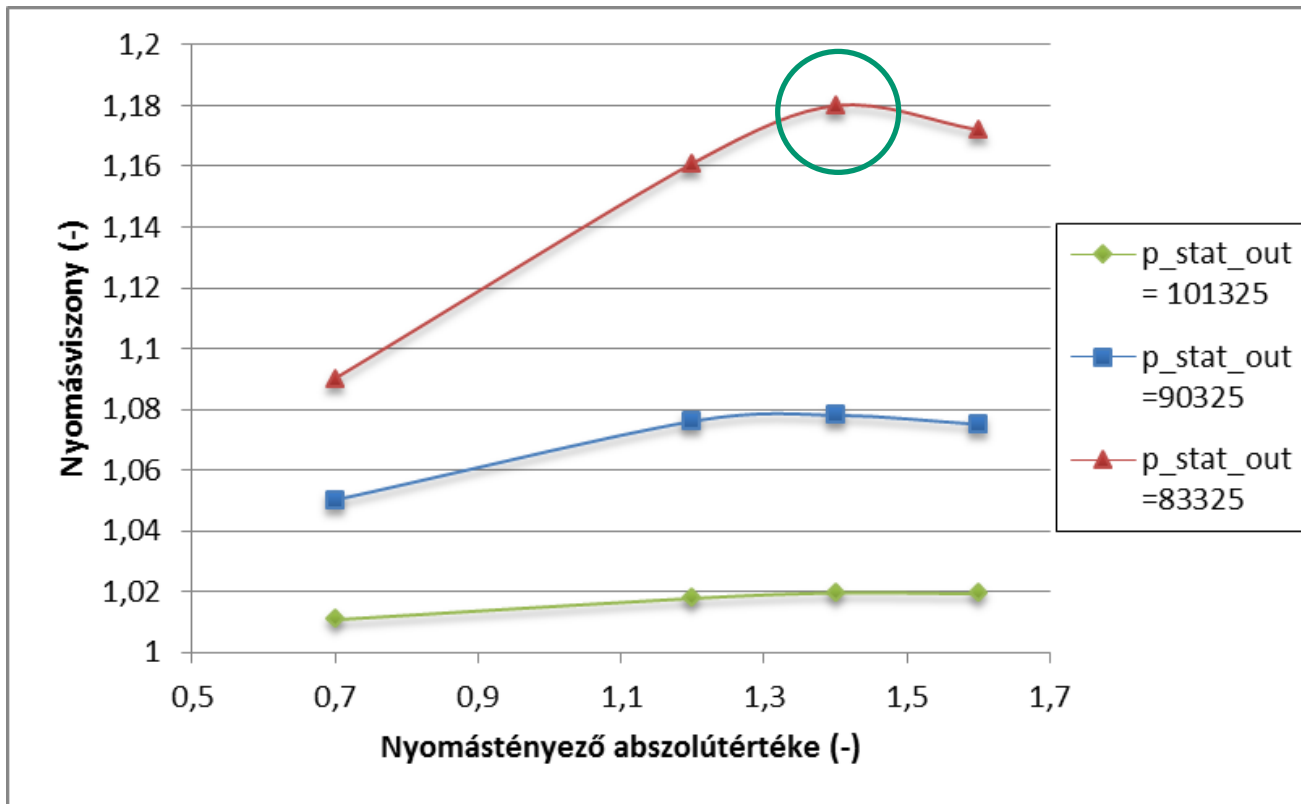


Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

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

Eredmények lapátrácsra 10 inverz iterációt követően:

$$\pi_{ks,static} = 1,18$$



Mach-szám:
← 0,3
← 0,5
← 0,62

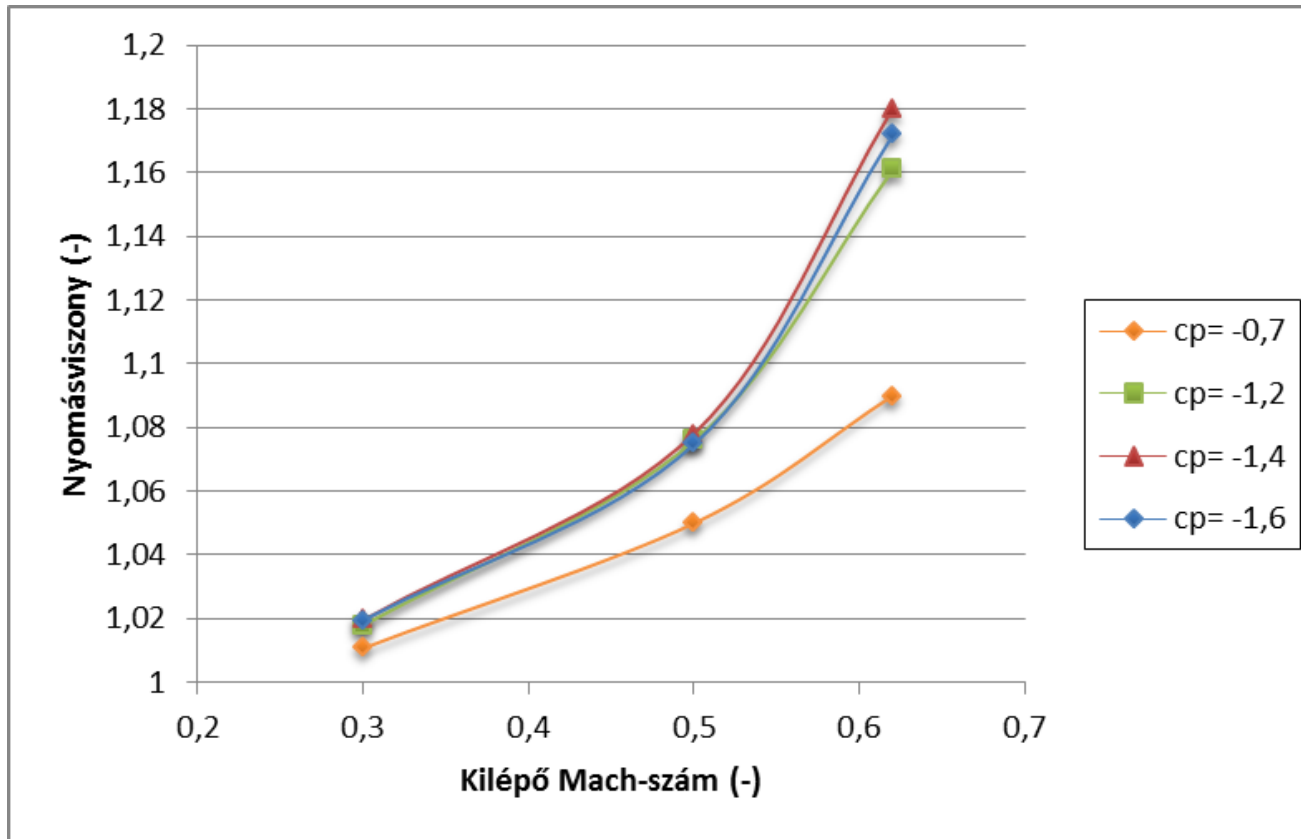
$$\pi = \frac{P_{stat,out}}{P_{stat,in}} = f(p_{stat,out}, C_p)$$



