The EMPIR initiative is co-funded by the European Union's Horizon 2020 research and innovation programme and the EMPIR Participating States

MetHyInfra

EMPIR CALL 2020

Metrology infrastructure for high-pressure gas and liquified hydrogen flows

CFD Workshop

Part 2: CFD for Critical Flow Venturi Nozzles (CFVNs)

"Challenges, approaches and results"

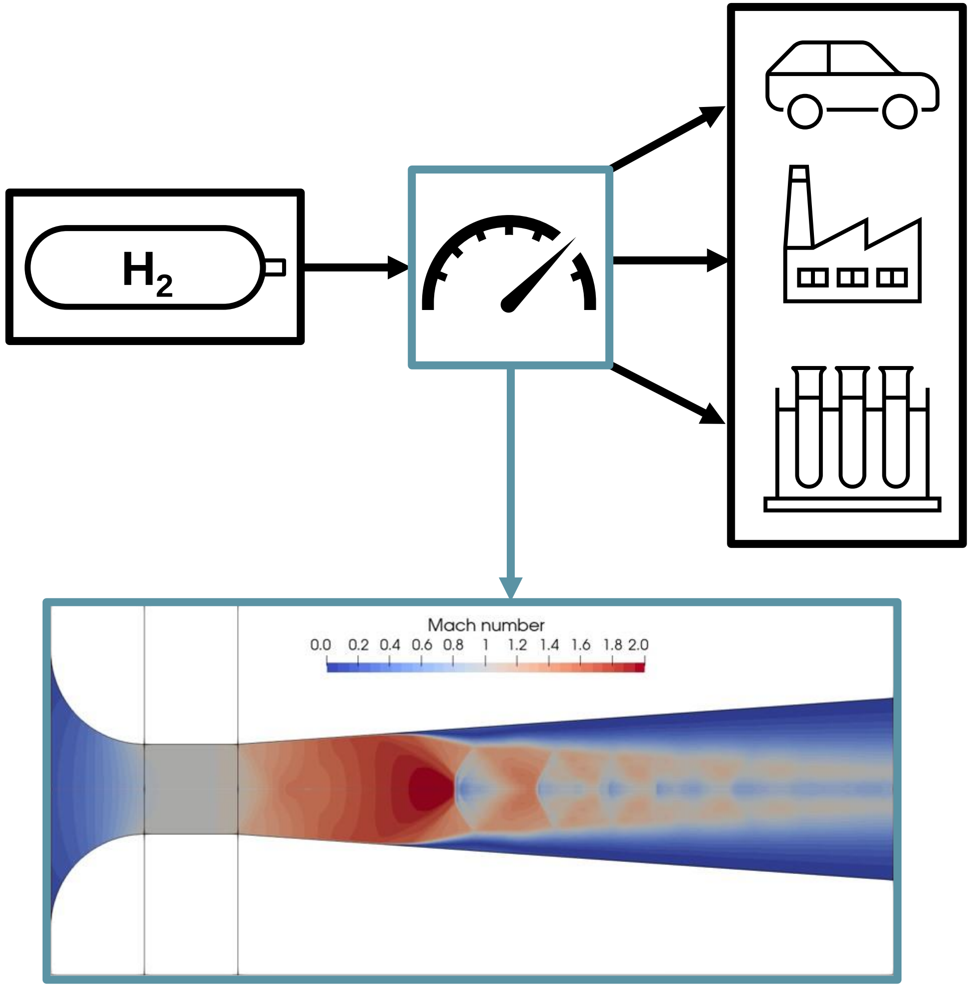
Sonja Schmelter and Sebastian Weiss
Working Groups 8.41 “Modelling and Simulation” and 1.45 “Hydrogen Quantity Metering”
Physikalisch-Technische Bundesanstalt (PTB), Berlin, Germany

Motivation

Hydrogen at high pressures

- promising energy carrier for use in climate-neutral applications in industry and transport
- needs to be stored at high pressures (up to 1000 bar) due to its low volumetric energy density

➢ to monitor hydrogen consumption, a verifiable flow measurement is required



Critical flow Venturi nozzles (CFVNs)

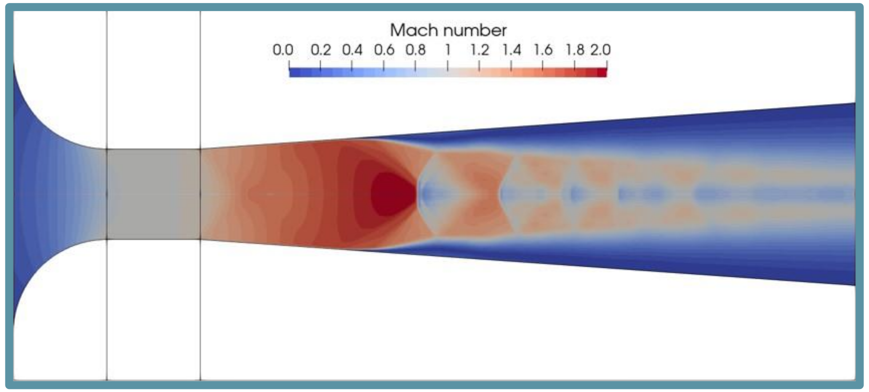
- state-of-the-art secondary standard gas flow meter
- gas accelerates to *Ma* = 1 (critical flow)
- international standards are limited to air and natural gases up to 200 bar

➢ **Flow simulation of hydrogen at high pressures**

CFD for high-pressure hydrogen flows

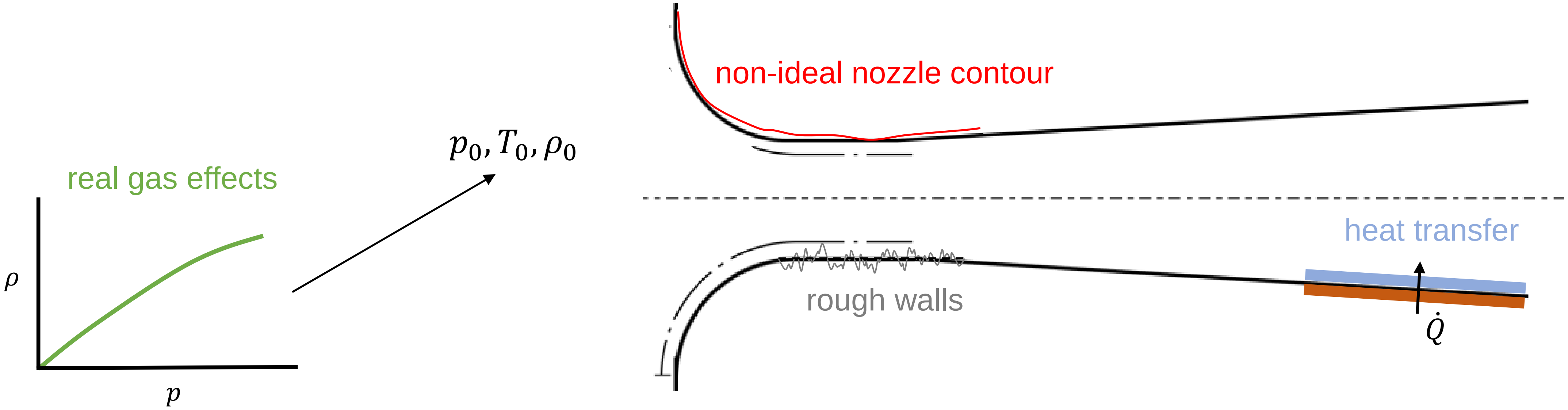
Aim:

Better understanding of the physics of
high-pressure hydrogen flows inside the nozzle



Need:

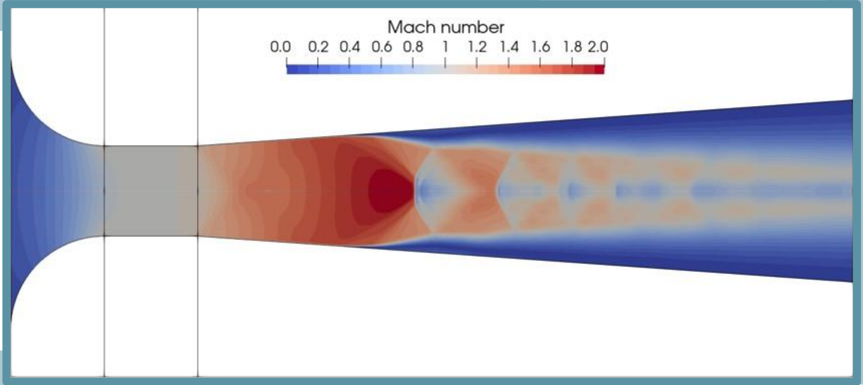
CFD model that takes the most relevant **real gas effects** into account,
modeling of **non-ideal, rough, and non-adiabatic walls**



Challenges

For a better understanding of the flow physics:

develop a CFD model that includes real gas effects, wall roughness, and heat transfer



Towards a “dry” calibration:

transform data
generated by dimensional
characterization
into parameters
of a CFD-applicable
computer model
of the physical process

For identifying the most significant parameters influencing the flow through critical nozzles:

perform efficient parameter studies
based on a validated CFD model

Approach

Start with simple model and extend it step by step:

- Choice of appropriate solver
- Choice of appropriate turbulence model
- Inclusion of real gas effects
- Consideration of non-ideal shapes and rough walls
- Consideration of non-adiabatic walls

Start with studying the influence of different parameters / effects systematically before considering specific nozzles / setups

MetHyInfra

Approach

Real gas model

Surrogate model

1. Study the influence of different parameters systematically

Choice of turbulence model

Parametrized shape (toroidal → cylindrical)

Rough walls

Heated walls

Validation with existing data
(e.g., from the literature)
or with experiments designed for this purpose

3. Substitute CFD model by cheaper surrogate model

→ allows to study systematic influence of various parameter combinations

→ basis for the “dry” calibration of nozzles

2. Modeling specific nozzles / settings
(e.g., from the MetHyInfra project)

Dimensional charac-terization of nozzles
→ real shape
→ roughness level

Operating conditions

Boundary conditions on wall

Validation with corresponding experiments

Realization in MetHyInfra project

CFD work summarized in **one work package**

- Title: **“Development of a CFD model for high pressure hydrogen flows”**
- Partners involved: **PTB, ESI**
- Work divided into **4 tasks**

Task 1:
Literature review and strategy for CFD modeling

Task 2:
CFD model for critical nozzle flows

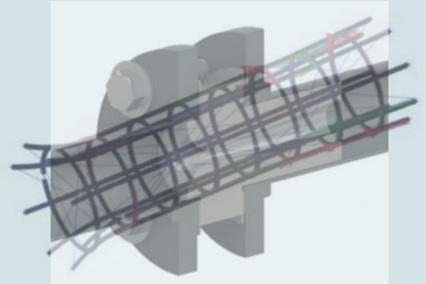
Task 3:
CFD model for high pressure hydrogen flows

Task 4:
Influence of rough and non-adiabatic walls

Interaction with other parts of the project

METAS:

Dimensional characterization of nozzles



real shape

+ known roughness

NEL, Cesame, JV, METAS, MH, UL:

Experiment

$c_{D,\ \mathrm{exp}}$

Comparison

Validation

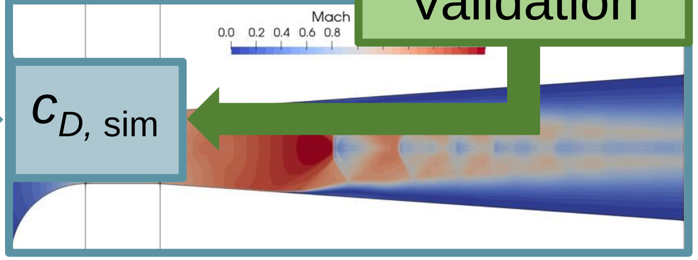
ESI, PTB:

Parametrized mesh

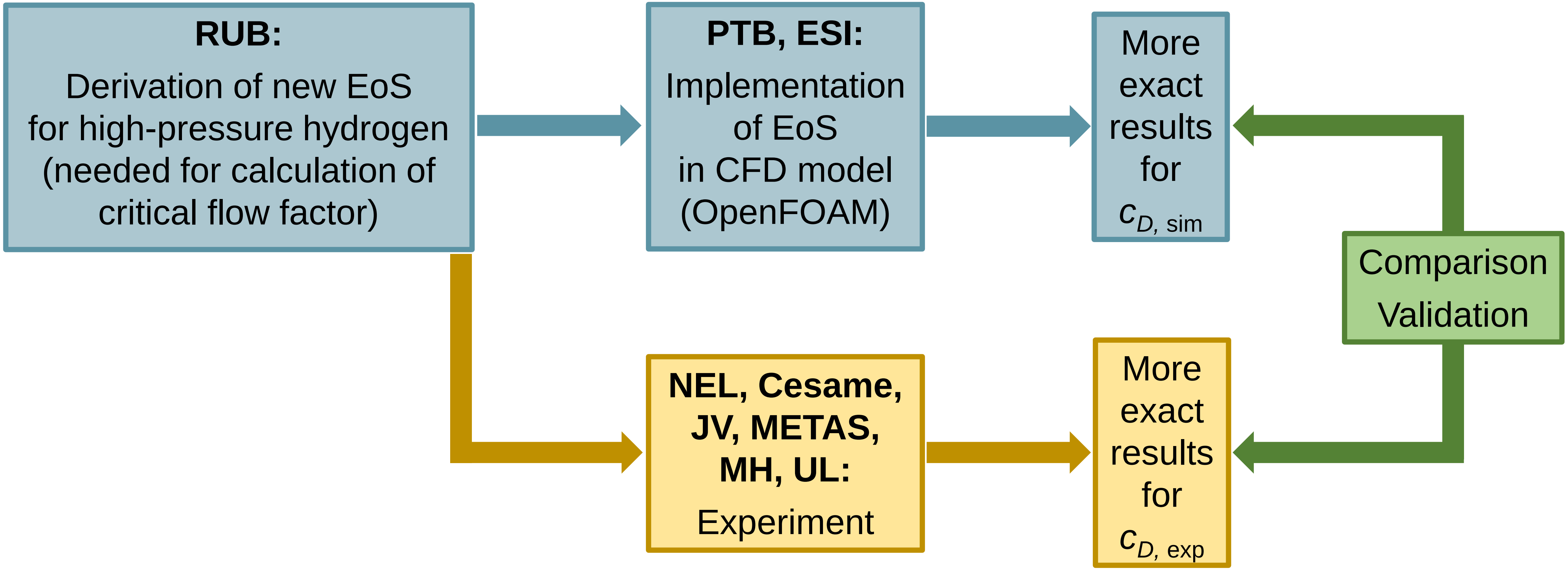
PTB, ESI:

CFD model

$c_{D,\ \mathrm{sim}}$



Interaction with other parts of the project



Realization in MetHyInfra project

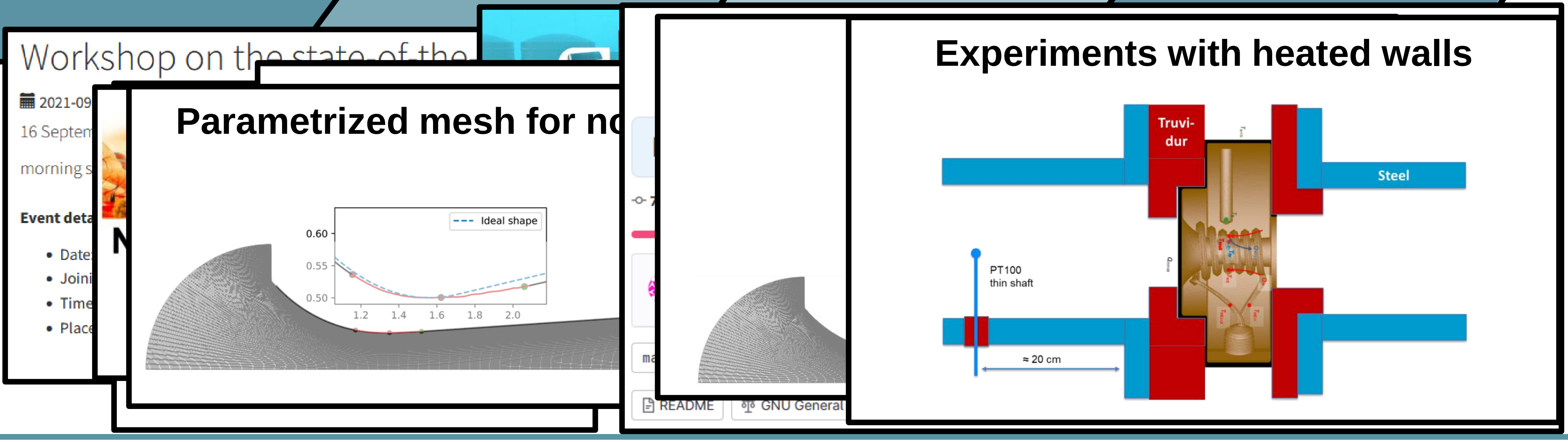
Development of a CFD model for high pressure hydrogen flows

Task 1:
Literature review and strategy for CFD modeling

Task 2:
CFD model for critical nozzle flows

Task 3:
CFD model for high-pressure hydrogen flows

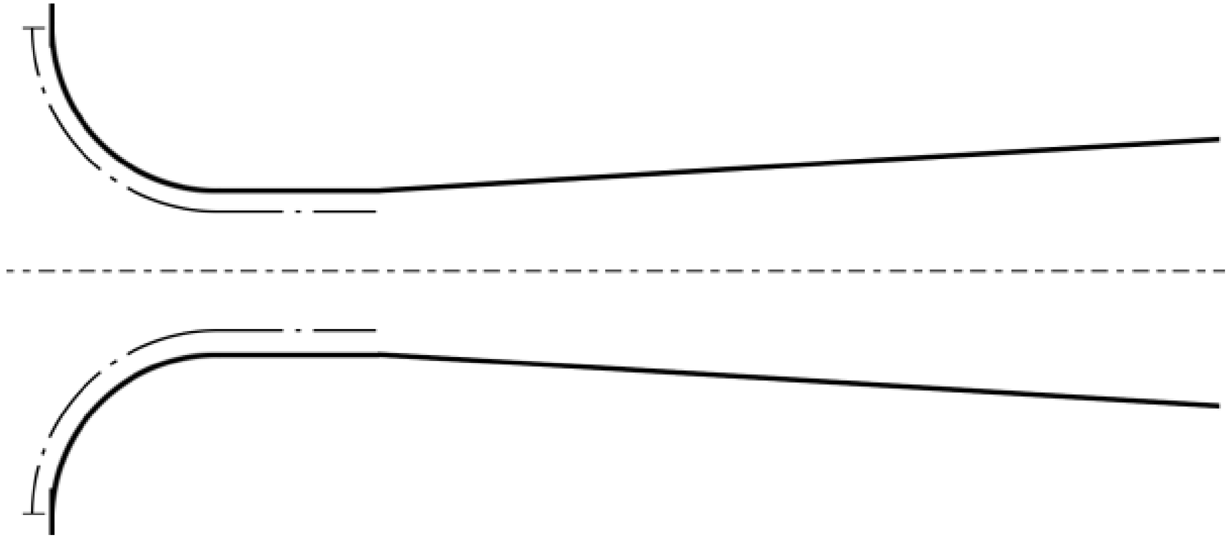
Task 4:
Influence of rough and non-adiabatic walls



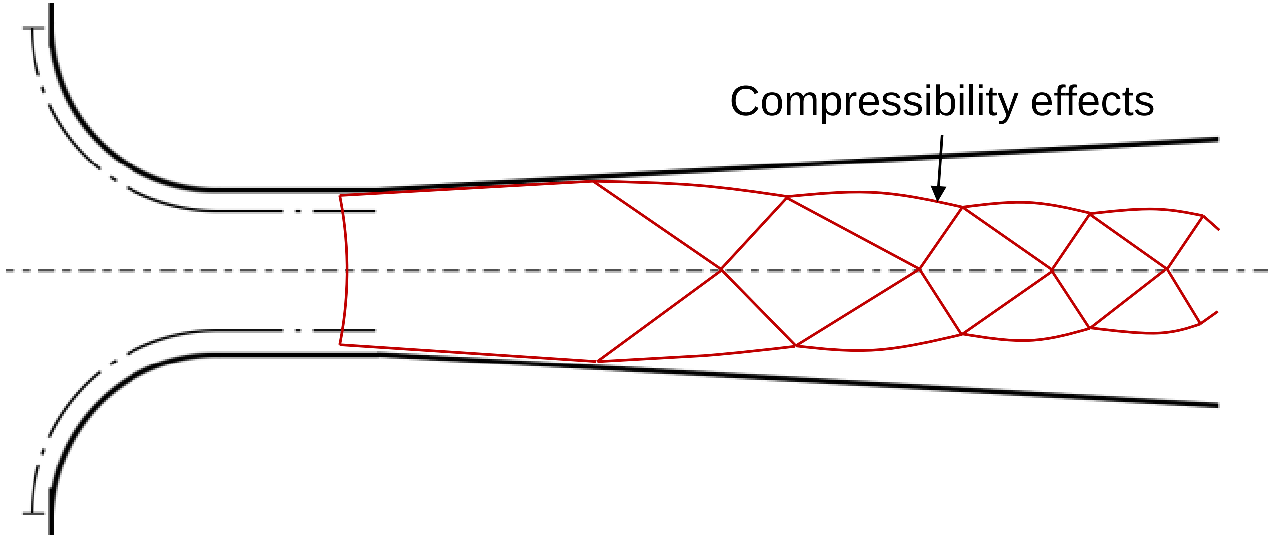
Results

Come back to CFVN (critical flow Venturi nozzle)

and consider different effects

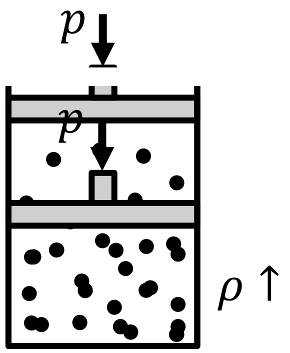


Compressibility effects



Compressibility effects

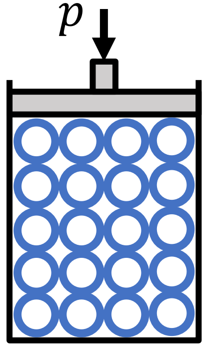
Compressible fluid



$$a_{compr.} = \sqrt{\kappa R_M T} \quad \text{(Ideal gas)}$$

- Fluid properties change after arrival of sound wave

Incompressible fluid

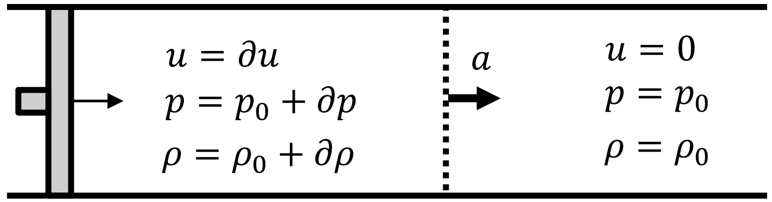


$$a_{incompr.} \to \infty$$

- Fluid properties change instantaneously

Speed of sound

- Propagation velocity of small pressure disturbances in a medium



Piston moves at velocity ∂u

Wave moves away from piston at velocity a

- From conservation law of mass and momentum, the speed of sound can be derived as

$$a = \sqrt{\left(\frac{\partial p}{\partial \rho}\right)_s}$$

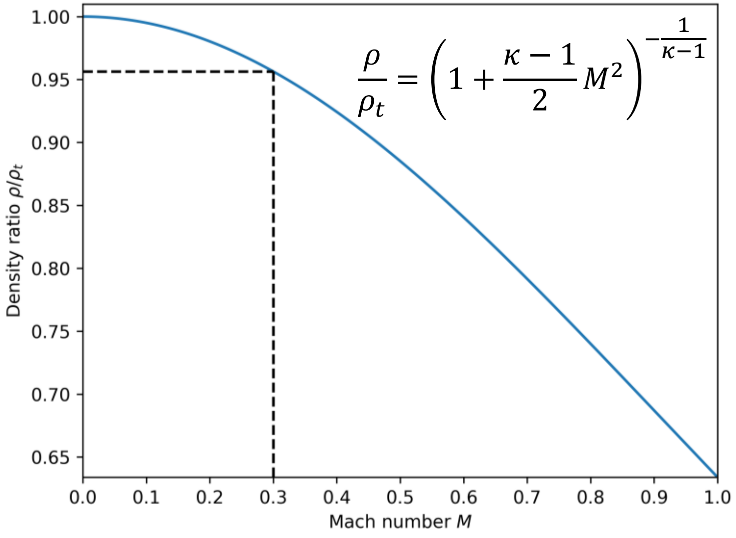
Compressibility effects

Mach number

- Ratio of flow velocity to local speed of sound

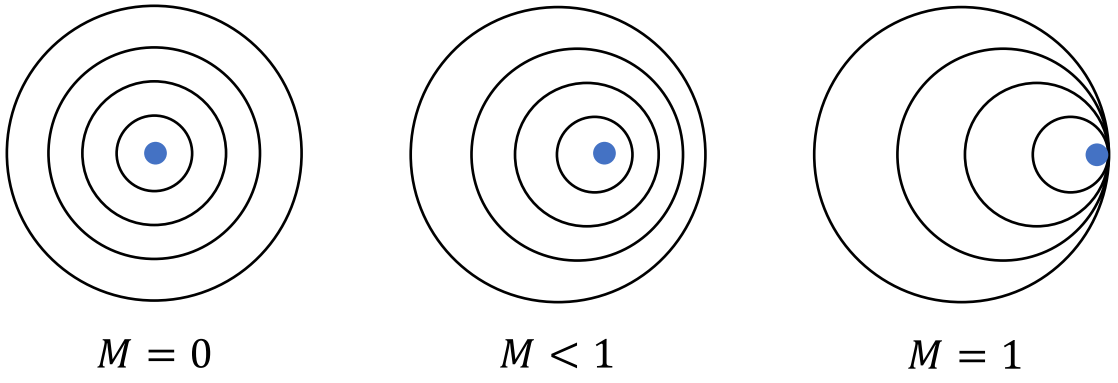
$$M = \frac{u}{a}$$

- Typically, flows are considered compressible when Mach number exceeds 0.3



- At *M = 0.3*, change in density is ca. 5% and can't be neglected

- What happens when flow reaches *M = 1*?

