Numerical Simulation of the Performances of Centrifugal Pumps on a Numerical Test Rig

Content

1. Motivation
2. Numerical test rig „IDS“
   - Geometric Set, Preprocessing and grid generation
3. Simulation approaches
   - Level of detail
   - Fluid flow modeling

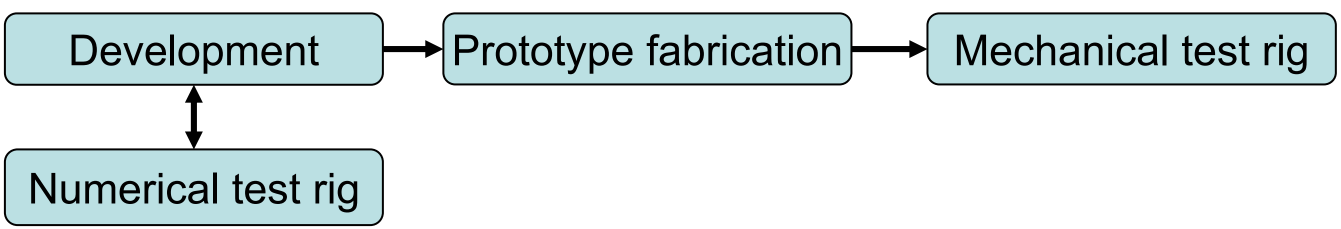
Motivation

- FLM develops design, simulation and optimization tools to support the development process of turbo machinery
- Aim of the work performed was to prove the fitness and reliability of these tools as a numerical test rig.
  - given pump geometry of a specific user
  - “blind test”

- Introduce CFD simulation tools into the design process.

“classical” design approach

CAE-based design approach



IDS

IDS Integrated **D**esign **S**ystem

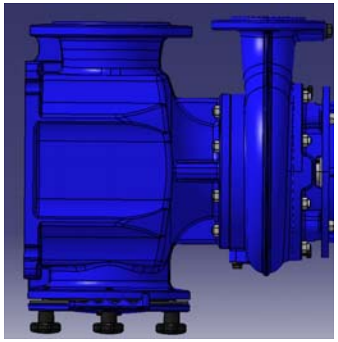
Features:

-Geometry tools
-“automatic” grid generator
-Several solvers
-“single click” post processing unit
-Automatic run system
-Block structured grids are needed

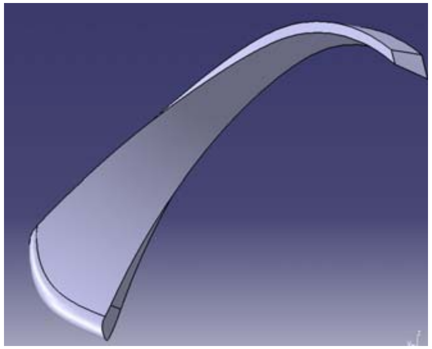
Needed inputs:

-Geometric data from the three main parts
-Physical conditions from the fluid
-Design point data from the pump