Optimalizáció; inverz tervezőeszköz kidolgozása és alkalmazása

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

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



$$\pi = \frac{P_{stat,out}}{P_{stat,in}} = f(p_{stat,out}, C_p)$$



Mellékletek

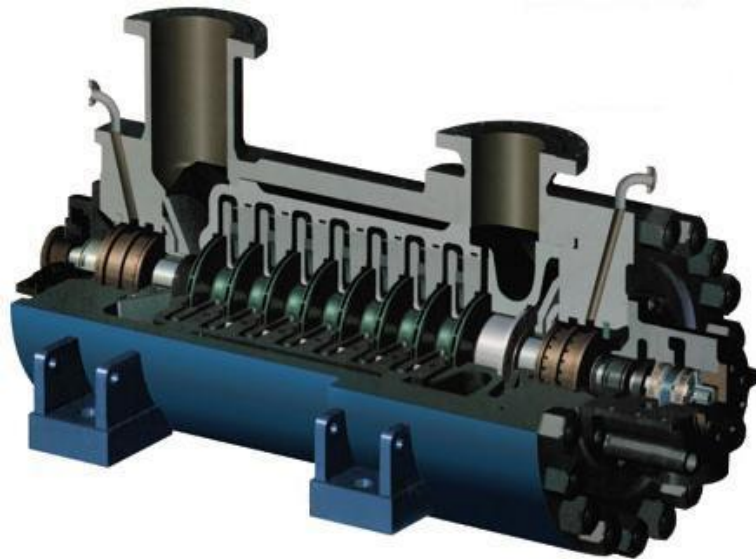
CFD és Inverz

tervezőeszköz ipari

alkalmazásai

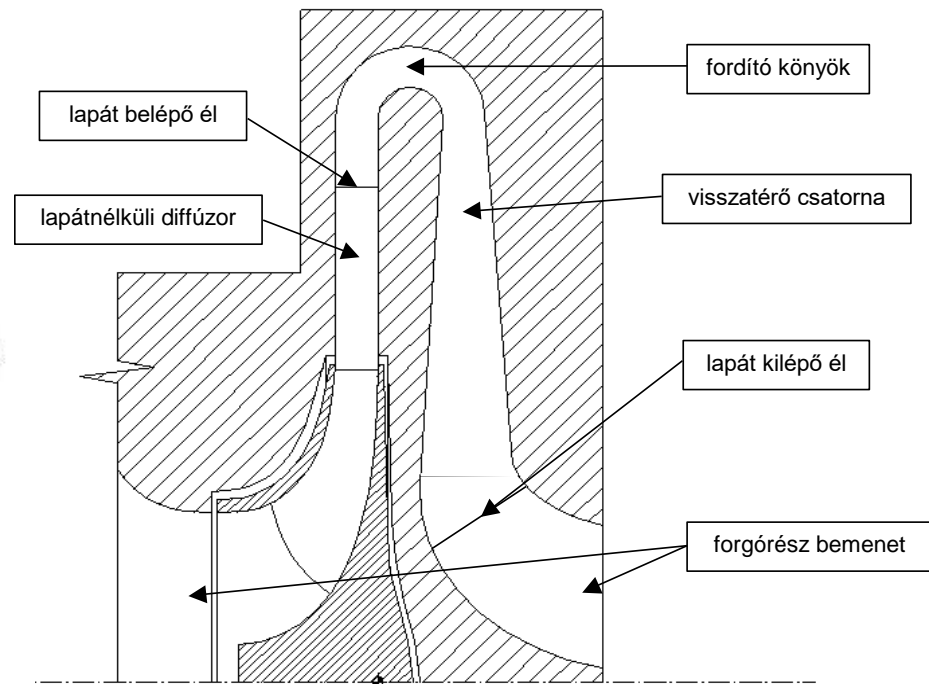


Többfokozatú centrifugál kompresszor összekötő- csatorna lapátozásának tervezése – Ipari alkalmazás