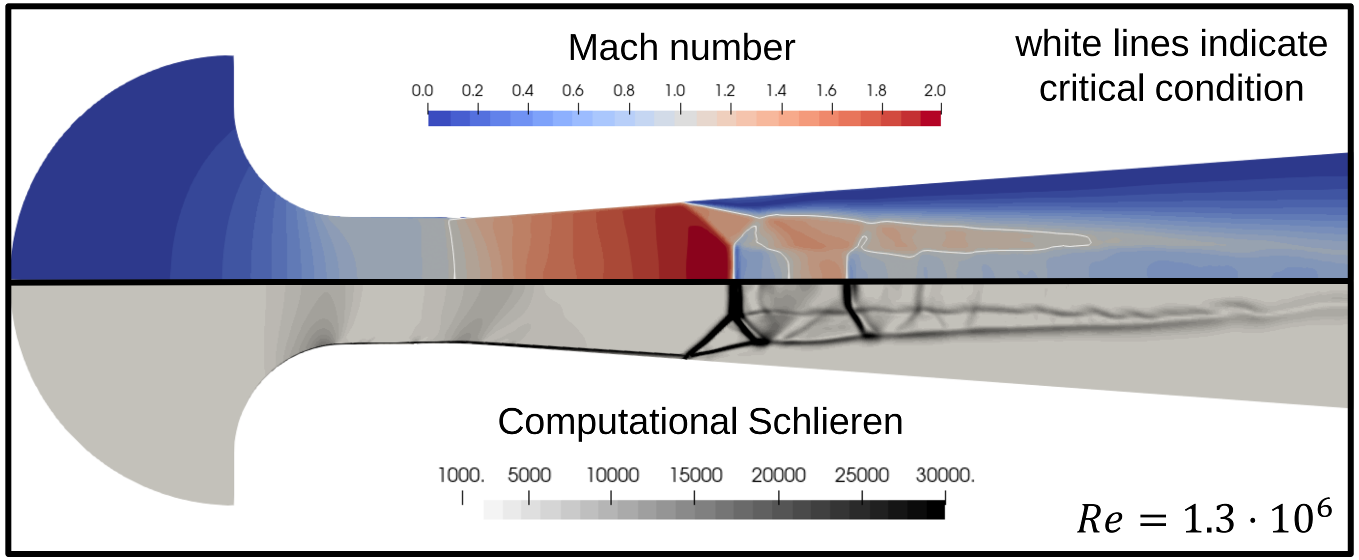


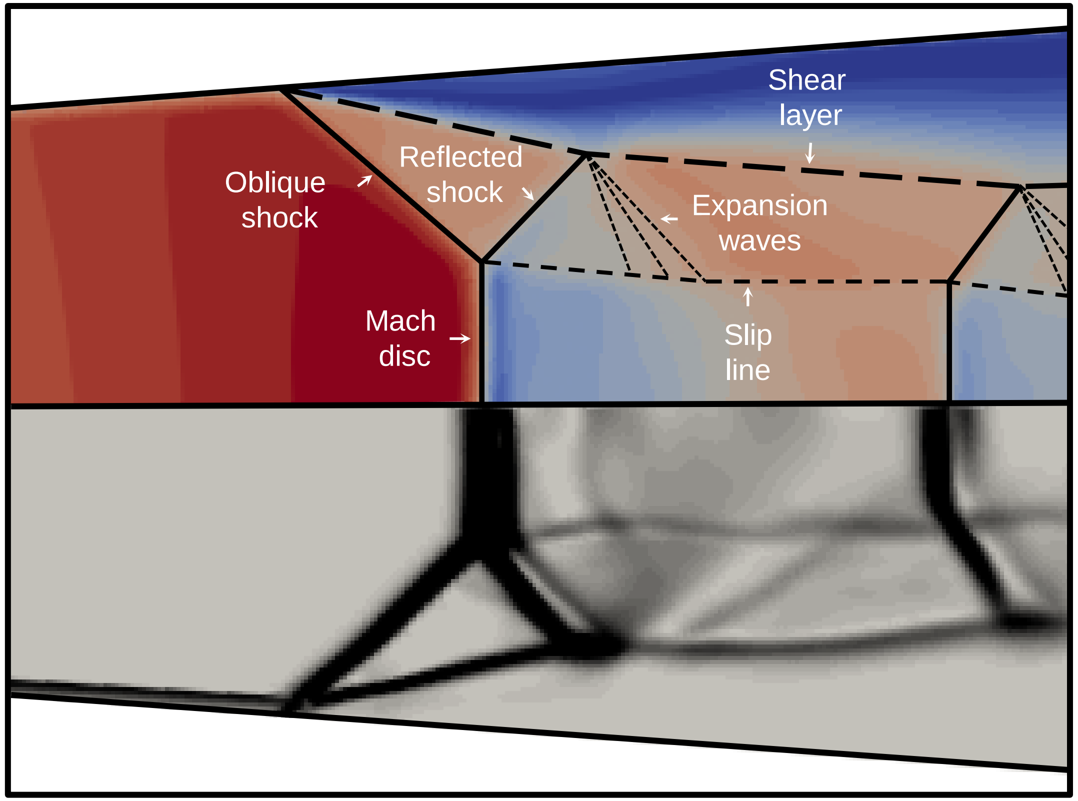
- Waves in front of object get compressed
- In flows, where *M ≥ 1*, small disturbances can't propagate upstream
- In the context of critical nozzles:
  - Flow rate only depends on upstream conditions

Compressibility effects

- A compressible flow solver is used for the critical nozzle flow
  - ➢ here: sonicFOAM

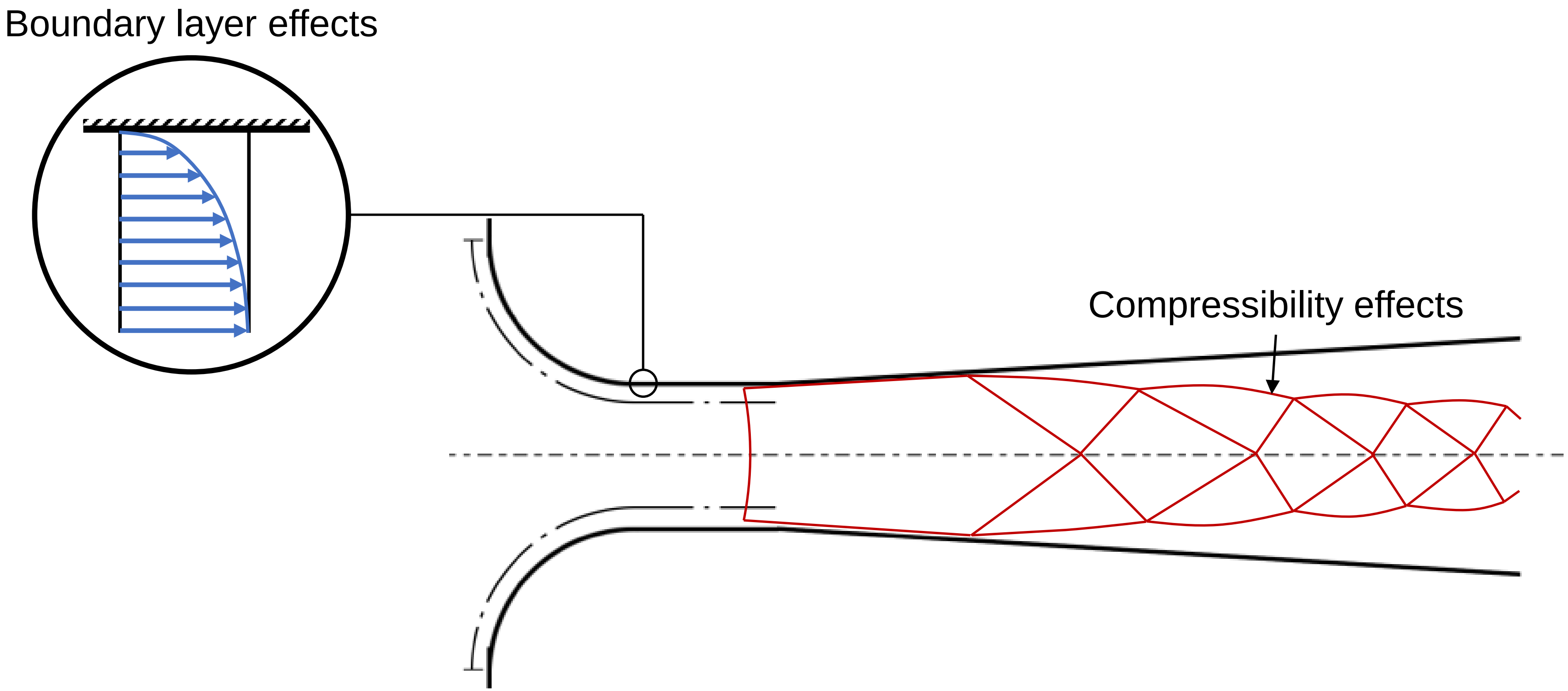
- Let's have a look at the flow field





➢ Critical flow (*Ma* = 1) is reached in nozzle throat (see white line)

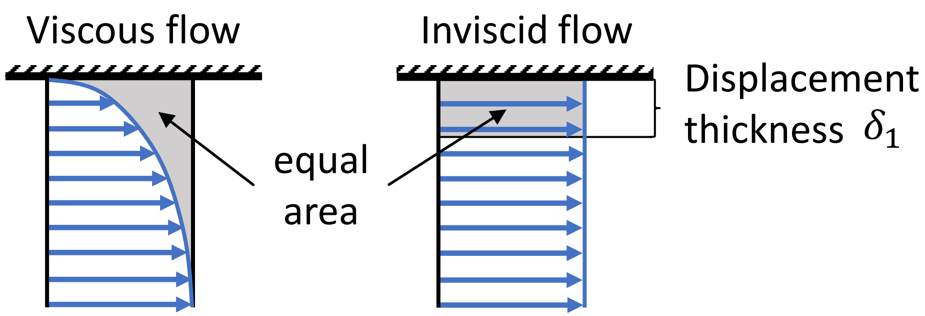
Boundary layer effects



Boundary layer effects

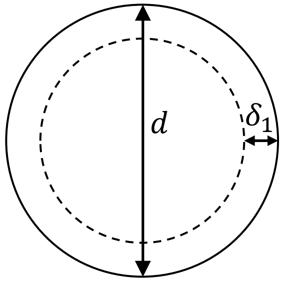
Boundary layer (BL)

- ➢ thin region close to wall, where viscous effects on velocity profile are significant



Displacement thickness

- ➢ reduces effective cross-sectional area



$$C_D \sim \left(1 - \frac{2\delta_1}{d}\right)^2$$

Reynolds number

- ➢ Ratio of inertial to viscous forces

$$Re = \frac{\rho u d}{\mu}$$

Laminar BL

- Smooth, rectified flow
- Occurs at lower *Re* values
- ➢ Viscous forces dominate

Turbulent BL

- Chaotic, swirling flow
- Occurs at higher *Re* values
- ➢ Inertial forces dominate

- In CFD, RANS turbulence models with wall functions are used
- ➢ How is turbulence actually modeled?
- ➢ And what are wall functions for?

Turbulence modeling (1)

- Navier Stokes equations (incompressible, easier to explain):

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho u_i u_j\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right]$$

- Reynolds-averaged Navier Stokes (RANS) equations:

$$\frac{\partial(\rho \bar{u}_i)}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho \bar{u}_i \bar{u}_j\right) = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right) - \rho\overline{u_i' u_j'}\right]$$

Reynolds decomposition

$$u(\boldsymbol{x},t) = \bar{u}(\boldsymbol{x}) + u'(\boldsymbol{x},t)$$
$$p(\boldsymbol{x},t) = \bar{p}(\boldsymbol{x}) + p'(\boldsymbol{x},t)$$

Mean component

Fluctuating component

- The Reynolds-averaging process results in an additional stress term:

$$-\rho\overline{u_i' u_j'}$$

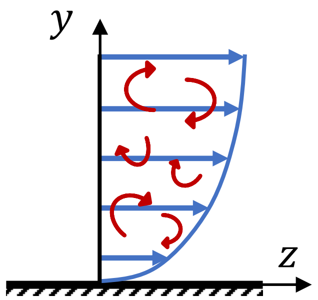
- In order to solve the RANS equations, we need to express the Reynolds stress in terms of the mean flow quantities

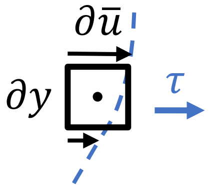
➢ This is the turbulence closure problem

Turbulence modeling (2)

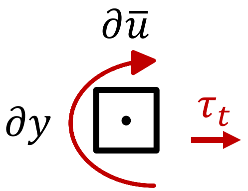
MetHyInfra

- Consider a simple turbulent shear flow in 2D (like close to the wall of the nozzle):





Mean flow



Eddy

➢ The fluid element is sheared by the mean flow and by the eddy

Shear stress from mean flow (viscous shear):

$$\tau = \mu \frac{\partial \bar{u}}{\partial y}$$

Shear stress from eddies / turbulence is given by Reynolds stress:

$$\tau_t = -\rho \overline{u'v'}$$

<u>Eddy viscosity model (or Boussinesq‘s hypothesis:)</u>

$$\boxed{-\rho \overline{u'v'} = \mu_t \frac{\partial \bar{u}}{\partial y}}$$

with eddy / turbulent viscosity μ_t

➢ μ_t needs to be modeled in turbulence model

General form

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

μ_t is considered isotropic

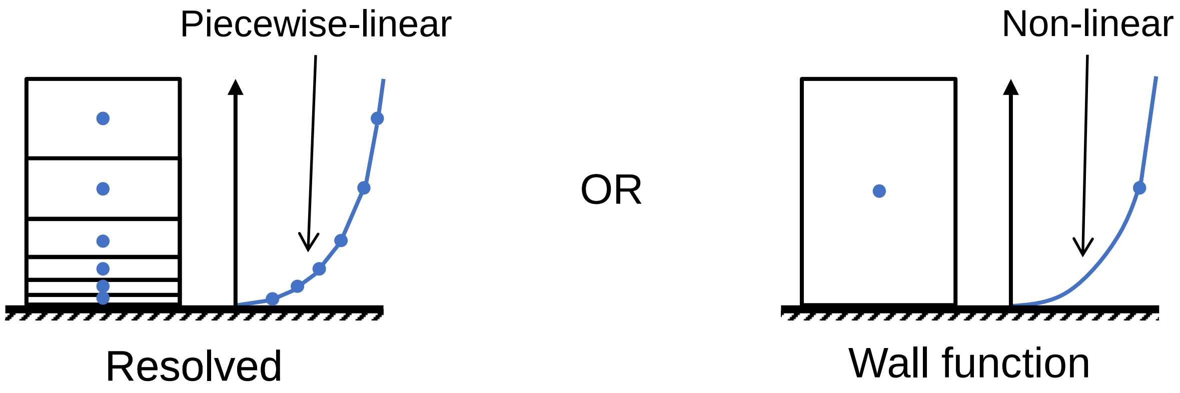
Turbulence modeling (3)

k-ω Shear Stress Transport (SST) model

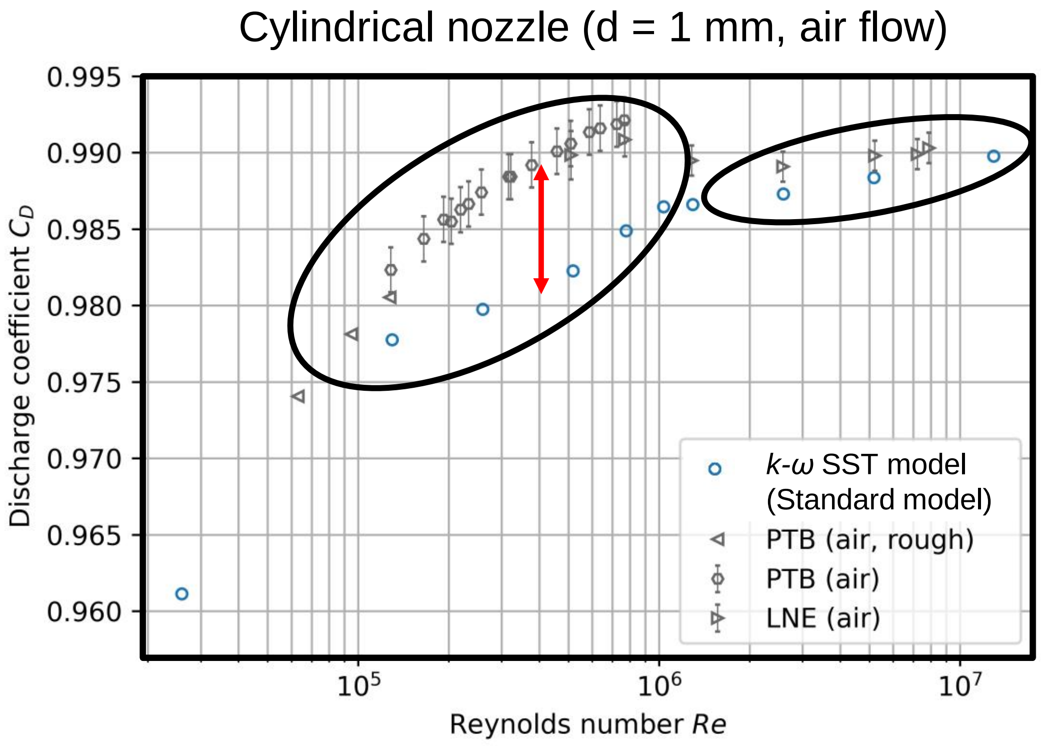
- In order to determine eddy viscosity, two additional transport equations are solved:
  - Turbulence kinetic energy *k*
  - Turbulence specific dissipation rate *ω*
- Combines strenghts of *k-ω* model (inner region of boundary layer) and *k-ε* model (free stream)