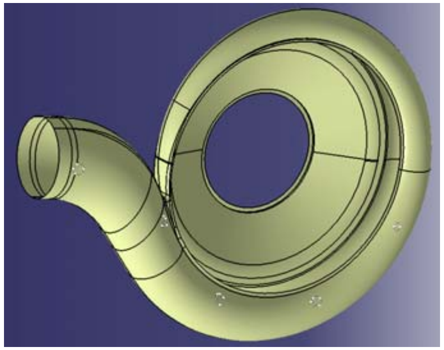
The three main parts



Casing



Runner blade

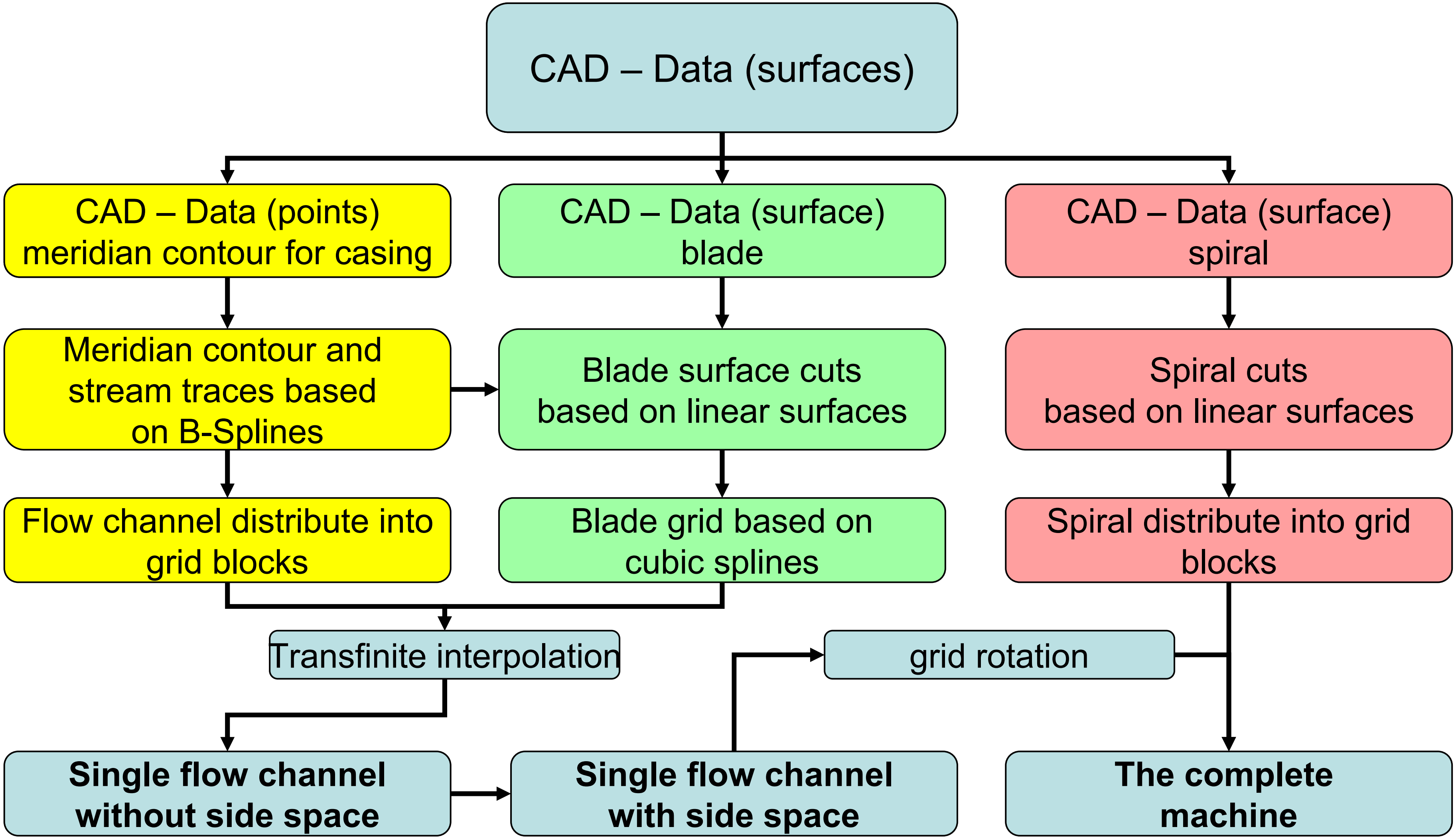


Spiral

Content

1. Motivation
2. Numerical test rig „IDS“
   - Geometric Set, Preprocessing and grid generation
3. Simulation approaches
   - Level of detail
   - Fluid flow modeling

Geometric data



CAD – Data from the casing (surfaces)

Requirements:

-z – axis in flow direction

-rotation symmetric geometries

-absolute coordinates for every part

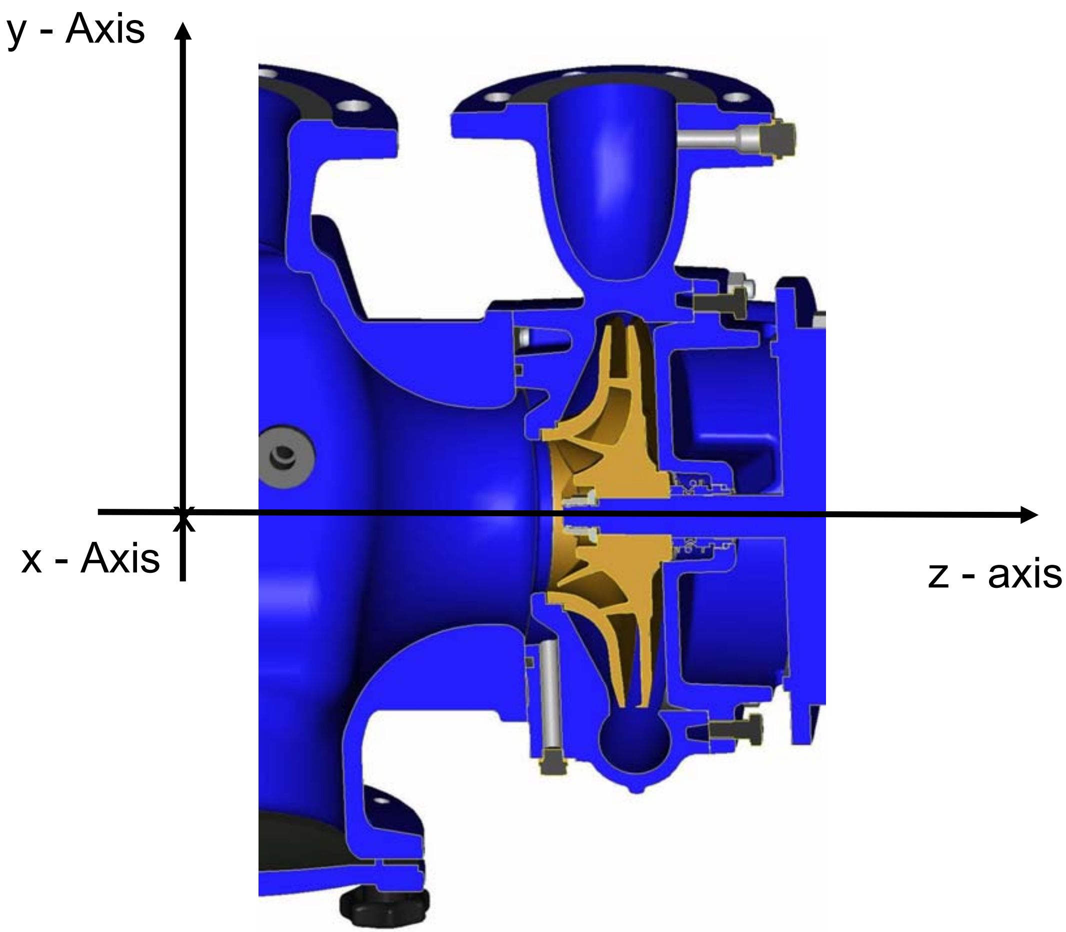
-rotation axis = z – axis

-measurement is meter

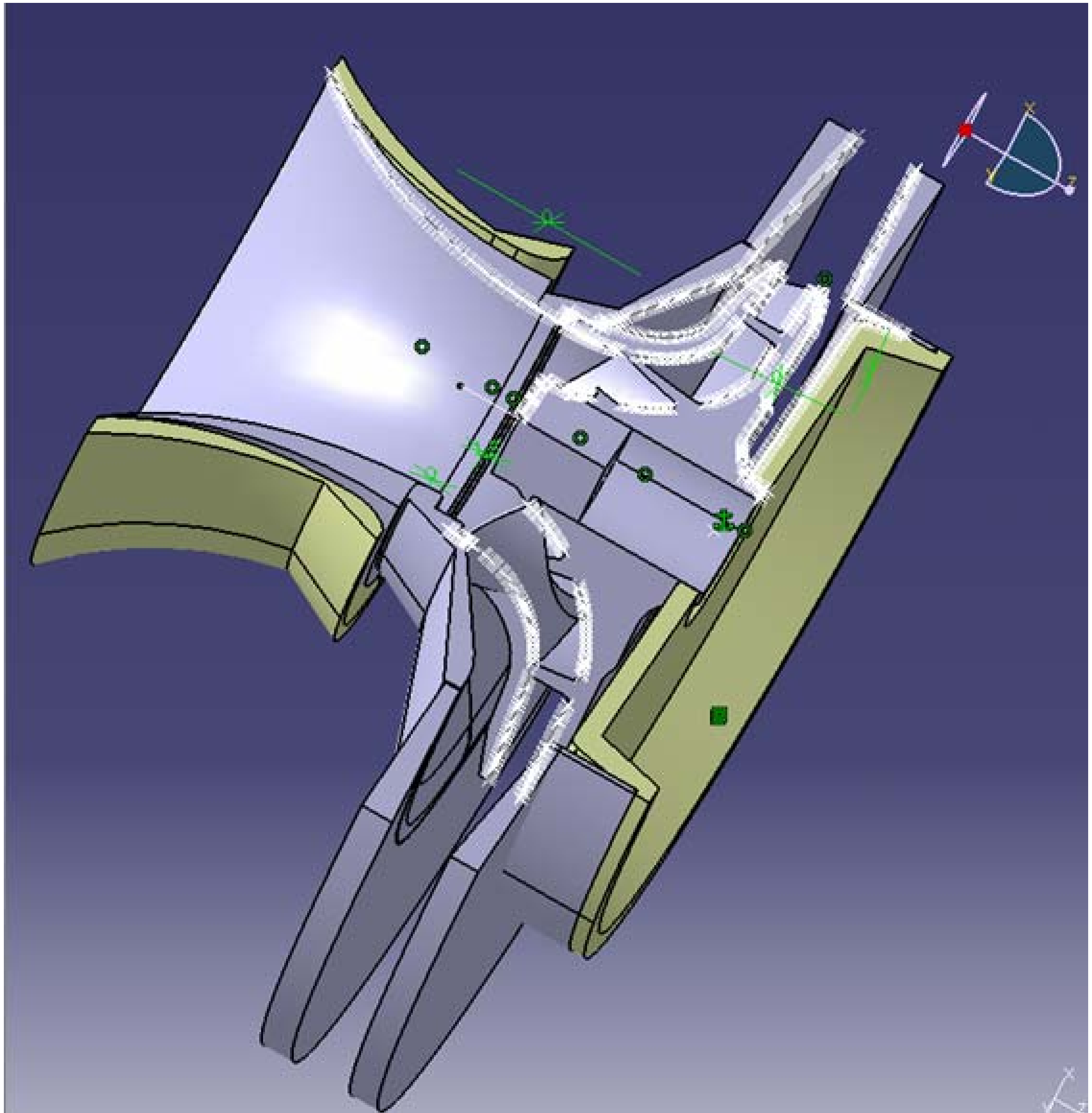
-distribute points along the contour

-right hand system

-CAD – Data file must be an *.igs - file



CATIA V5 drawing



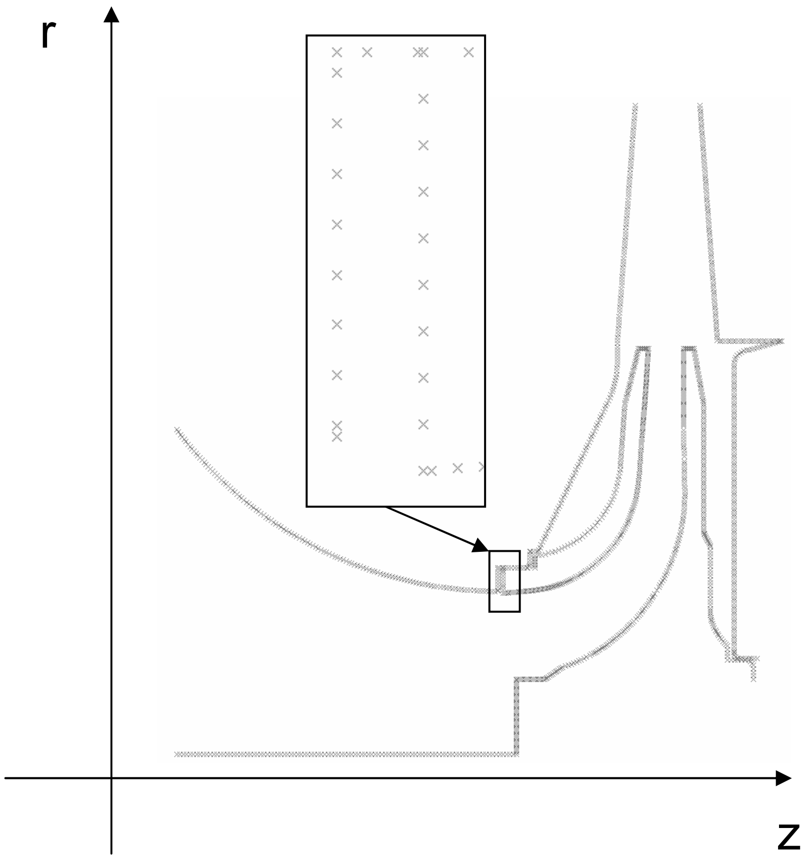
CAD – Data meridian contour (points)

The points from CAD - data file
will be transformed from (x, y, z) – coordinates
into cylinder coordinates with the following rule

$$r = \sqrt{x^2 + y^2}$$

$$z = z$$

→every point has an own coordinate (r, z)



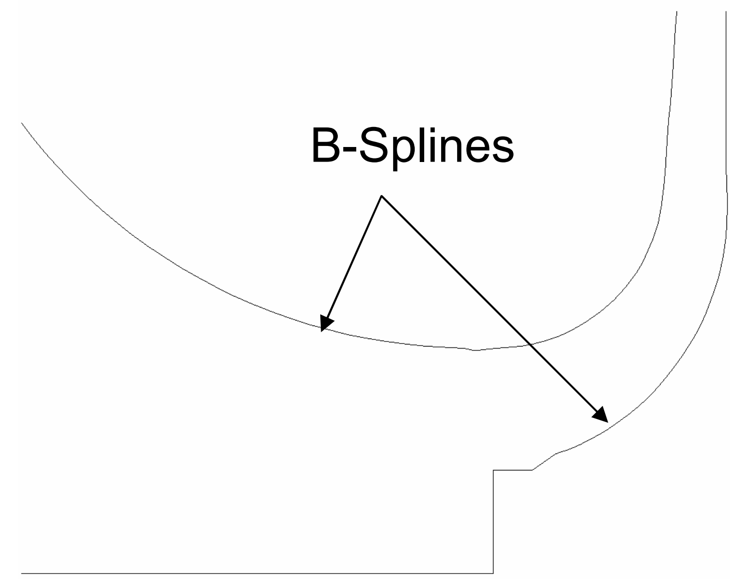
Meridian contour and stream traces based on B-Splines