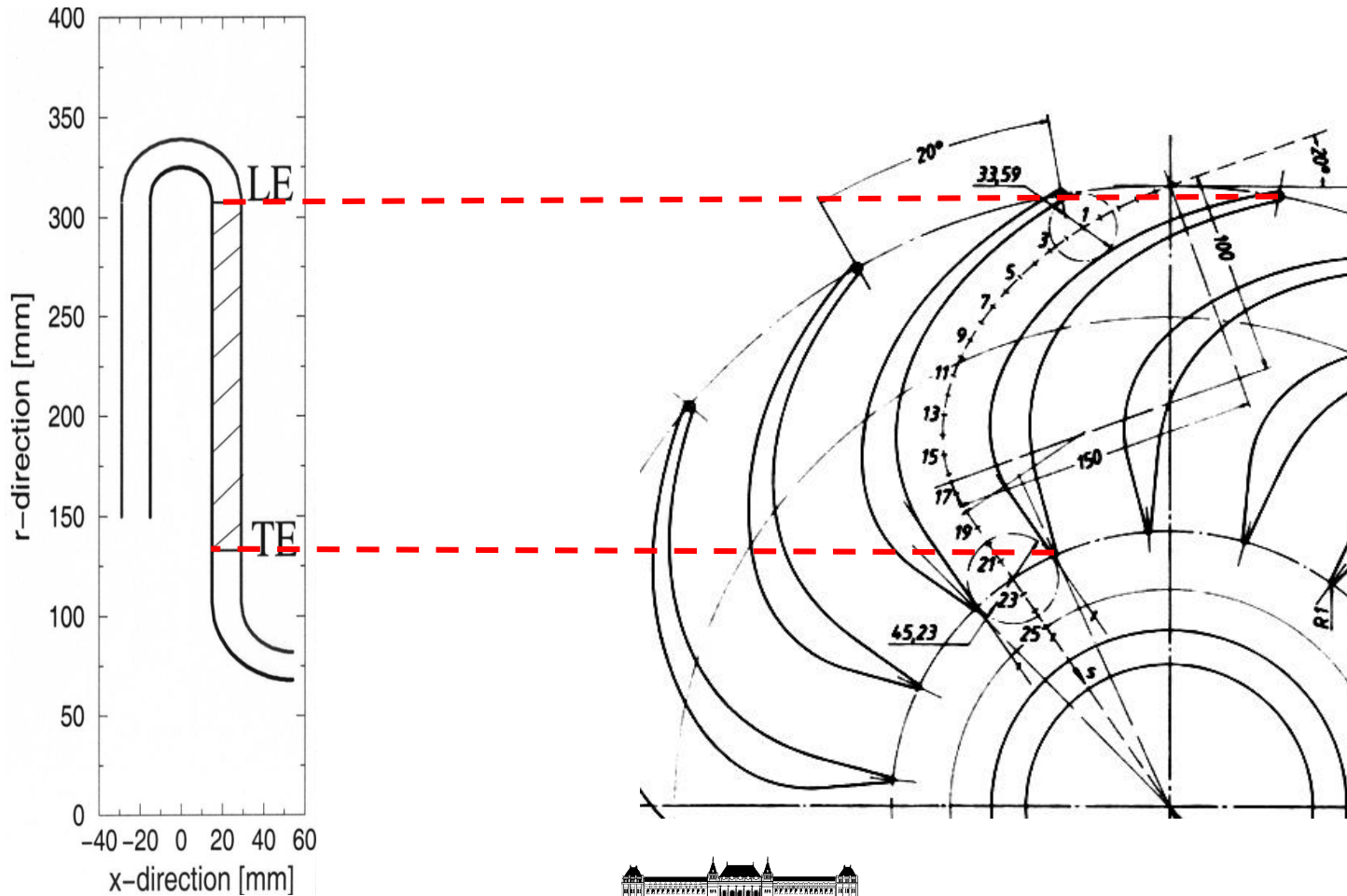


A-C többfokozatú kompresszor

tömegáram: 340 m³/h –ig,
nyomás: 345 bar-ig



Többfokozatú centrifugál kompresszor összekötőcsatorna lapátozásának tervezése – Ipari alkalmazás



CFX

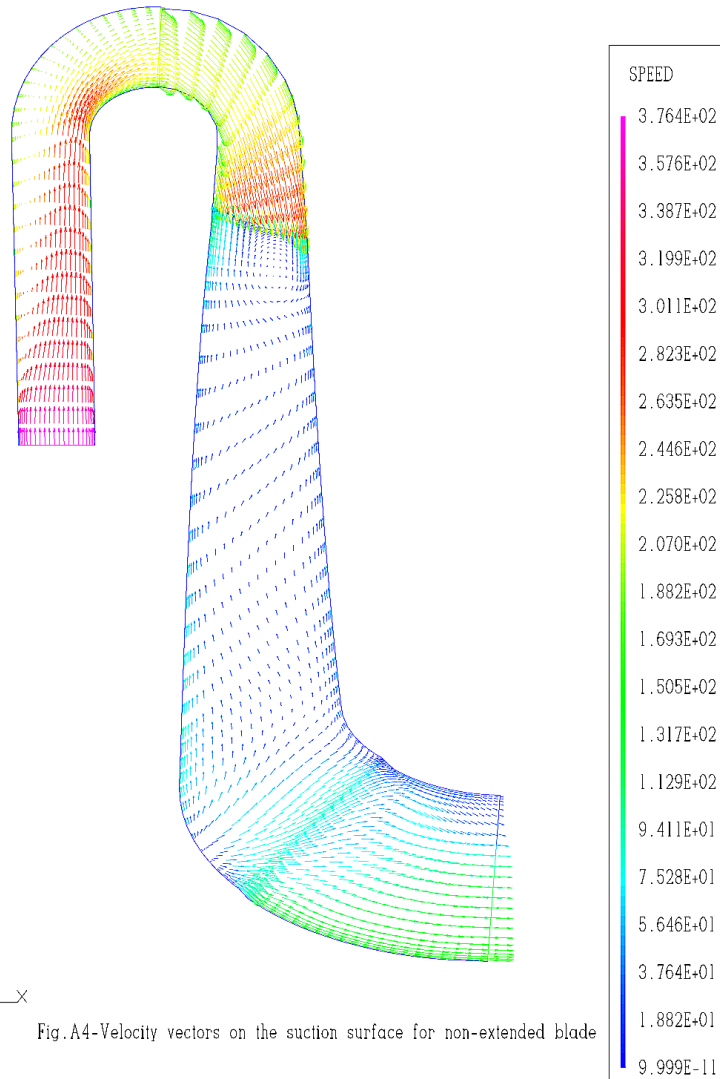


Fig.A4-Velocity vectors on the suction surface for non-extended blade

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

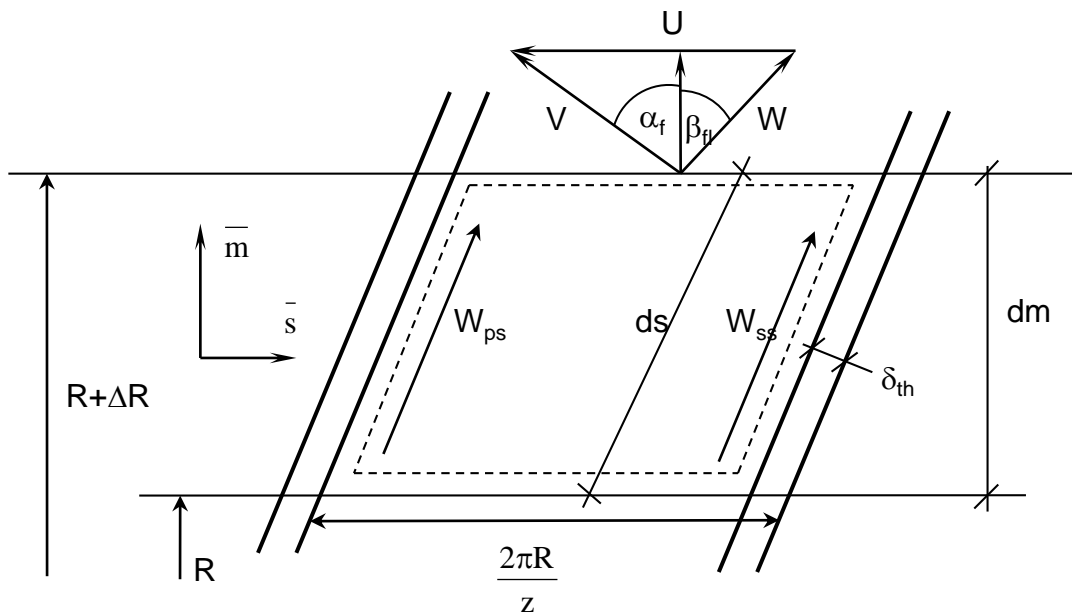
$$\omega = \frac{\overline{P}_{in}^{to} - \overline{P}_{out}^{to}}{\overline{P}_{in}^{to} - \overline{P}_{in}^{st}}$$