Automatic wall treatment

- Depending on y+ value (dimensionless wall distance):

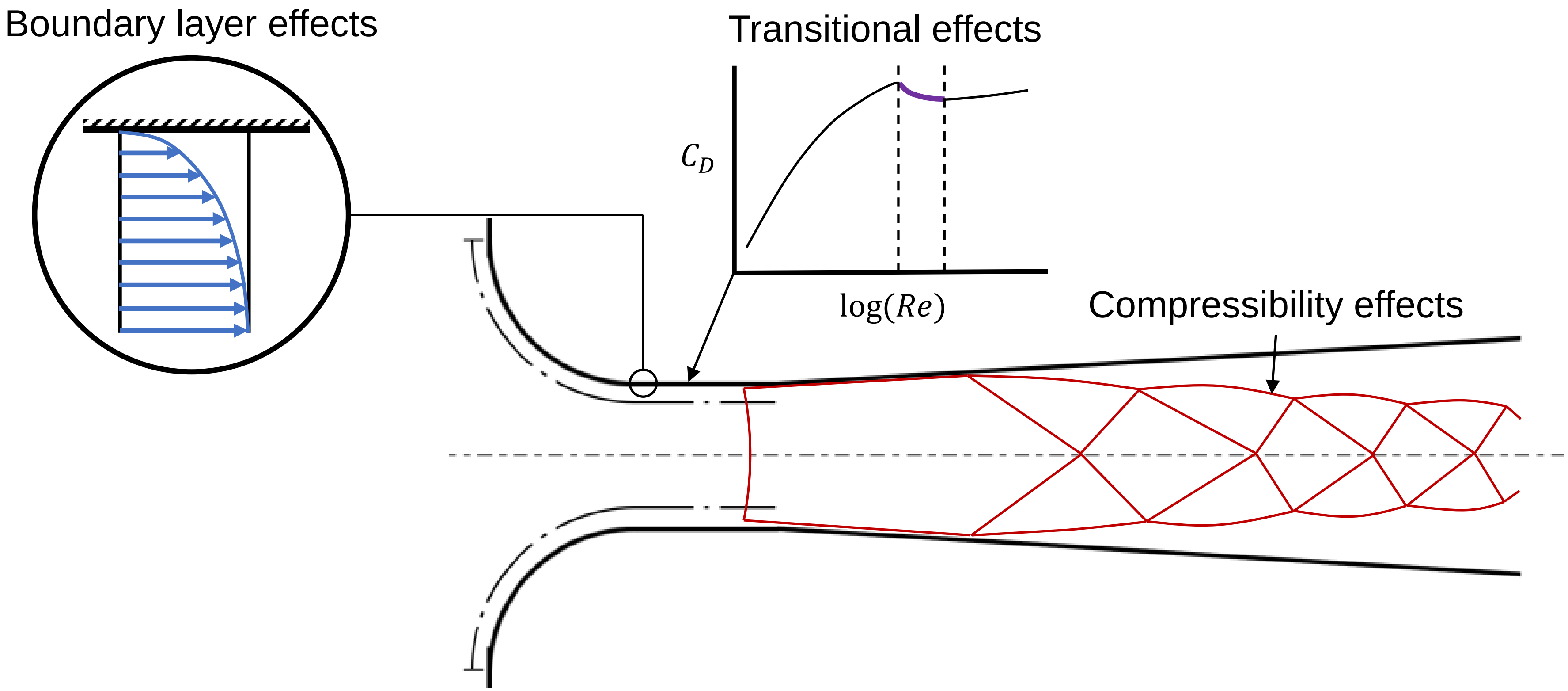


Results



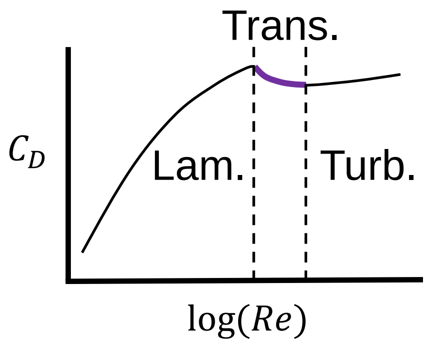
- Performs well for high Re numbers (turbulent region)
- Underperforms for low Re numbers (laminar region)
- ➢ Better predictive model required

Transitional effects



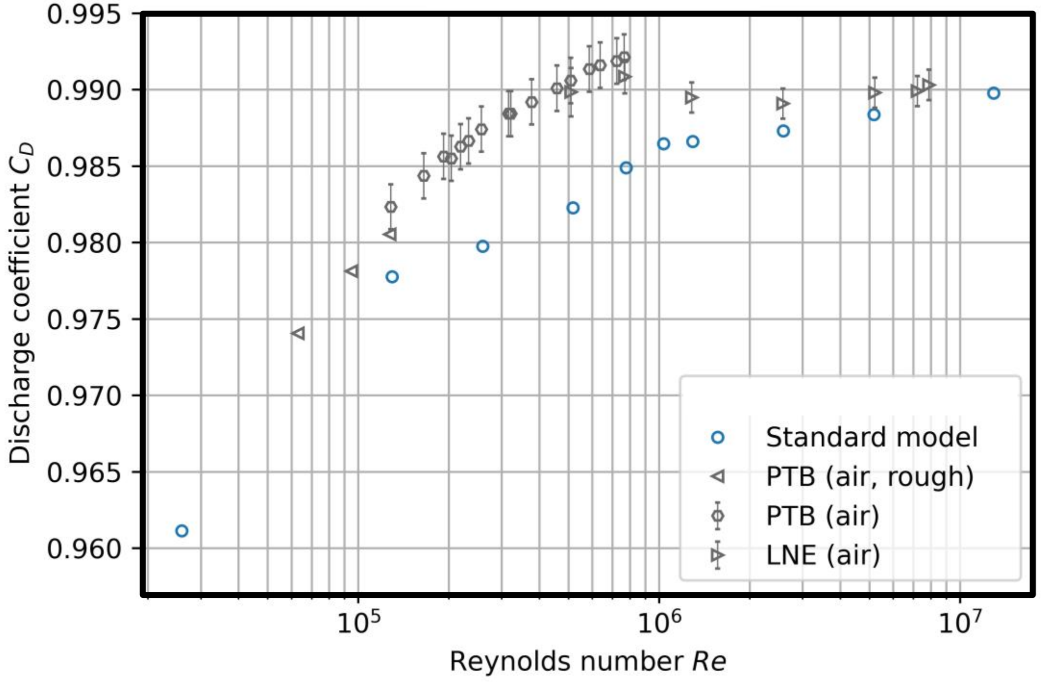
Transitional effects

- Transition describes the process of a laminar boundary layer becoming turbulent
- It occurs in a specific Reynolds number *Re* range (typically at *Re* = 10^6 for CFVNs)



- In CFD, transitional turbulence model uses trigger parameter γ to model transition process, where
  - $\gamma = 0$ for laminar flow
  - $\gamma = 1$ for turbulent flow
  - $0 < \gamma < 1$ for transitional flow

Cylindrical nozzle (d = 1 mm, air flow)



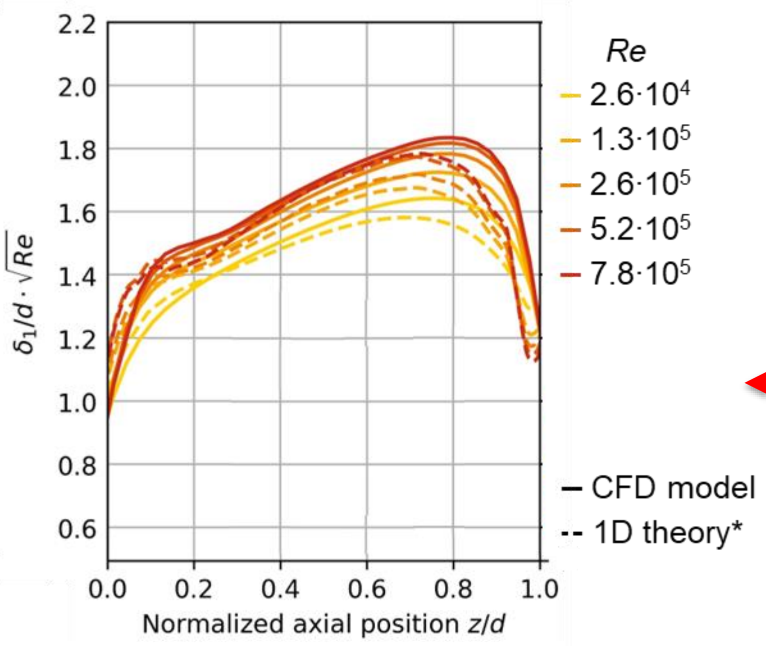
- ○ Standard model in good accordance with experimental data for higher *Re*
- ◇ Transitional model in good agreement in entire *Re* range

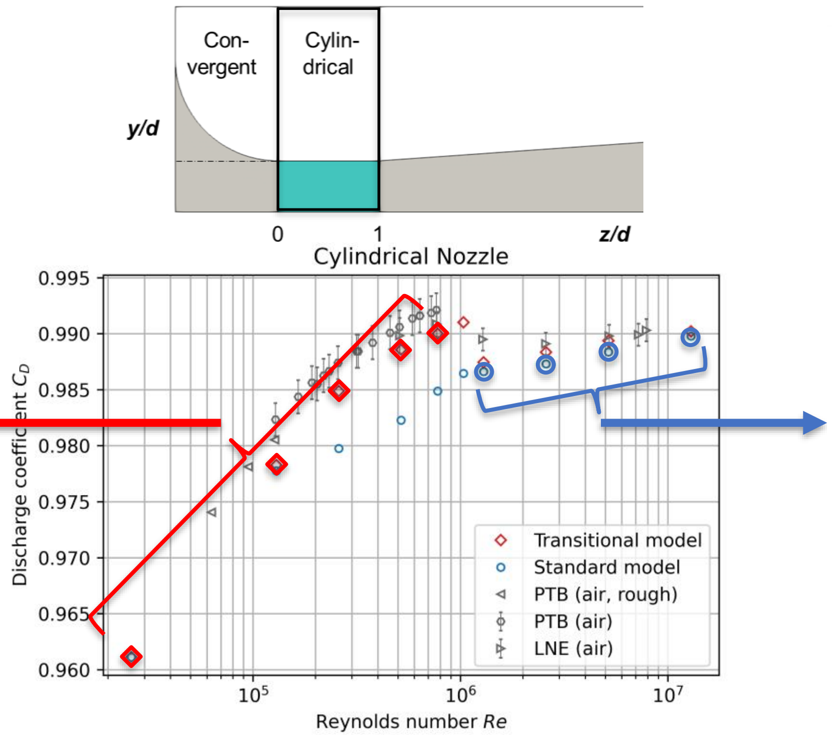
Displacement thickness

Definition

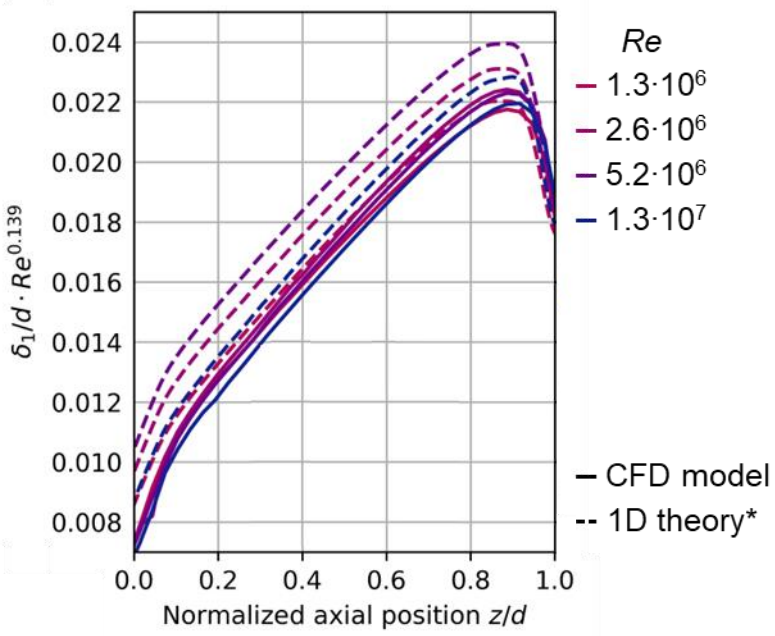
$$\delta_1 = \int_0^{\delta_{max}} \left(1 - \frac{\rho U}{(\rho U)_{max}}\right) dy$$