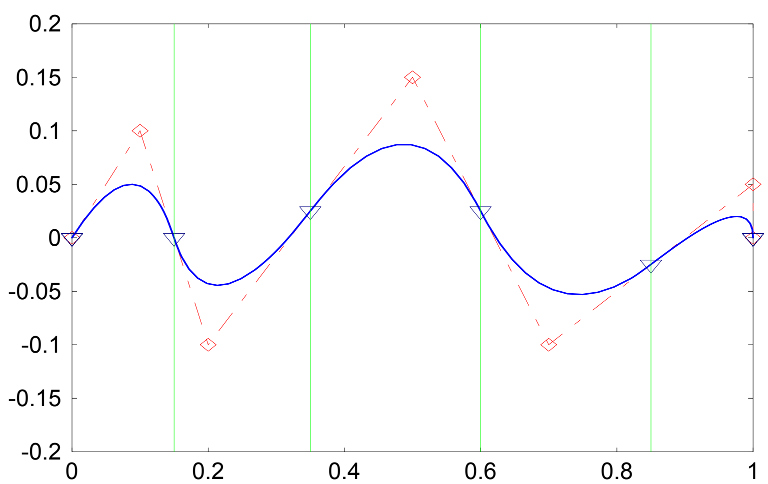
Uniform B-Spline:

order	k = 3
Describer points	n = 7
⇨support vector elements	l = n + k = 10
⇨inner knots	IK = l – 2k = 4
⇨sections	p = IK + 1 = 5

Support vector T:

T = T(0, 0, 0, 0.2, 0.4, 0.6, 0.8, 1.0, 1.0, 1.0)





Recursive Formula

$$N_{i,k}(t) = \frac{(t - t_i)N_{i,k-1}(t)}{t_{i+k-1} - t_i} + \frac{(t_{i+k} - t)N_{i+1,k-1}(t)}{t_{i+k} - t_{i+1}}$$

$$N_{i,1}(t) = \begin{cases} 1: & \text{für } t_i \leq t < t_{i+1} \\ 0: & \text{sonst} \end{cases}$$

t_i: Knot, elements of the support vector T
t: Curve parameter

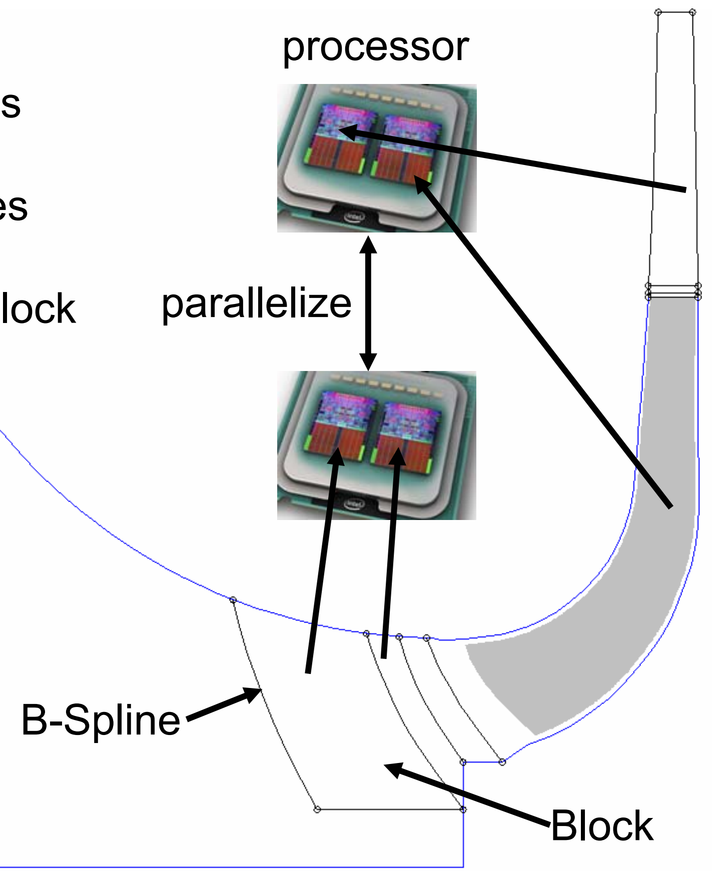
Flow channel distribute into grid blocks

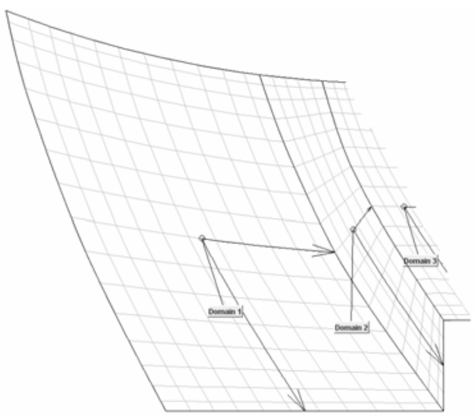
<u>Grid – blocks:</u>

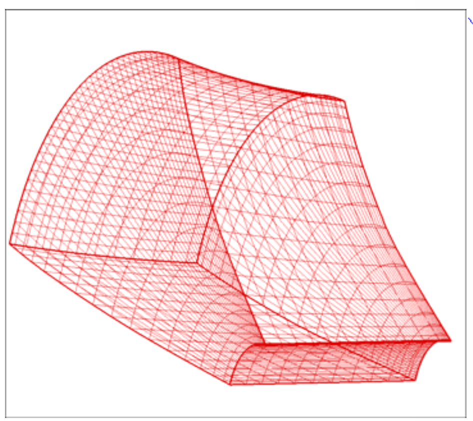
-distribute the flow channel into calculation areas