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



Többfokozatú centrifugál kompresszor összekötőcsatorna lapátozásának tervezése – ÁLT – Ipari alkalmazás

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



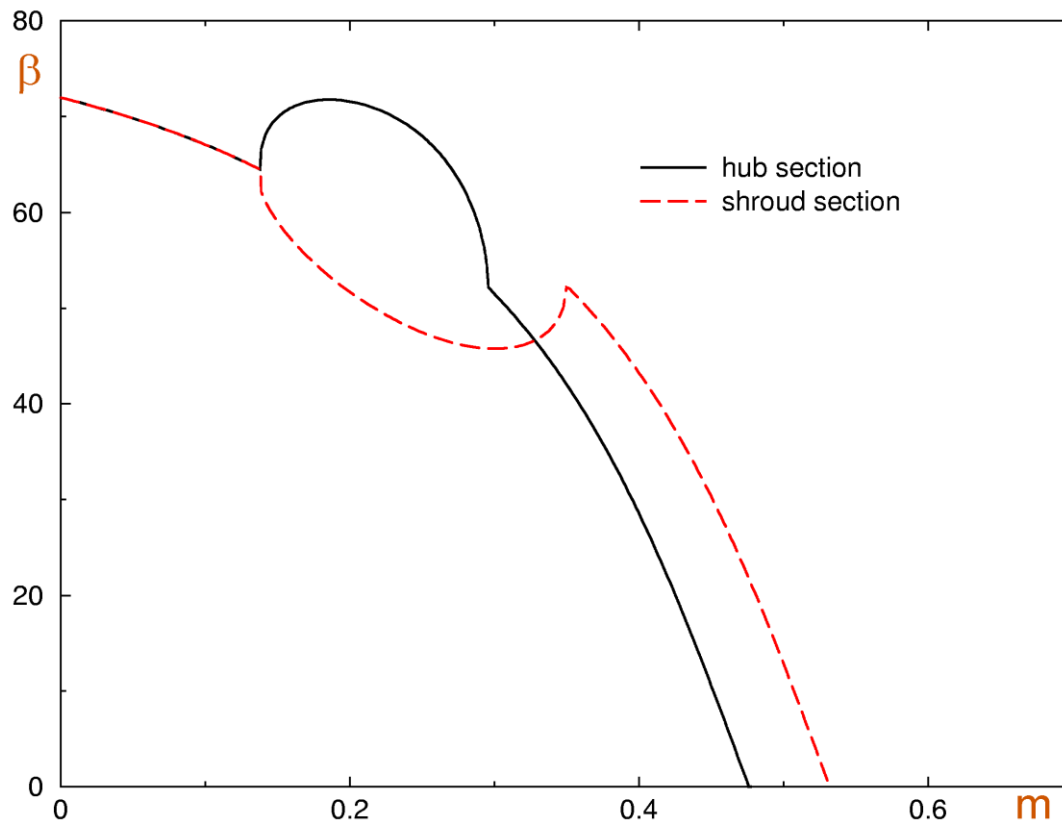
$$W_{ps} - W_{ss} = \left(\frac{2\pi}{z} \right) \frac{C^{t1}}{\cos \beta} \frac{d\beta}{dm}$$

$$R \cdot W_m = C^{t1}$$

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

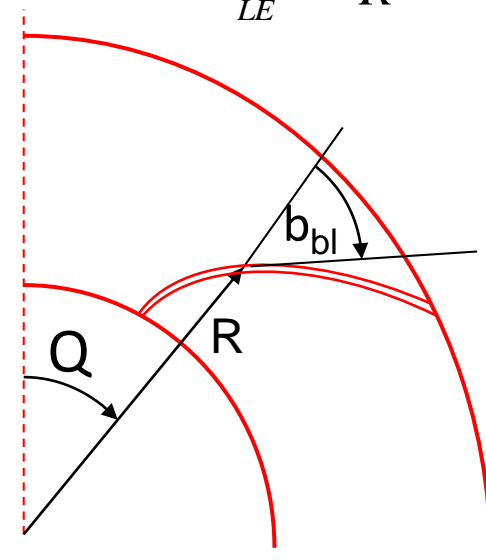


Többfokozatú centrifugál kompresszor összekötőcsatorna lapátozásának tervezése – ÁLT – Ipari alkalmazás