Laminar region





Turbulent region

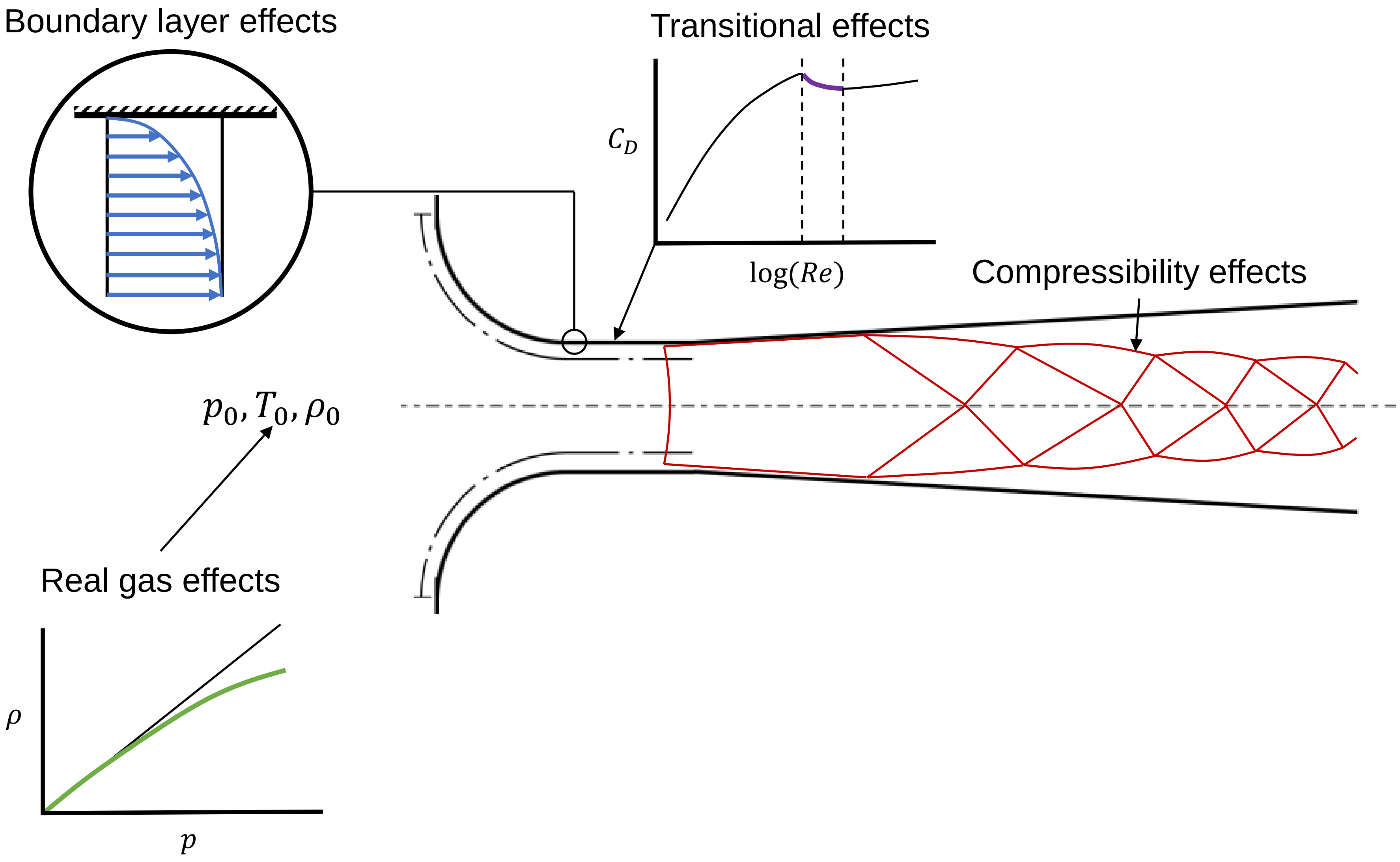


Self-similar for $\sqrt{Re}$

* Based on integral methods for solving the boundary layer equations (by B. Mickan)

Self-similar for $Re^{0.139}$

Real gas effects



Real gas effects

Motivation

What is the difference between a real and an ideal gas?	
Ideal gas Particles have no volume Collisions are elastic No interactions between particles	Real gas Particles have volume Collisions are non-elastic Intermolecular forces
Real gases behave like ideal gases: At high temperatures At low pressures	

- Nozzle flow for pressures up to 100 MPa needs to be considered
- ➢ Real gas effects are relevant and cannot be neglected
- ➢ Consideration of appropriate real gas model in CFD simulation

	Ideal gas model	Real gas model (e. g. Peng-Robinson)
Equation of State (EoS)	$p \cdot V_m = R \cdot T$	$\left(p + \underbrace{\frac{\alpha \cdot a}{V_m^2 + 2b \cdot V_m - b^2}}_{\text{allows for intermolecular forces}}\right) \cdot (V_m - \underbrace{b}_{\text{allows for effect of particle volume}}) = R \cdot T$ ➢ already available in OpenFOAM

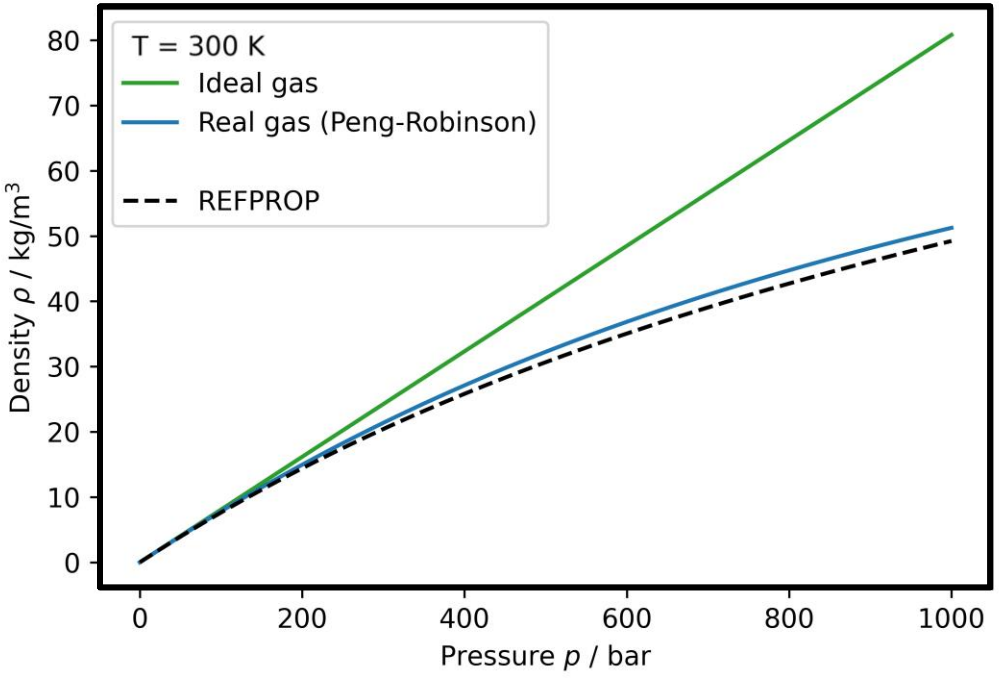
Real gas effects

Real gas model

- High deviation in density at high pressures between
  - Ideal gas (–) and
  - real gas (**---**)
- Real gas model of Peng & Robinson (–) is available in OpenFOAM
  - However, deviation is still up to 5 % from REFPROP

➢ A more precise real gas model for hydrogen is implemented based on REFPROP data

Hydrogen



Real gas effects

New model

Full temperature range

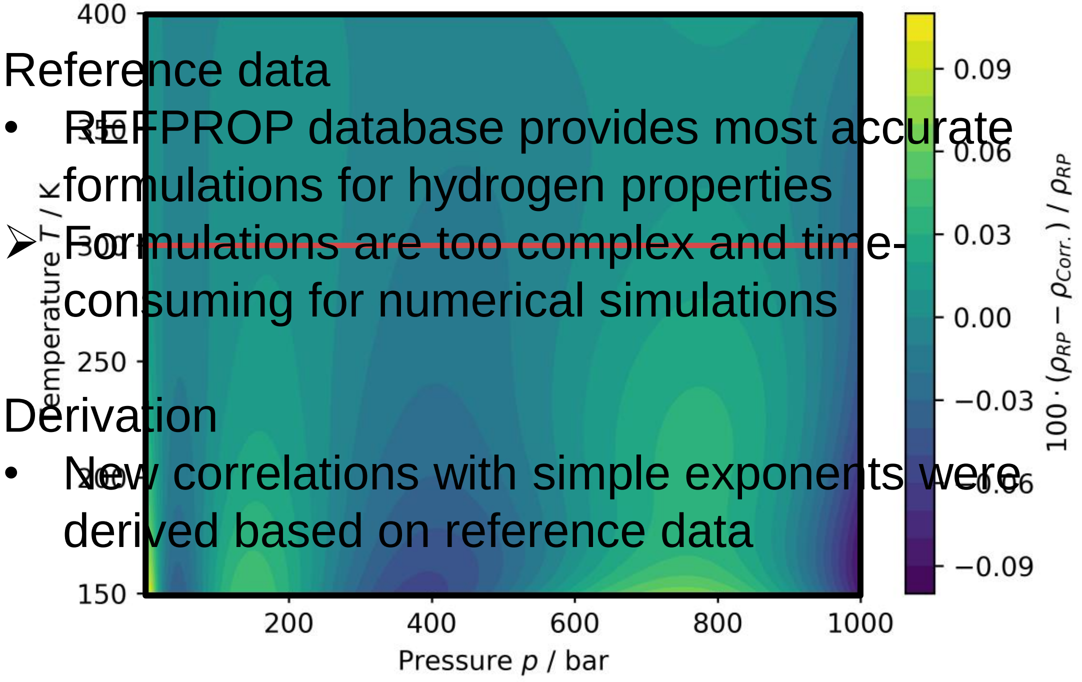
Reference data

- REFPROP database provides most accurate formulations for hydrogen properties

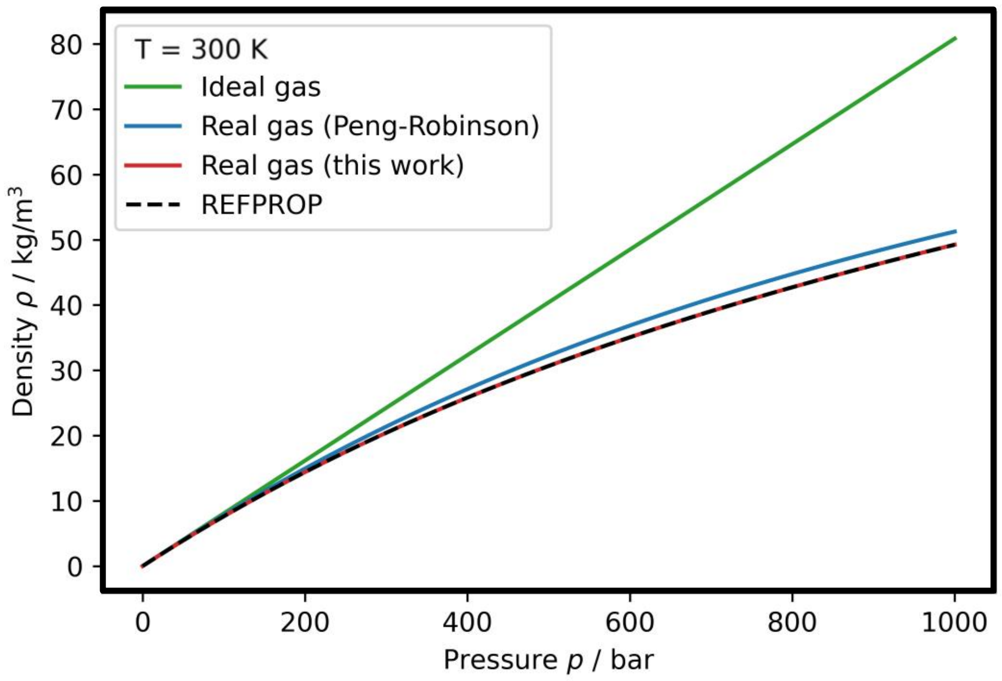
➢ Formulations are too complex and time-consuming for numerical simulations

Derivation

- New correlations with simple exponents were derived based on reference data



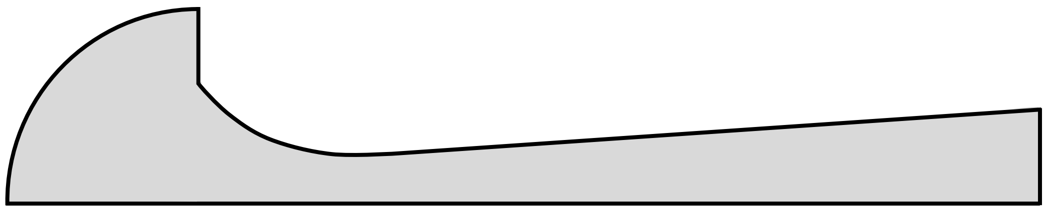
Hydrogen



➢ Error is decreased from 5 % (–) to 0.1 % (–)