-block boundaries consists of B-Splines and lines

-grid size can be adapted individually to every block







Grid generation with transfinite interpolation

CAD – Data (surface) Blade

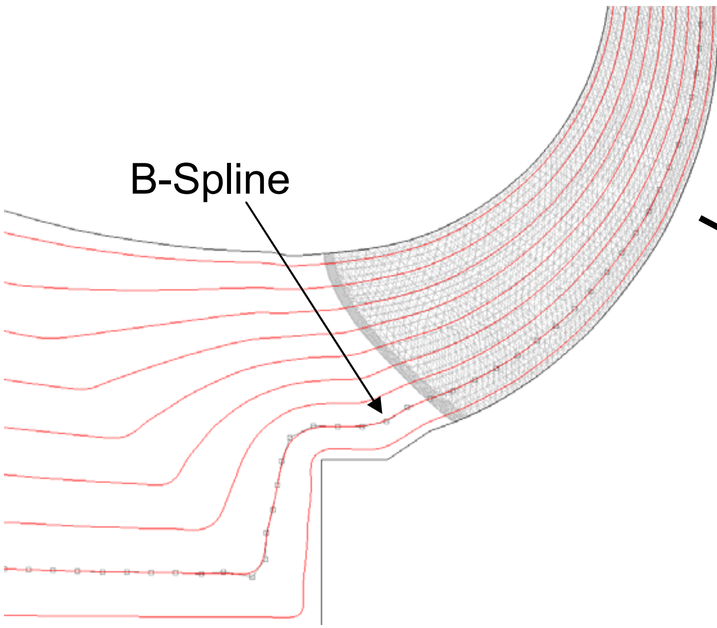
Requirements:

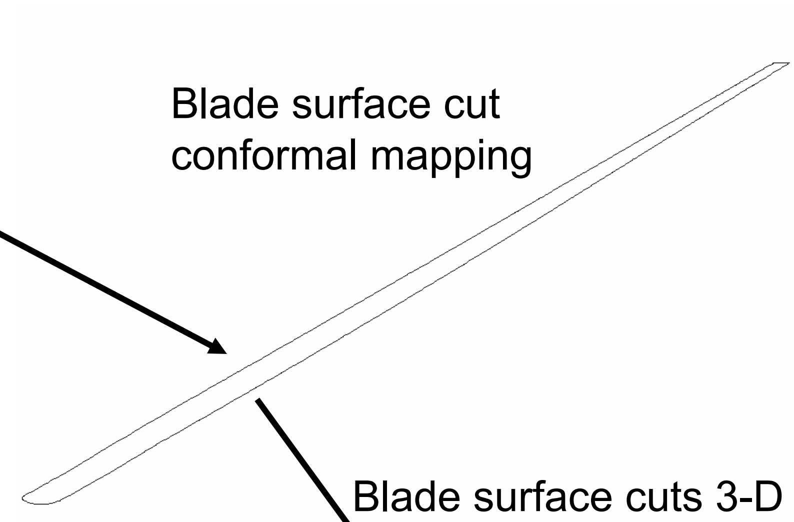
-positive rotation around the z – axis

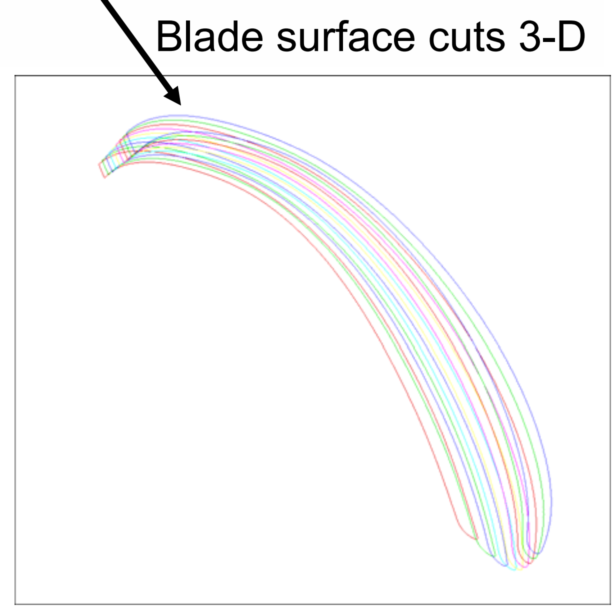
-the positioning coordinates must fits
into the meridian contour