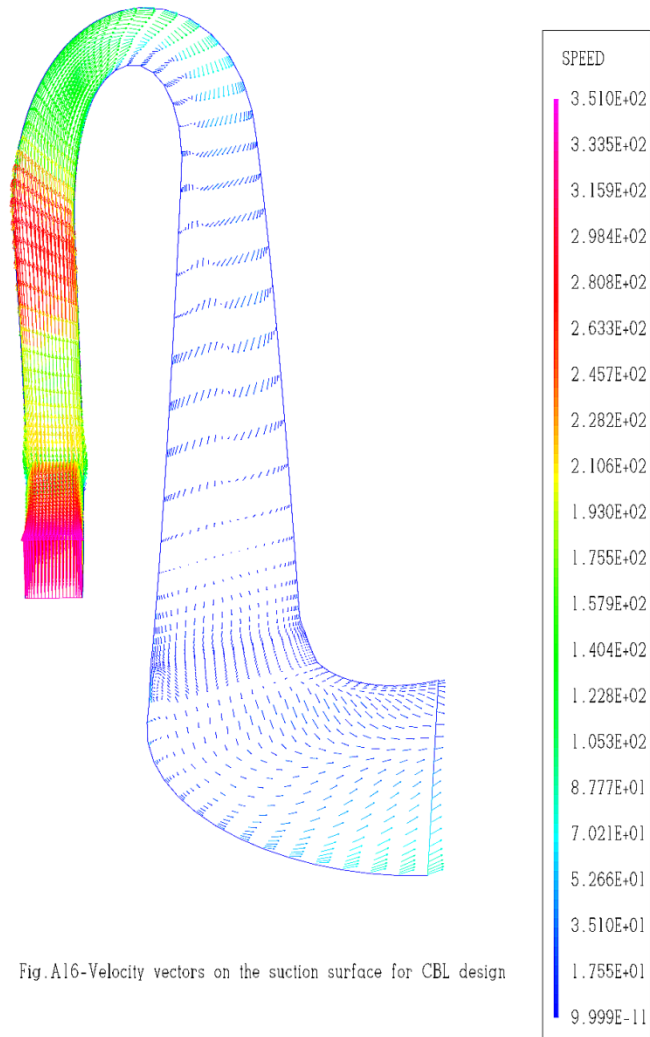


$$\beta_{bl} = \frac{C^{t1}}{C^{t2}} \int_{LE}^{TE} \cos \beta \, dm$$

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



CFX



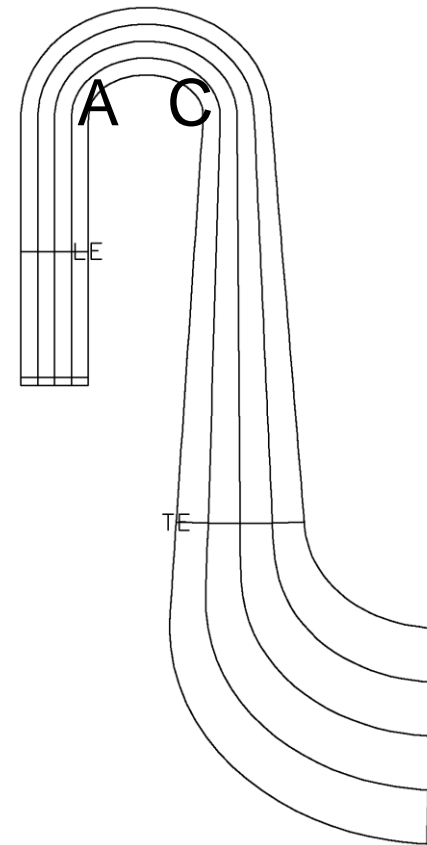
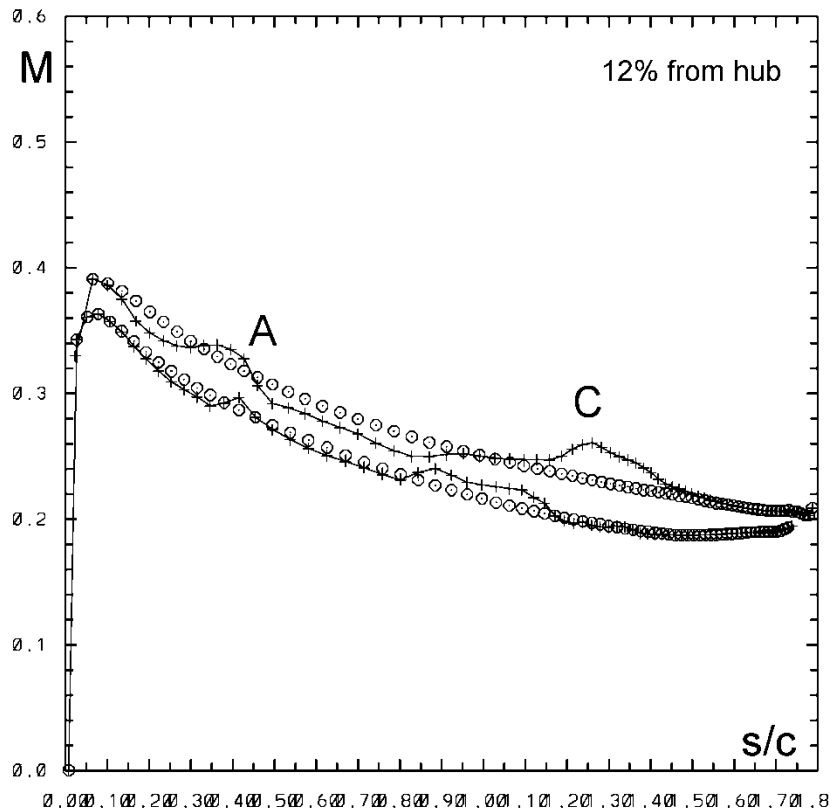
Többfokozatú centrifugál kompresszor összekötő- csatorna lapátozásának tervezése – ÁLT – Ipari alkalmazás

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

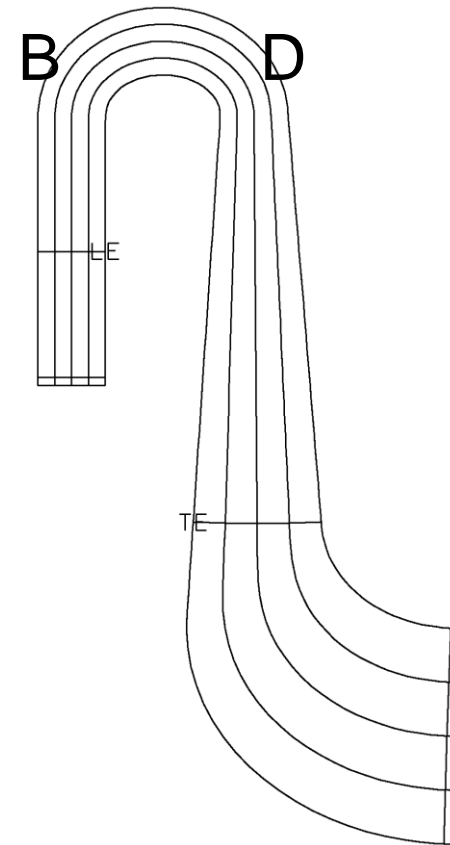
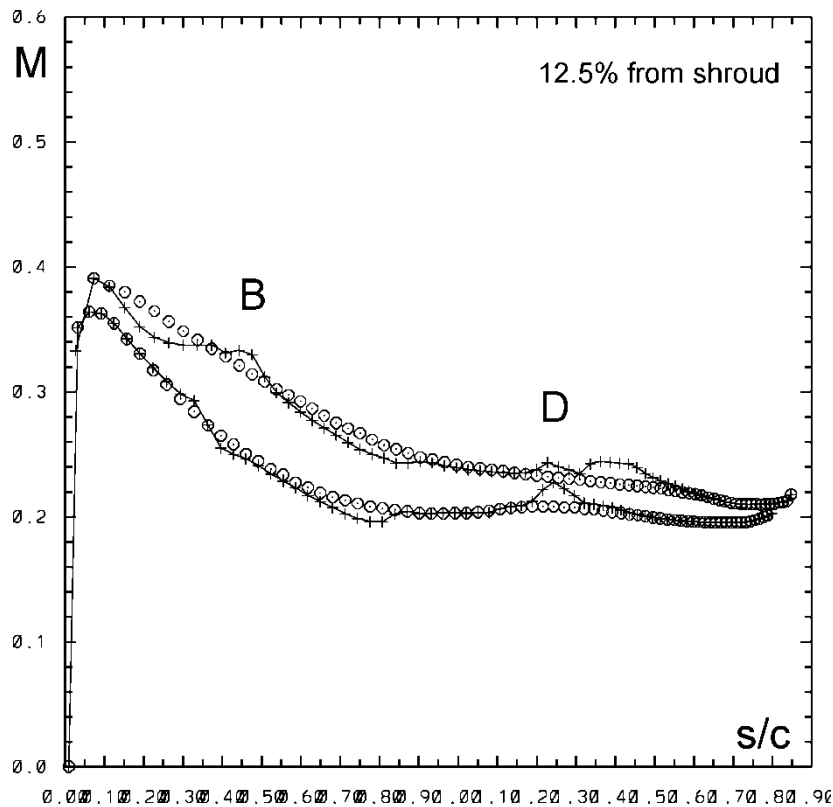
$$C_p = \frac{\overline{p}_{out}^{st} - \overline{p}_{in}^{st}}{\overline{p}_{in}^{to} - \overline{p}_{in}^{st}}$$