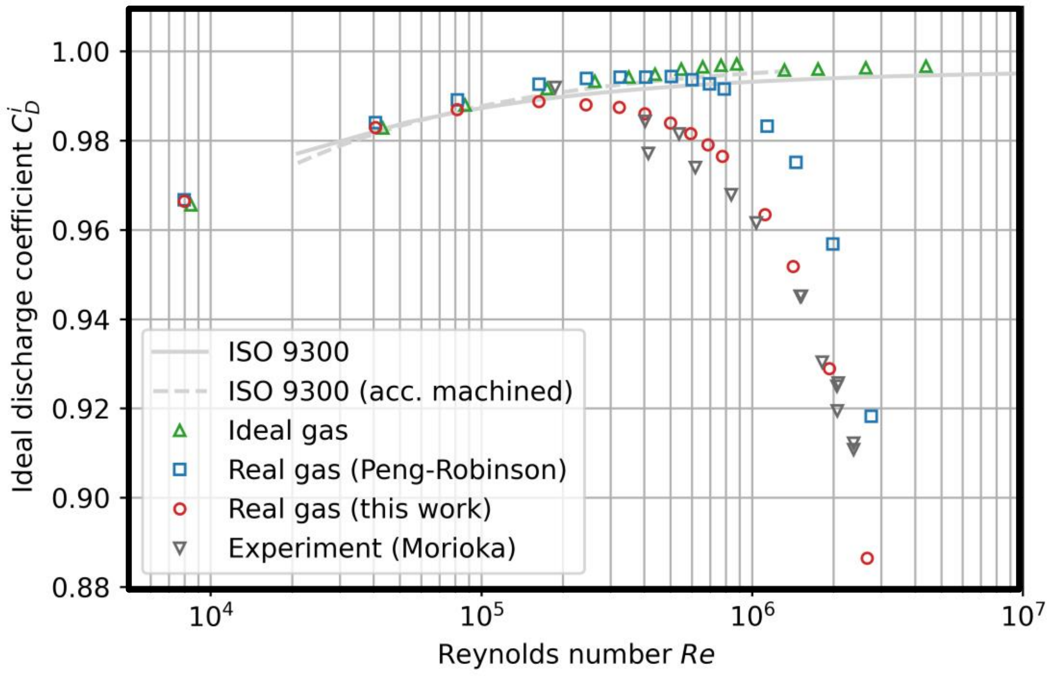
Real gas effects

Validation



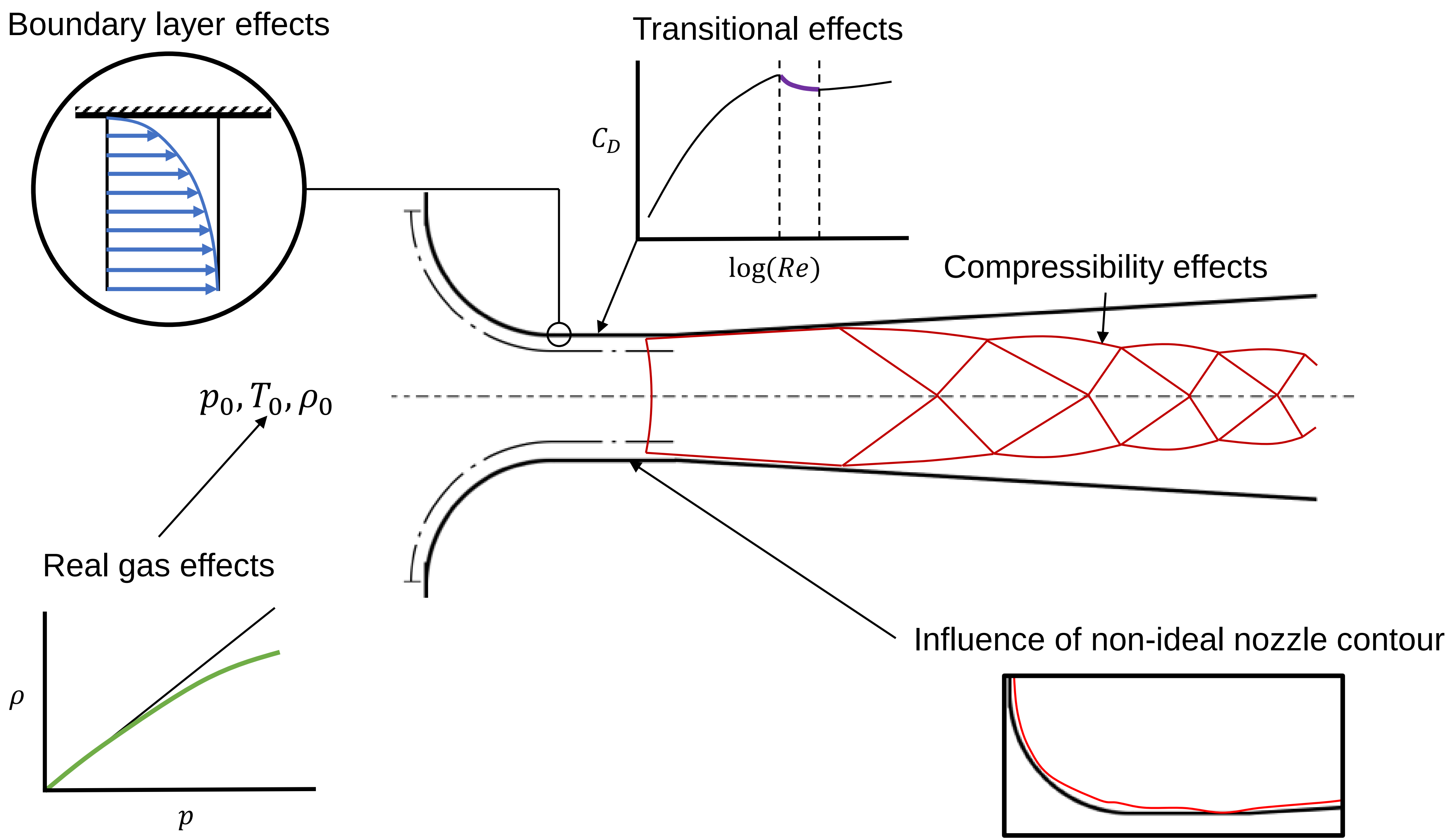
- Validation simulations of hydrogen nozzle flow with
  - △ Ideal gas model
  - □ Real gas model by Peng & Robinson
  - ○ Real gas model from this work
- compared with
  - ▽ Experiments by Morioka et al. (2011)

Toroidal nozzle (d = 0.6 mm, hydrogen flow)



➢ New implemented real gas model is in good accordance with experimental data

Non-ideal nozzle contours



Non-ideal nozzle contours

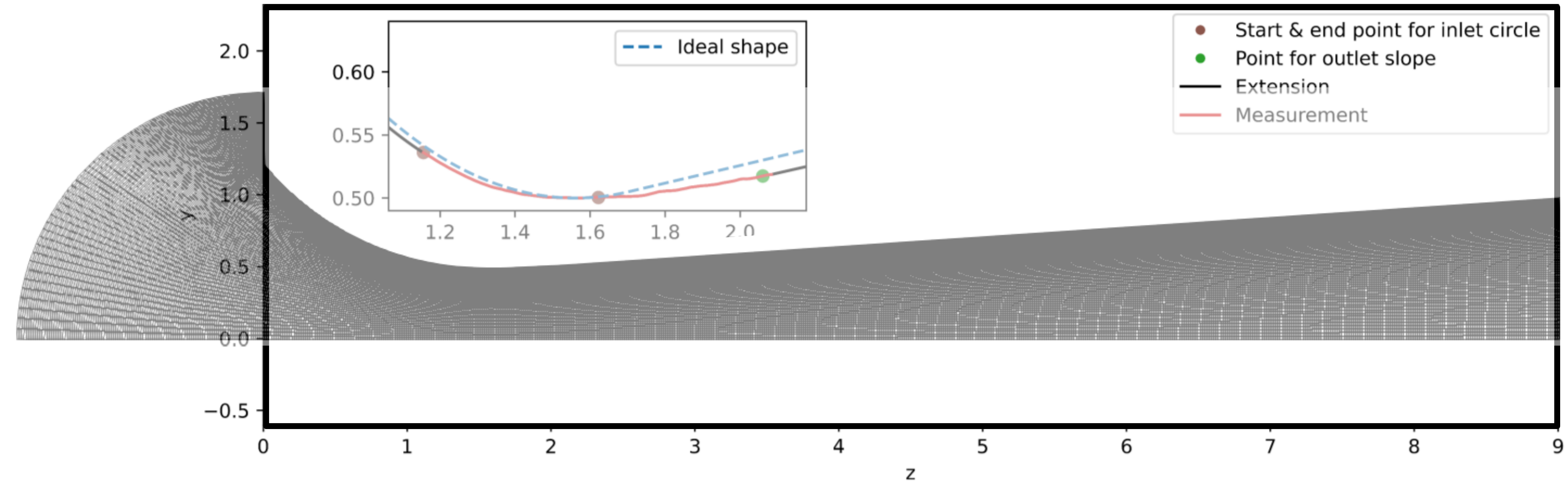
Geometry creation

- Measured nozzle contour (–) is extended (–) with
  - Circle at inlet
  - Straight line at outlet

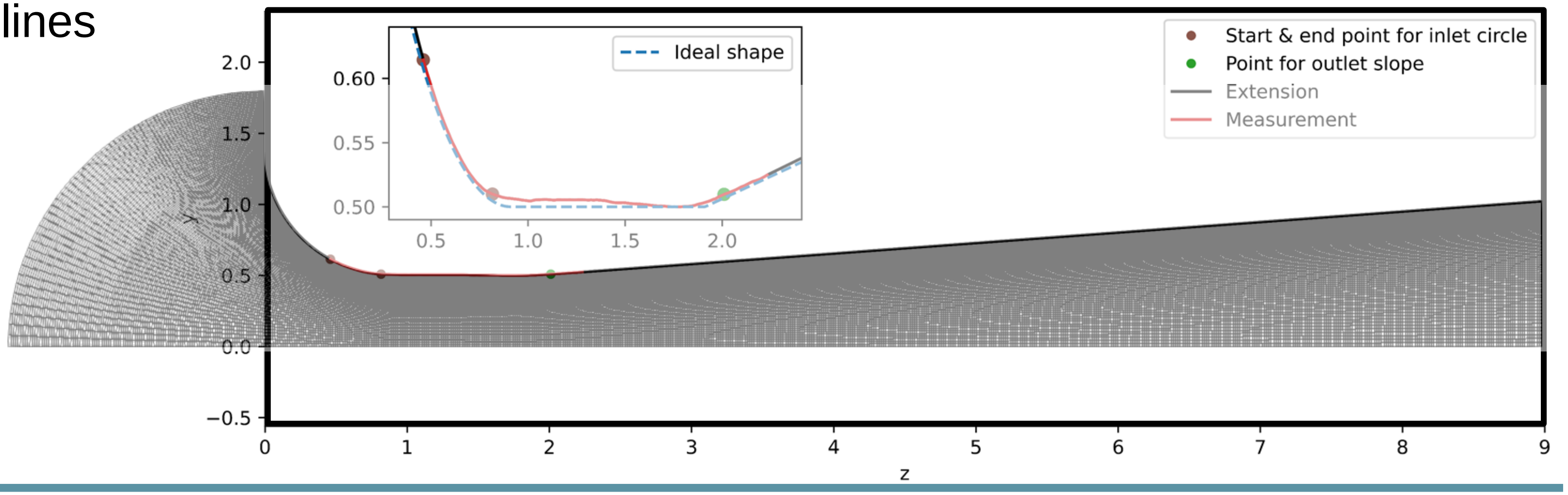
Mesh creation

- Hexaedral mesh
- Contour is implemented using B-splines
- Mesh can be created for
  - Ideal contours
  - Measured contours
  - Parameterized contours

Toroidal nozzle (d = 1 mm)



Cylindrical nozzle (d = 1 mm)

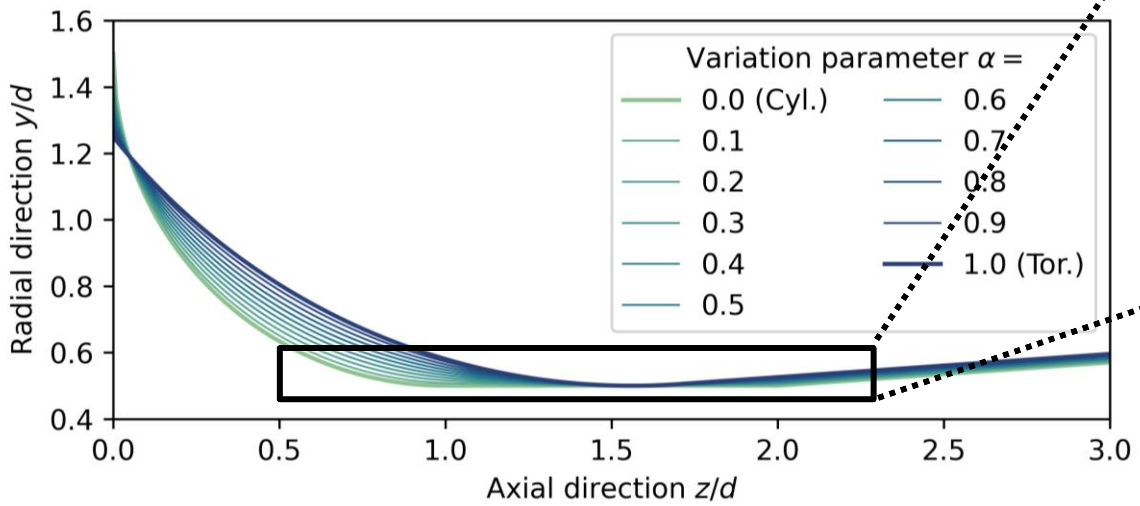


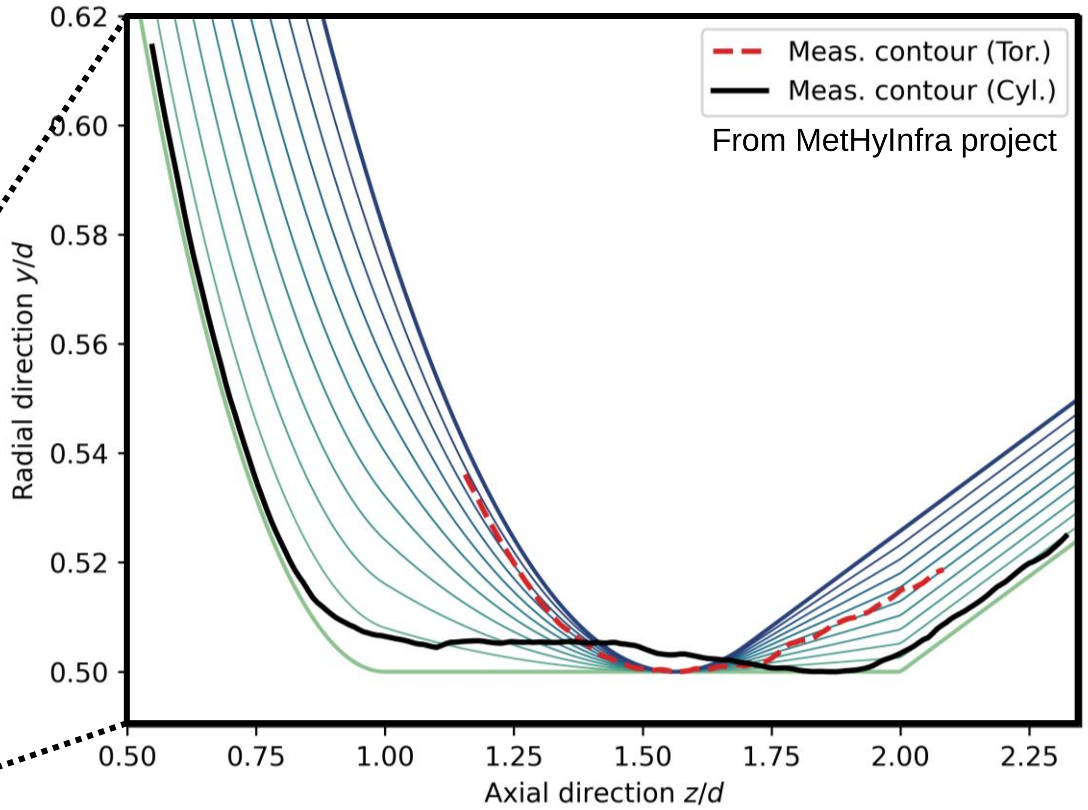
Parametric study

- Non-ideal nozzle shapes often lie between the two ideal shapes (cylindrical and toroidal)

➢ Let's consider a linear superposition of the ideal types via a variation parameter α with
  - $\alpha = 0$ for cylindrical shape
  - $\alpha = 1$ for toroidal nozzle shape
  - $0 < \alpha < 1$ for parametric shape