-CAD – Data file must be a *.stl – file

Blade surface cuts based on linear surfaces







stream traces consists of B–Splines

⇩

blade were cut with stream traces

⇩

blade surface cuts based on linear surfaces

⇩

blade is cut into slices

Blade grid based on cubic splines

Interpolating the cuts with cubic splines

Cubic parabola

$$f_i(x) = a_i x^3 + b_i x^2 + c_i x + d_i$$

i: Section number

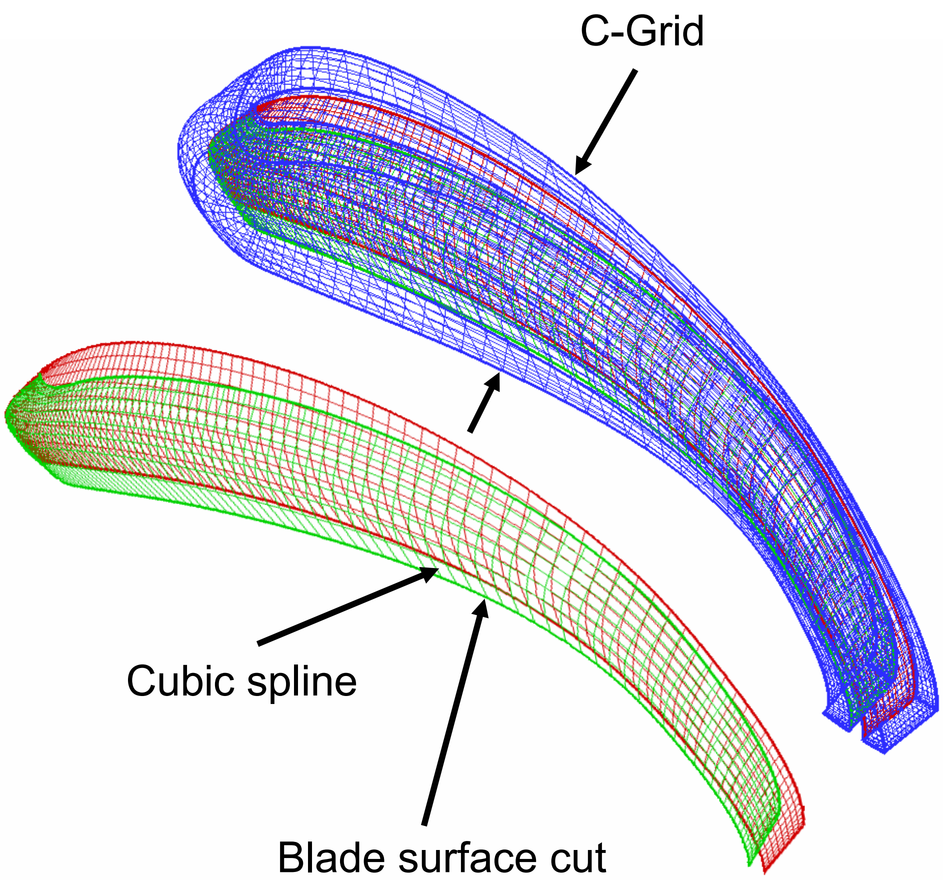
Number of unknown = 4 * number of sections

Number of Nodes = number of sections + 1

$$f_1(x_k) = f_2(x_k)$$

$$\frac{dy_1}{dx_k} = \frac{dy_2}{dx_k}$$

$$\frac{d^2 y_1}{dx_k^2} = \frac{d^2 y_2}{dx_k^2}$$



CAD – Data (surface) spiral

Requirements:

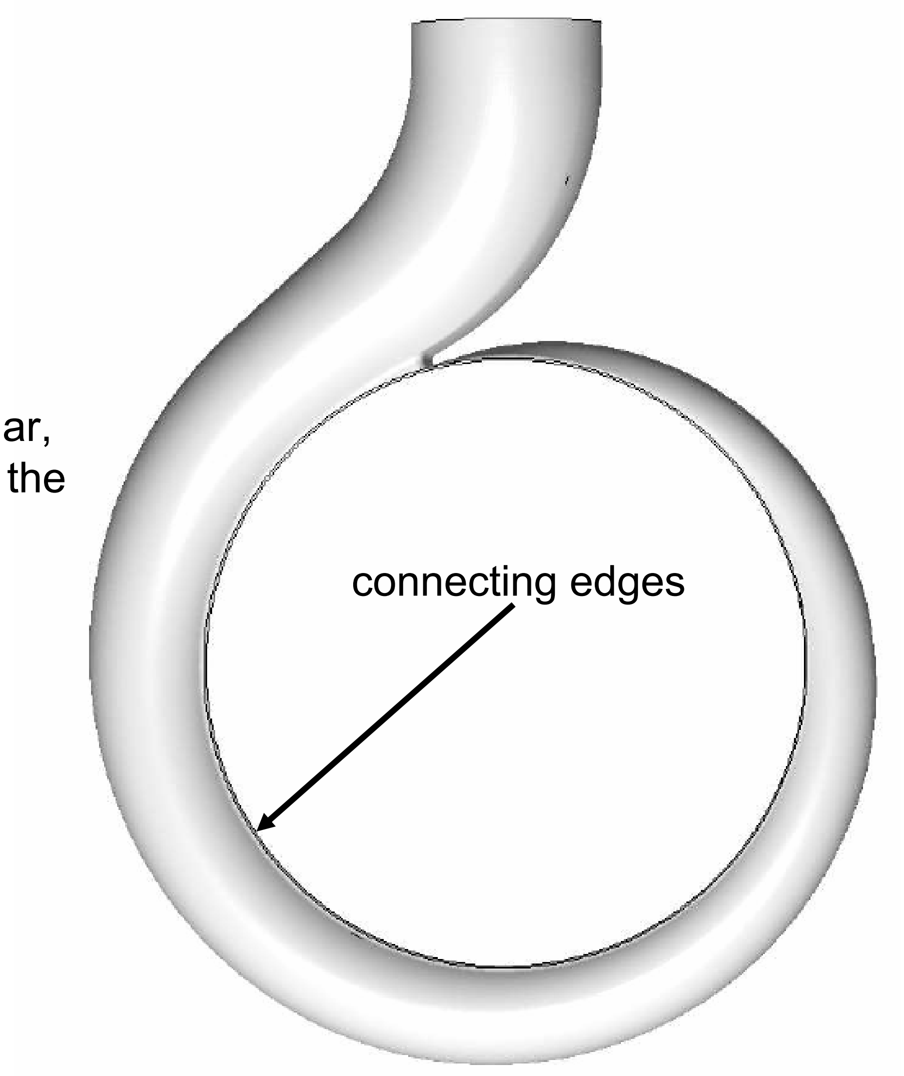
-positive rotation around the z – axis

-the positioning coordinates must fit to the exhaust from the pump

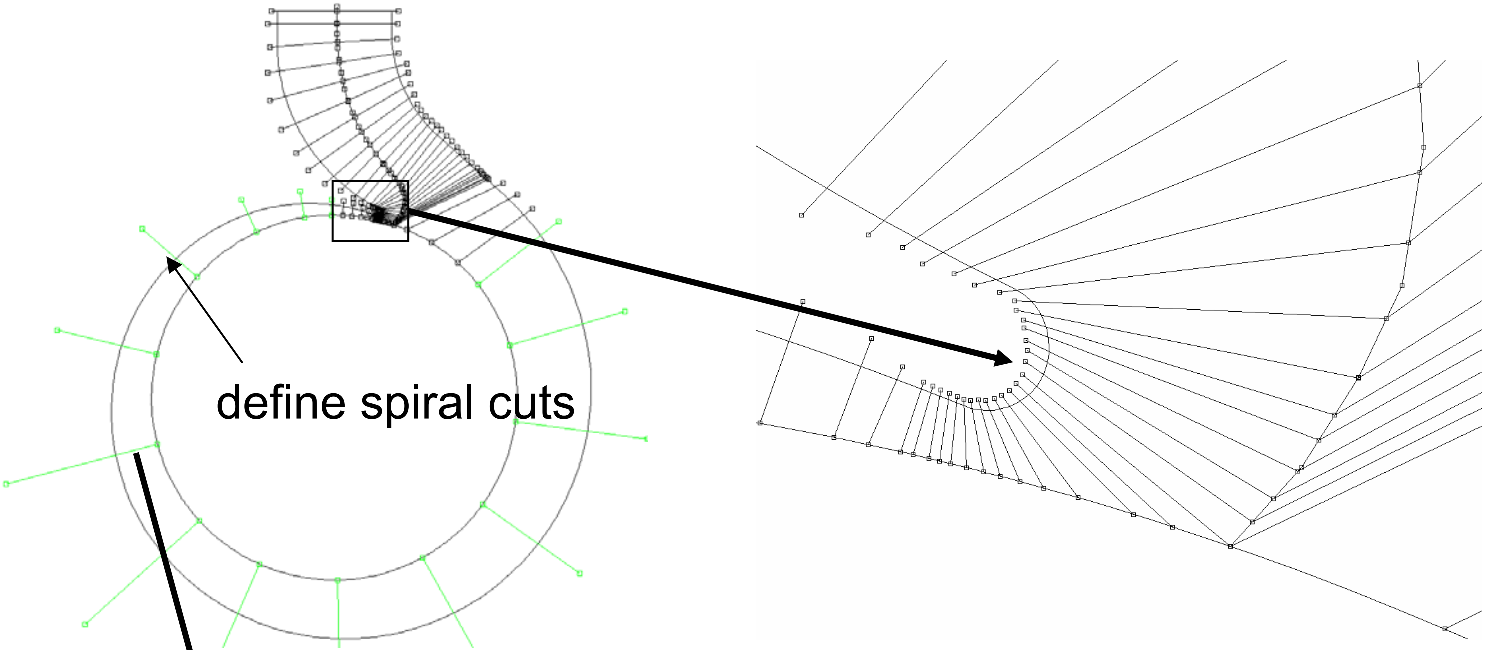
-the two connecting edges must be planar, parallel to each other and concentric to the origin