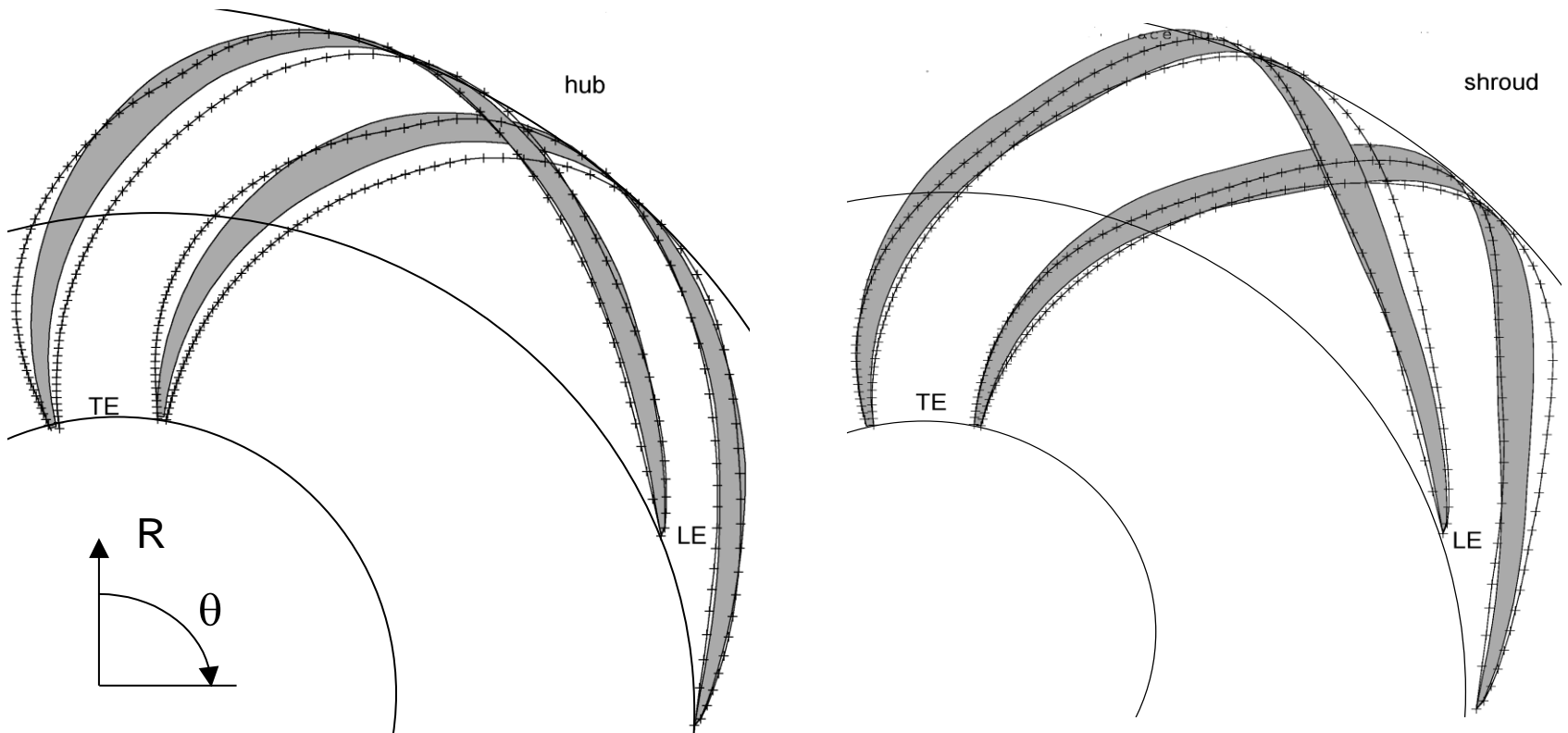
Többfokozatú centrifugál kompresszor összekötő- csatorna lapátozásának tervezése – Inverz Módszer – Ipari alkalmazás



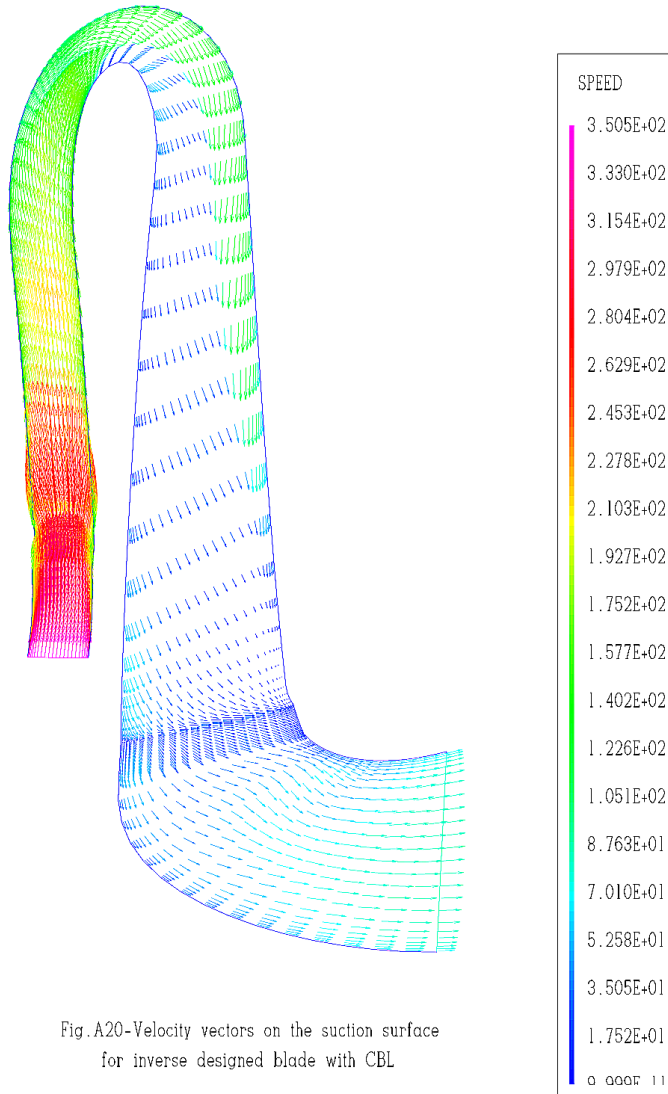
Többfokozatú centrifugál kompresszor összekötő- csatorna lapátozásának tervezése – Inverz Módszer – Ipari alkalmazás