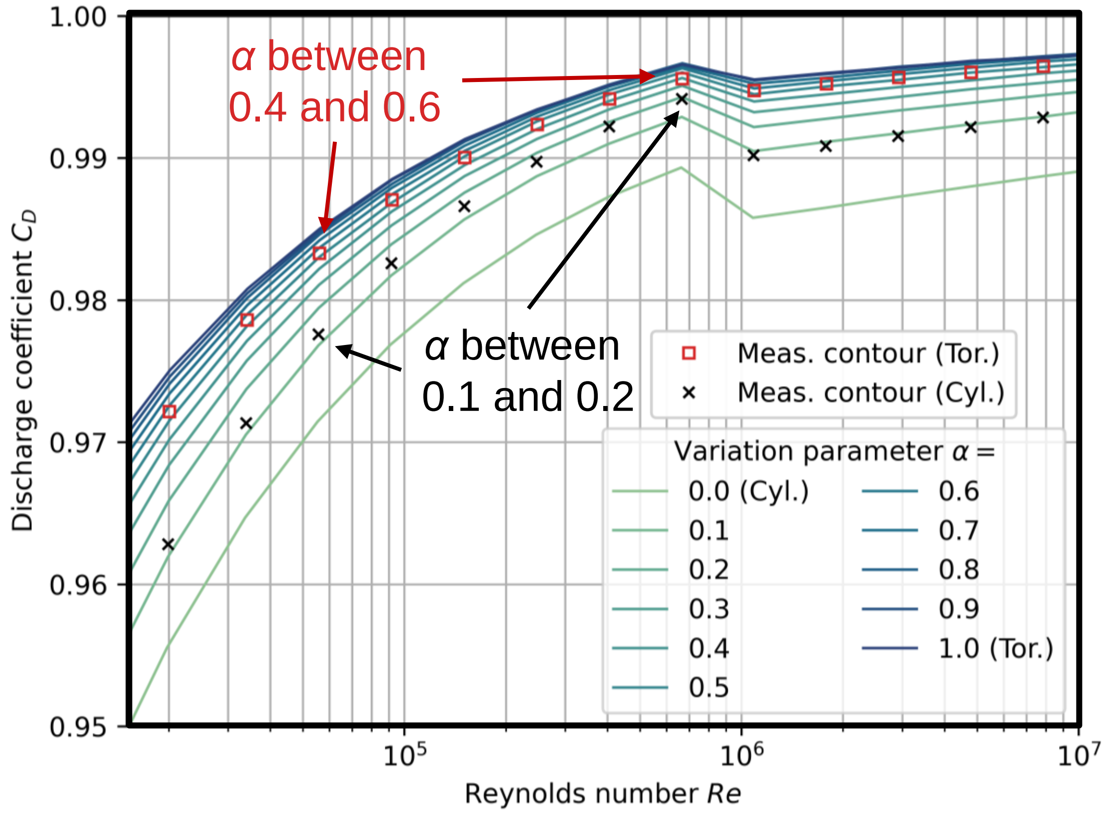
$$y(\alpha) = (1-\alpha)\cdot y_{Cyl} + \alpha \cdot y_{Tor}$$



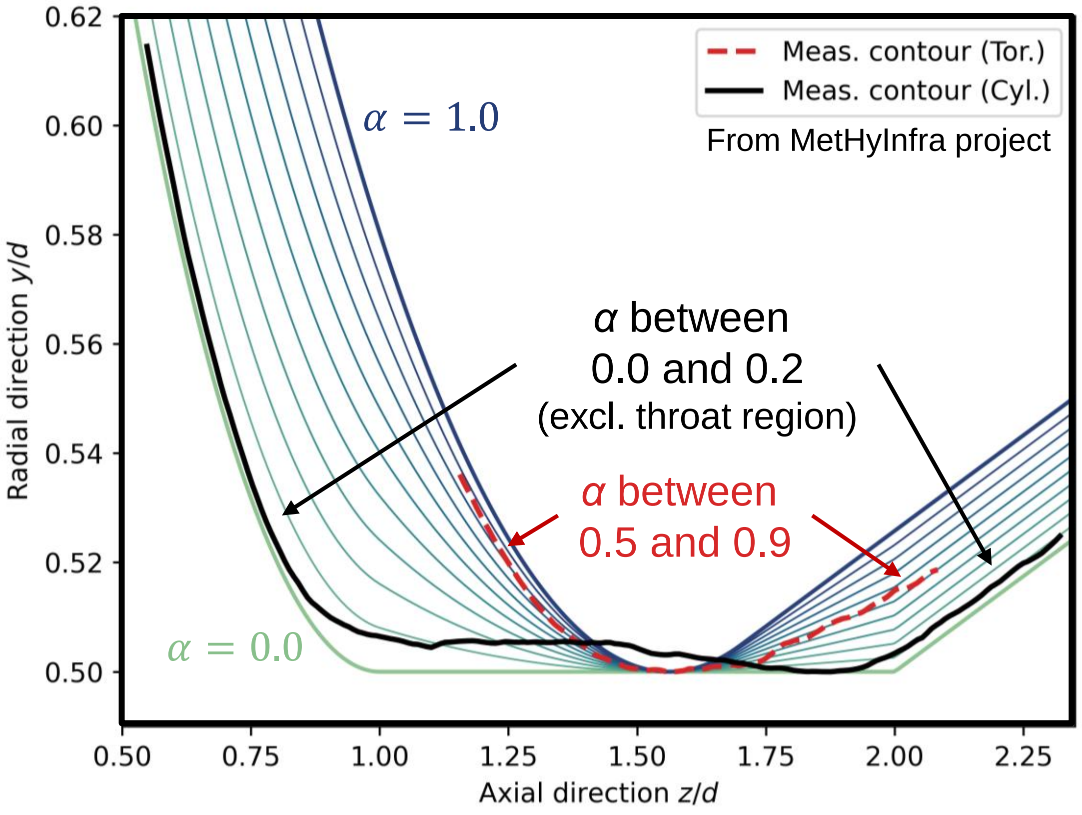


Parametric study

Results

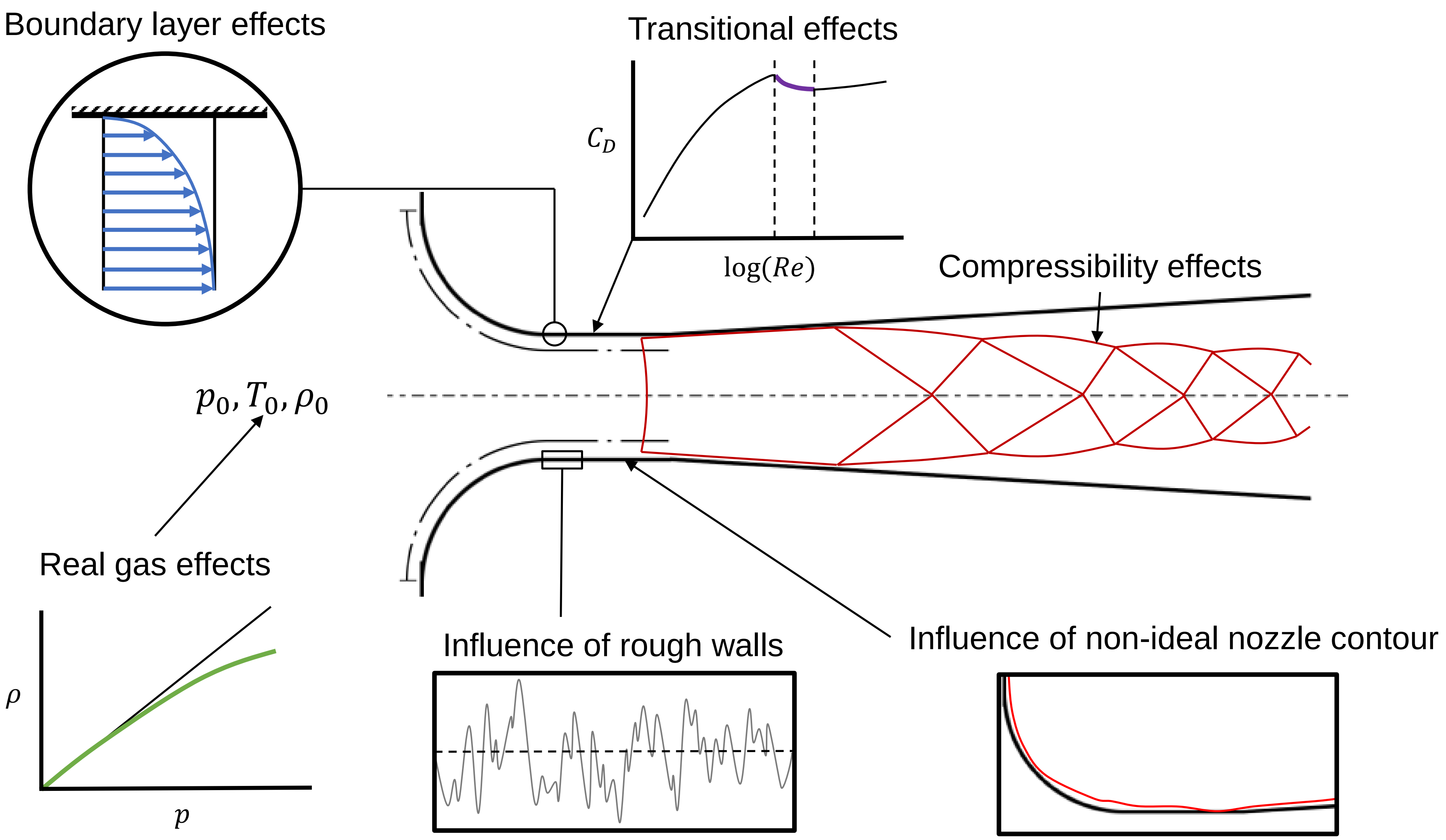


Nozzle contours



- ➢ In a first approximation, allocated α values of nozzle shapes coincide with those of the C_D curves

Wall roughness

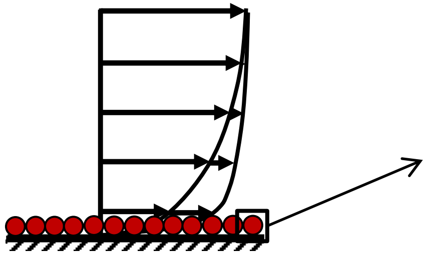


Wall roughness

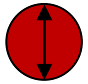
- Logarithmic law of the wall (smooth walls)

$$u^+ = \frac{1}{\kappa}\ln(Ey^+) \qquad 30 < y^+ < 200$$

- What happens for rough walls?



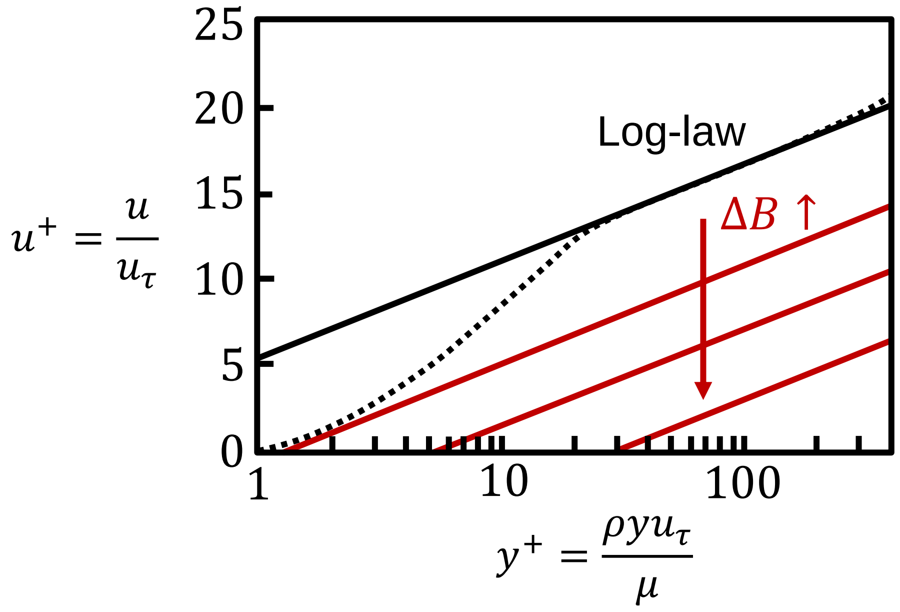
Sand grain roughness height

 K_s $\left(K_s^+ = \frac{\rho K_s u_\tau}{\mu}\right)$

- Sand grains increase wall shear stress and broaden the flow profile

➢ Logarithmic law of the wall (rough walls)

$$u^+ = \frac{1}{\kappa}\ln(Ey^+) - \Delta B$$



➢ ΔB causes the log-law curve to shift downwards

Three regions are distinguished

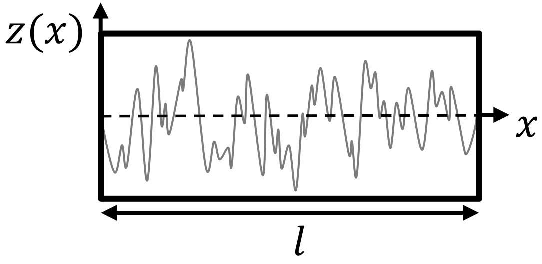
- Hydrodynamically smooth $K_s^+ \leq 2.25$
- Transitionally rough $2.25 < K_s^+ < 90$
- Hydrodynamically rough $90 \leq K_s^+$