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



CFX



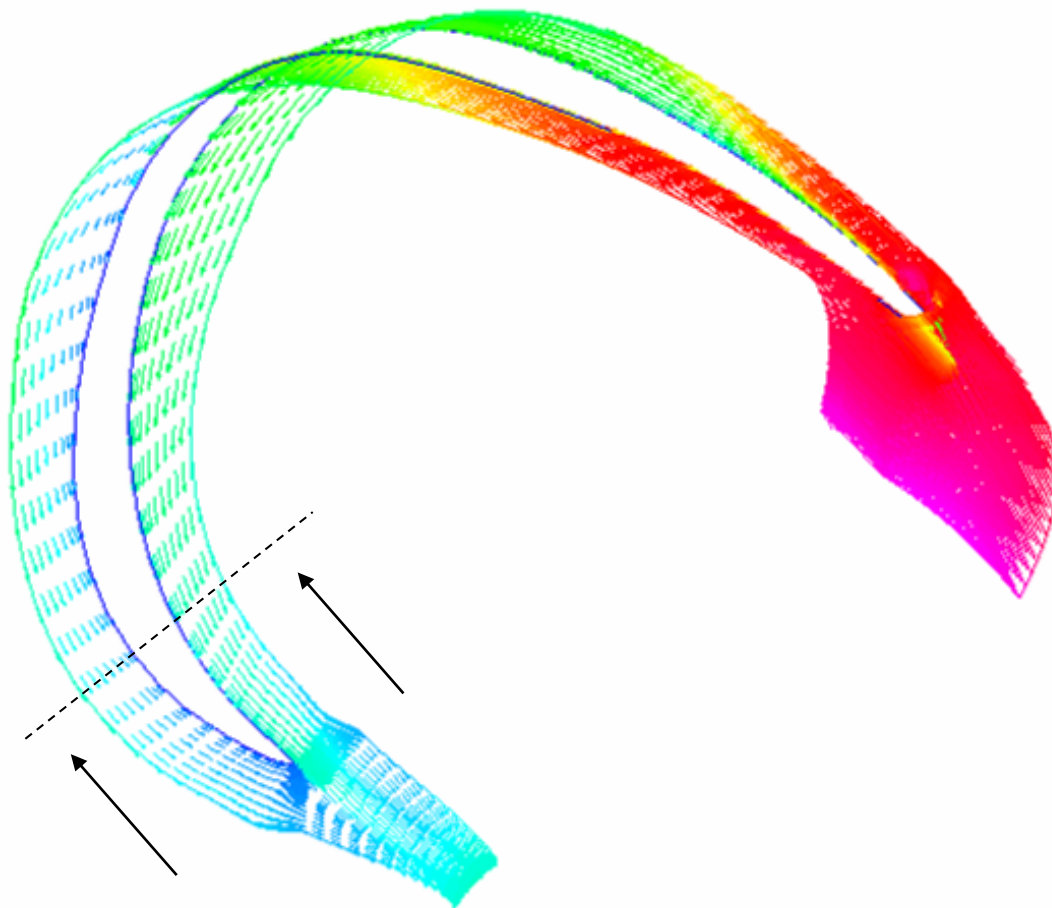
Többszintű centrifugál kompresszor összekötő-csatorna lapátjának tervezése – Inverz Módszer – Ipari alkalmazás

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

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



CFX



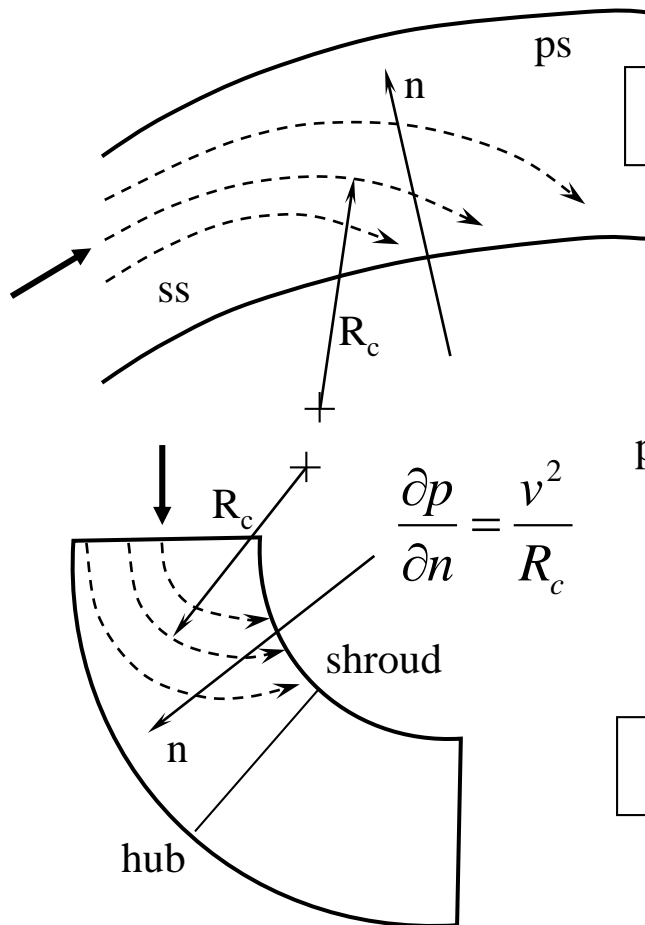
SPEED

3.508E-02
3.330E-02
3.154E-02
2.979E-02
2.804E-02
2.629E-02
2.453E-02
2.278E-02
2.103E-02
1.927E-02
1.752E-02
1.577E-02
1.402E-02
1.226E-02
1.051E-02
8.763E-03
7.010E-03
5.258E-03
3.508E-03
1.752E-03
9.999E-04