Wall roughness

Roughness average *Ra*

- ➢ Arithmetic average of absolute values of profile heights over evaluation length

$$Ra = \frac{1}{l}\int_0^l |z(x)|\,dx$$

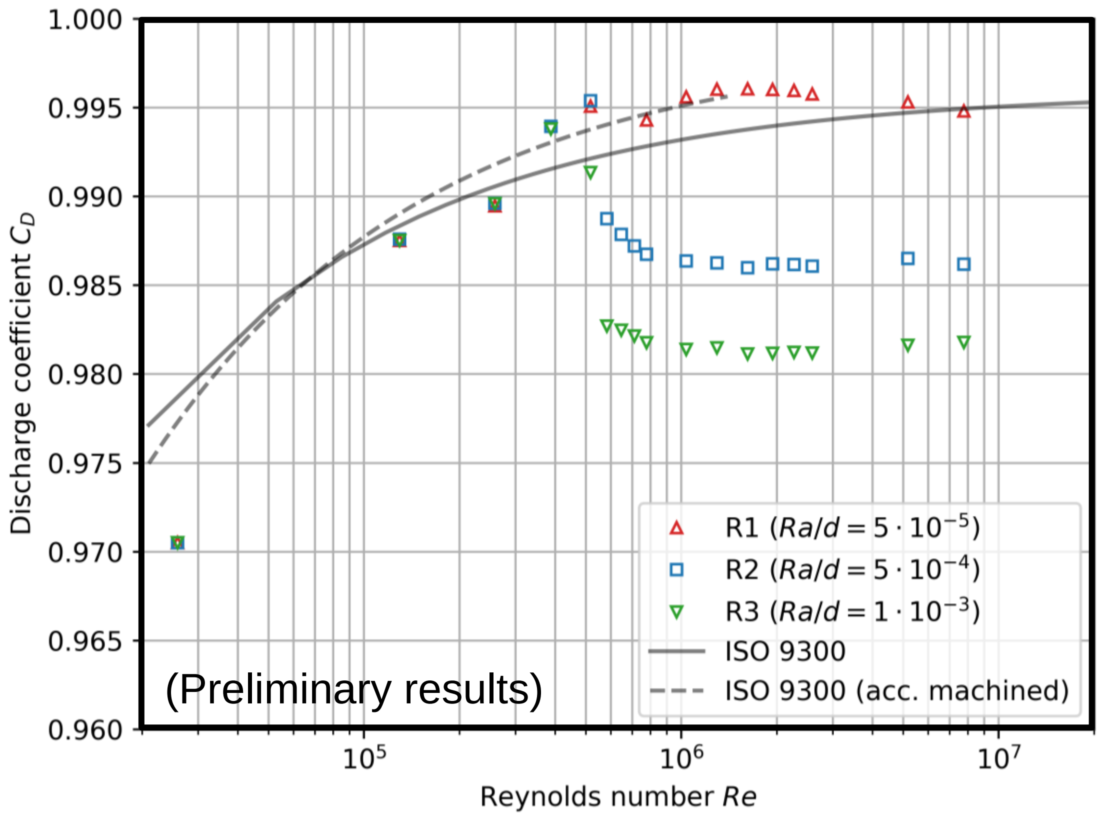


- Correlation between *Ra* and K_s for CFD analysis (based on literature study)

$$K_s \approx 5Ra$$

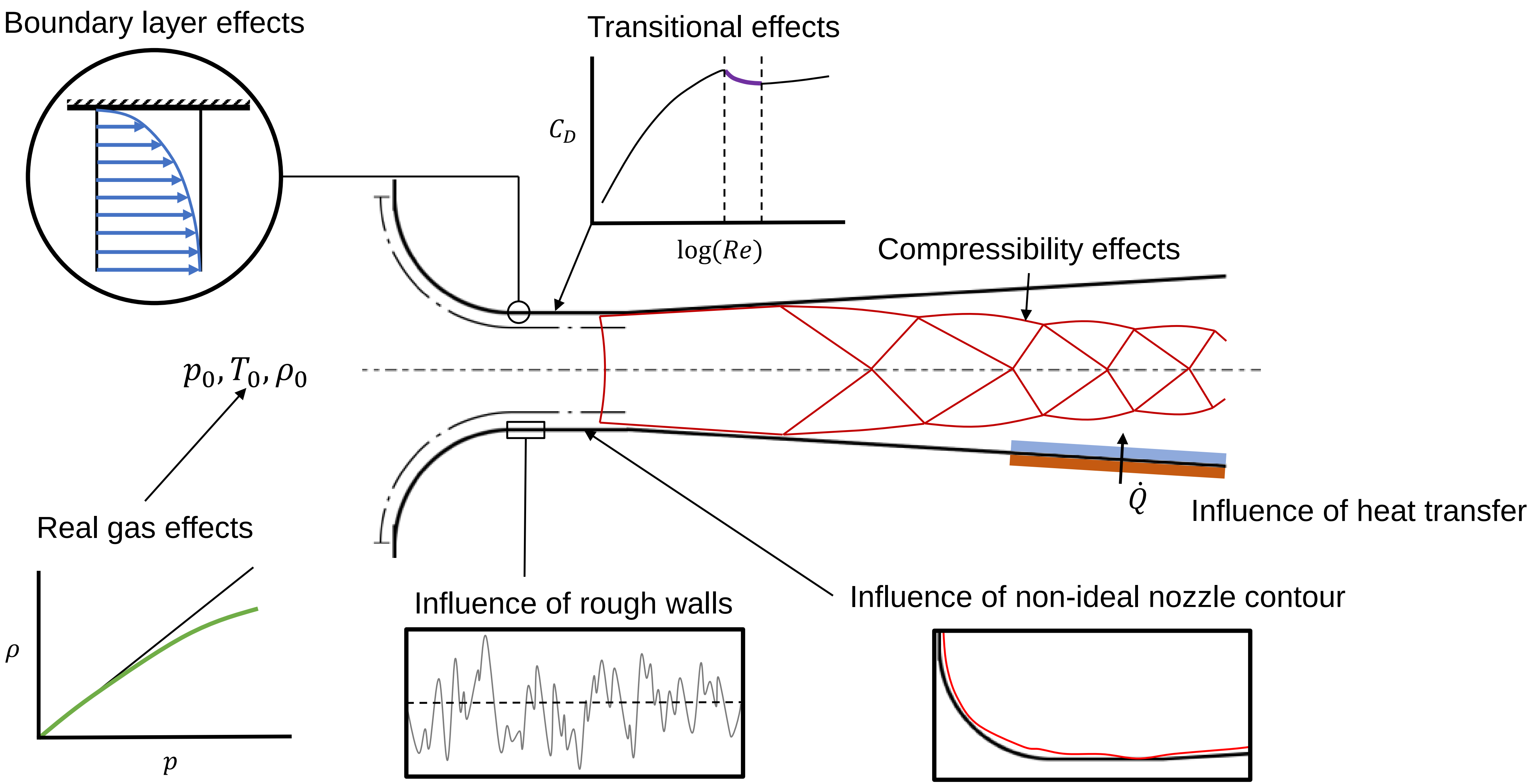
- Investigation of three different roughness levels:
  - △ R1 (*Ra* = 0.05 µm)
  - □ R2 (*Ra* = 0.50 µm)
  - ▽ R3 (*Ra* = 1.00 µm)

Toroidal nozzle
(d = 1.0 mm, hydrogen, ideal gas)



- ➢ Specific drop in turbulent region
- ➢ Transition occurs earlier

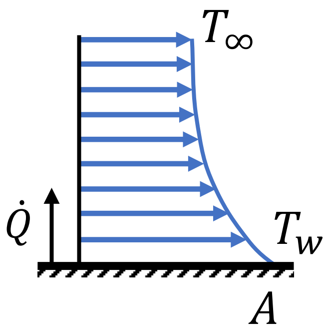
Heat transfer



Heat transfer

Thermal boundary layer (TBL)

- thin region close to wall formed due to temperature gradient between wall and core flow



Heat convection

$$\dot{Q} = hA(T_w - T_\infty)$$

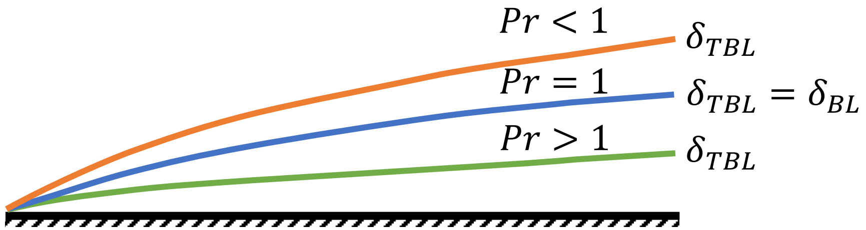
with heat transfer coefficient *h*

- As temperature in-/decreases in TBL, density de-/increases which affects mass flow rate $\left(\rho = \dfrac{p}{R_M T}\right)$

Prandtl number

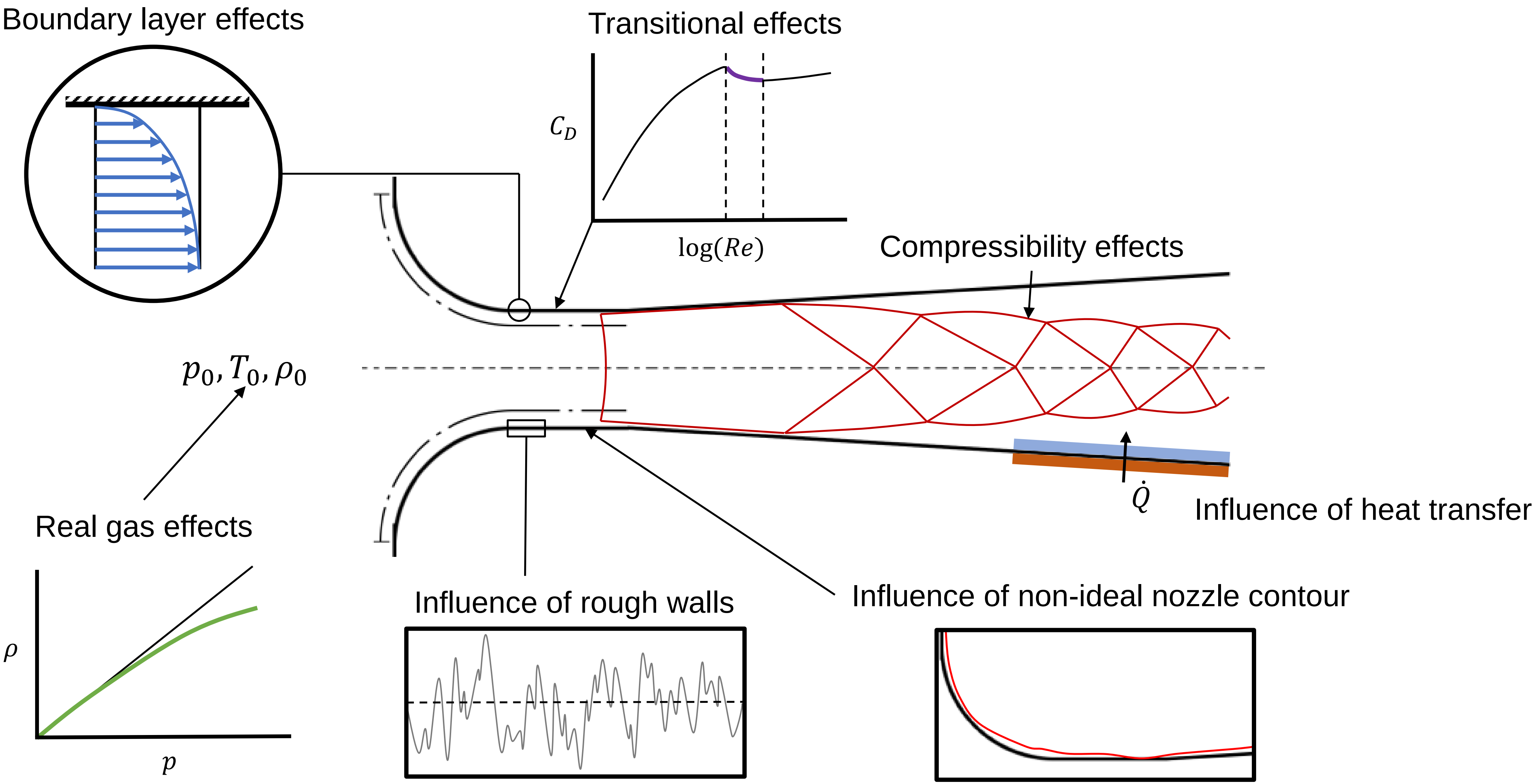
- Ratio of momentum to thermal diffusivity

$$Pr = \frac{\nu}{\alpha} = \frac{\mu/\rho}{k/(c_p\rho)} = \frac{c_p\mu}{k}$$



- For many gases, typically around 0.7

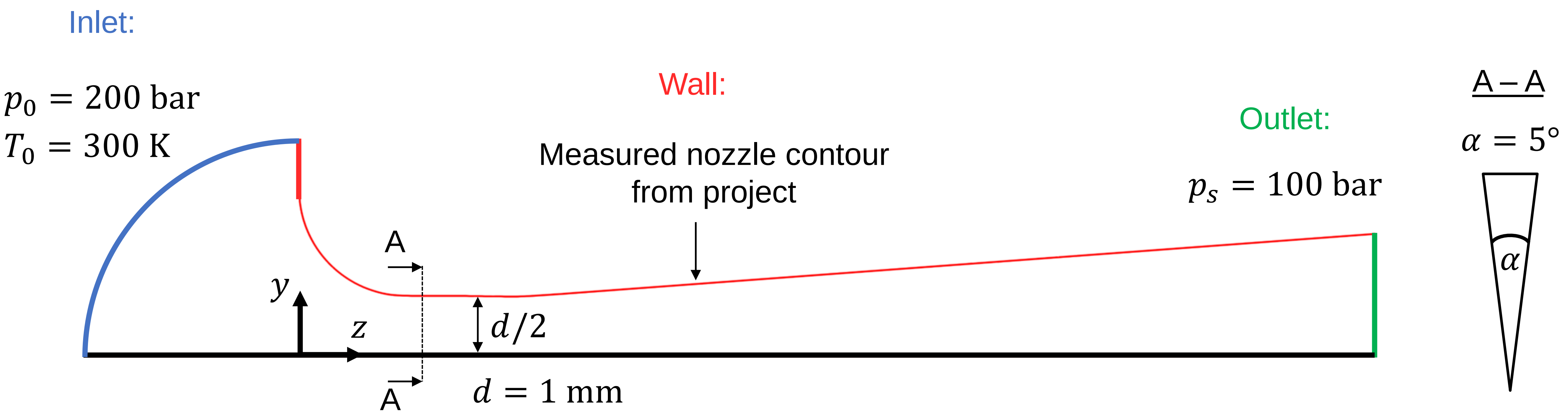
Thank you!



Boundary layer effects
Transitional effects
C_D
$\log(Re)$
Compressibility effects
p_0, T_0, ρ_0
$\dot{Q}$
Influence of heat transfer
Real gas effects
ρ
p
Influence of rough walls
Influence of non-ideal nozzle contour

Next session: Tutorial case

High-pressure hydrogen flow through a cylindrical non-ideal critical nozzle



MetHyInfra

Internal

PTB
Physikalisch-Technische Bundesanstalt
Braunschweig und Berlin

CESAME-exadébit SA

Justervesenet

METAS

RISE
Research Institutes of Sweden

TÜV SÜD
National Engineering Laboratory

VSL
Dutch Metrology Institute

external

THERMODYNAMIK
RUHR UNIVERSITÄT BOCHUM RUB

University *of Ljubljana*
Faculty *of Mechanical Engineering*

esi get it right®
MECAS ESI s.r.o.

Imperial College London

MAXIMATOR®
Maximum Pressure.

DNV

Perfection in fluids.
The right *flow* by German engineering.
ep-e.com

Ingenieurbüro
T. Steuer
its-tech.de

Bronkhorst®
Flow Technology

This project “Metrology infrastructure for high-pressure gas and liquified hydrogen flows” (MetHyInfra) has received funding from the EMPIR programme co-financed by the Participating States and from the European Union’s Horizon 2020 research and innovation programme under Grant agreement No [20IND11] .

EMPIR EURAMET

The EMPIR initiative is co-funded by the European Union's Horizon 2020 research and innovation programme and the EMPIR Participating States

MetHyInfra

Sonja Schmelter and Sebastian Weiss

Working Group 8.41 “Modelling and Simulation” and
Working Group 1.45 “Hydrogen Quantity Metering”
Physikalisch-Technische Bundesanstalt (PTB)
Berlin, Germany

sonja.schmelter@ptb.de and sebastian.weiss@ptb.de