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

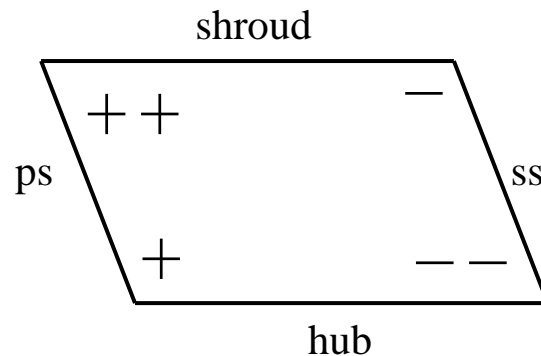


Többfokozatú centrifugál kompresszor összekötő- csatorna lapátozásának tervezése – Lapátelhajlítás – Ipari alkalmazás

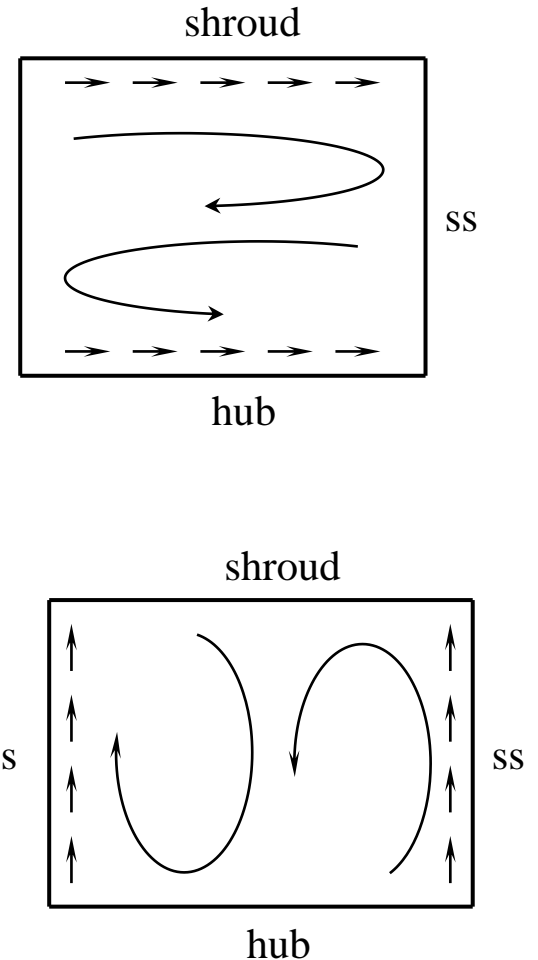


$$\frac{\partial p}{\partial n} = \frac{v^2}{R_c}$$

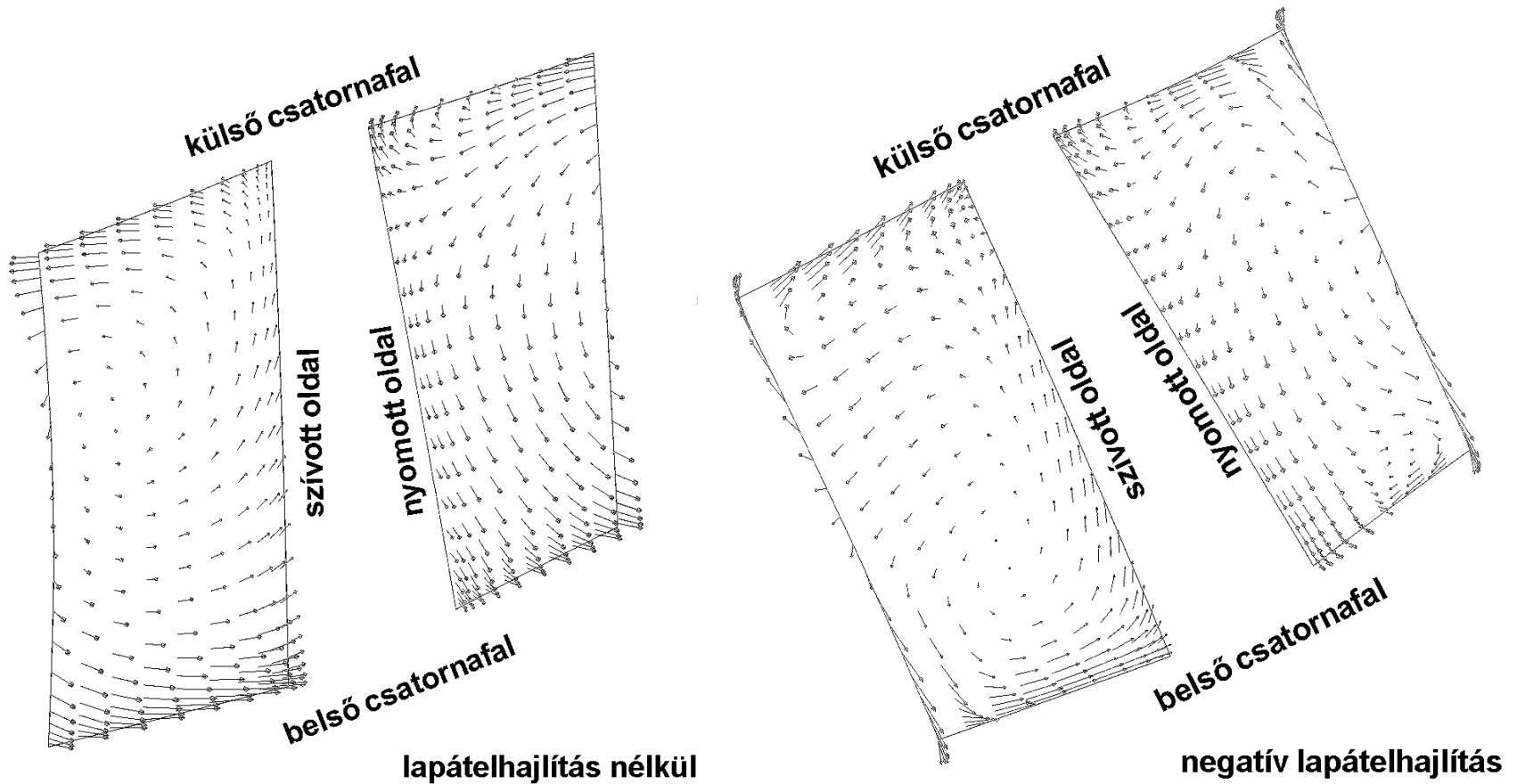
Csatorna örvény



Lapát örvény



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
ω	0.44813	0.3277	0.27847	0.263
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 Drive
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 Drive
Canonsburg, PA 15317, ansysinfo@ansys.com, <http://www.ansys.com>, USA, 2010

Köszönöm a figyelmet.

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