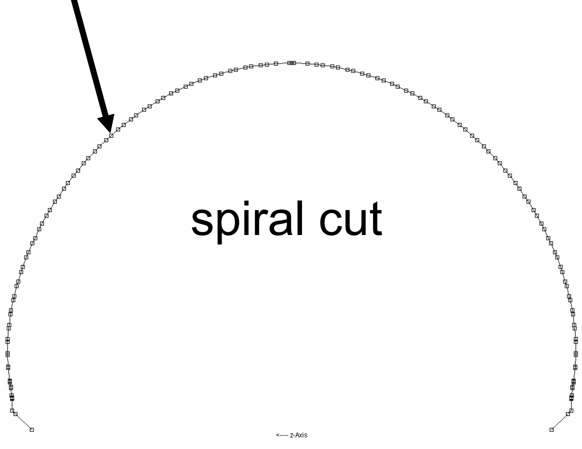
-the measurement is meter

-CAD – Data file must be a *.stl – file



Spiral cuts
based on linear surfaces





define cuts on the spiral

⇩

cut spiral with defined cuts

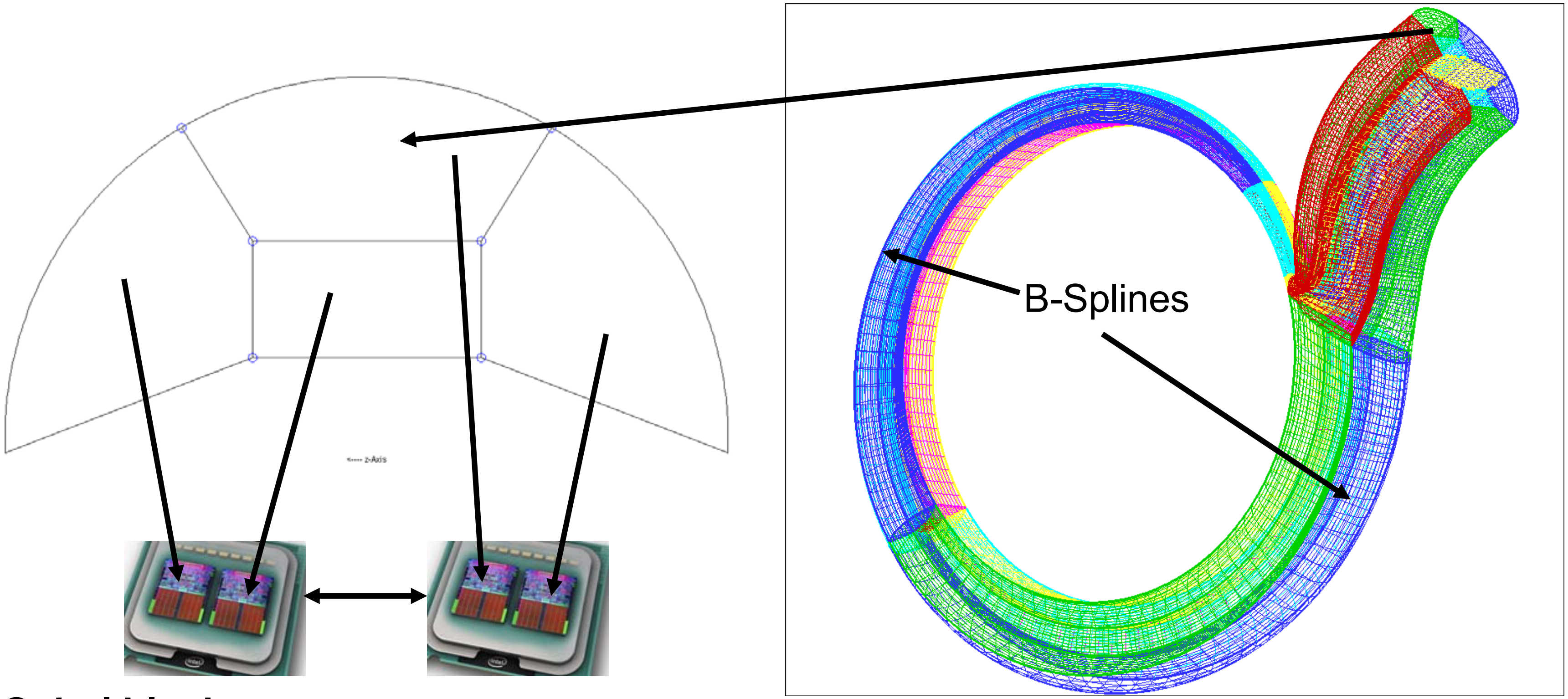
⇩

spiral cuts based on linear surfaces

⇩

spiral is cut into slices

Spiral segmentation into grid blocks



Spiral blocks:

-calculation areas segment the flow channel

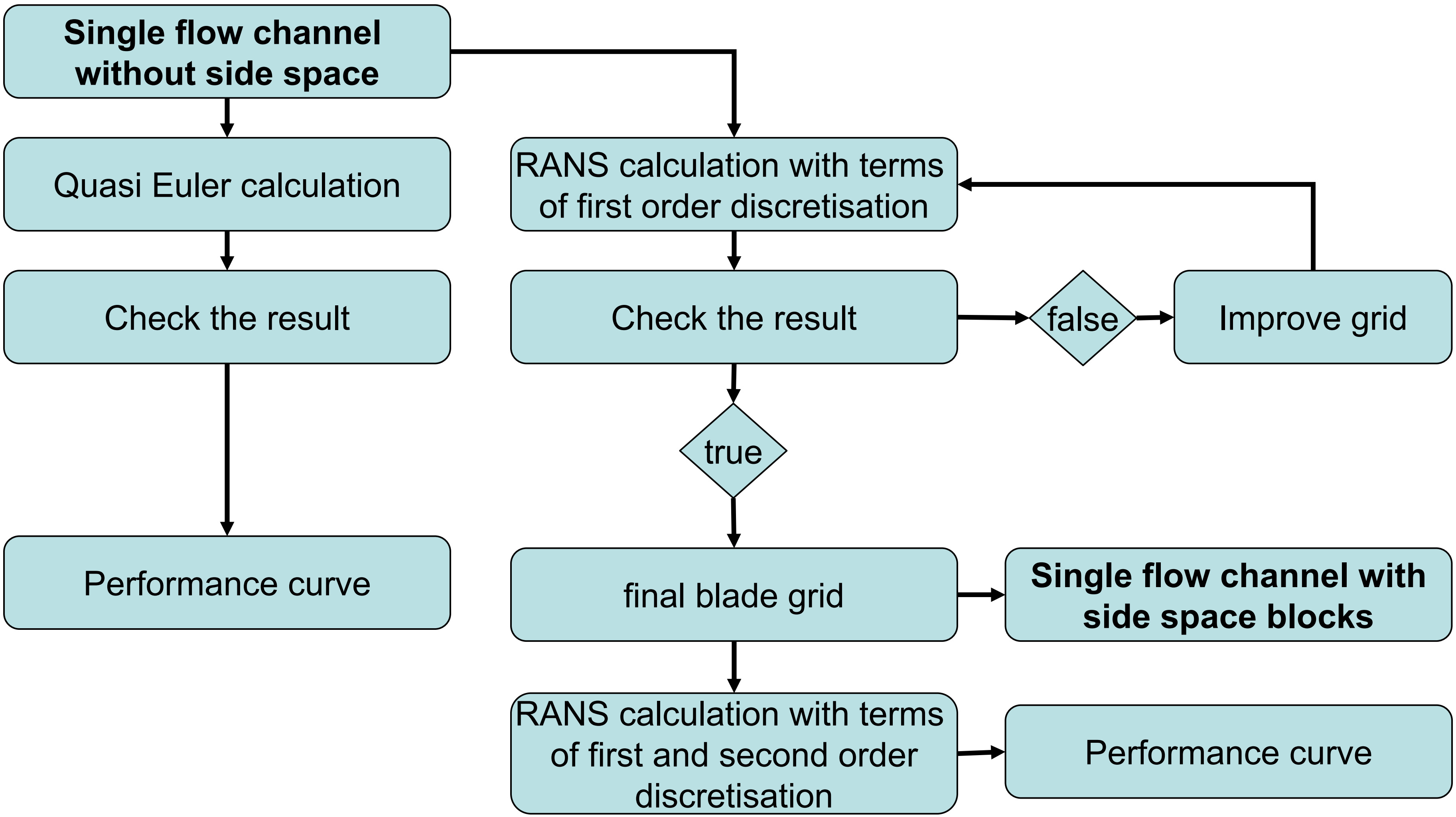
-lines in peripheral and radial direction consists of B-Splines

-complete spiral can be meshed with hexahedrons

Content

1. Motivation
2. Numerical test rig „IDS“
   - Geometric Set, Preprocessing and grid generation
3. Simulation approaches
   - Level of detail
   - Fluid flow modeling

Single flow channel without side space



Single – flow channel without side space

-Quasi Euler calculation for approximation

-RANS calculation with terms of first and second order discretisation

-Simplified outlet model

-Blade grid optimization

-Performance curves for operation characteristics

-Reference for further simulations

Inputs:

-Rotation frequency	ω
-Volume flow	$\dot{V}$
-Reference radius	r_{ref}
-Density of fluid	ρ
-Viscosity	ν
-Turbulence parameters	(k, ε)
-Number of blades	

$$k = 1{,}5 \cdot (Tu \cdot c_{Inlet})^2 \qquad c_{inlet} = \frac{\dot{V}}{A_{inlet}}$$

$$\varepsilon = C_{\mu}^{\,0{,}75} \frac{k^{1{,}5}}{l_{char}} \qquad C_{\mu} = 0{,}09 \qquad l_{char} \cong 0{,}07 r_{ref}$$

Quasi Euler calculation

-grid with large cells ⇨ low number of nodes

-no wall friction

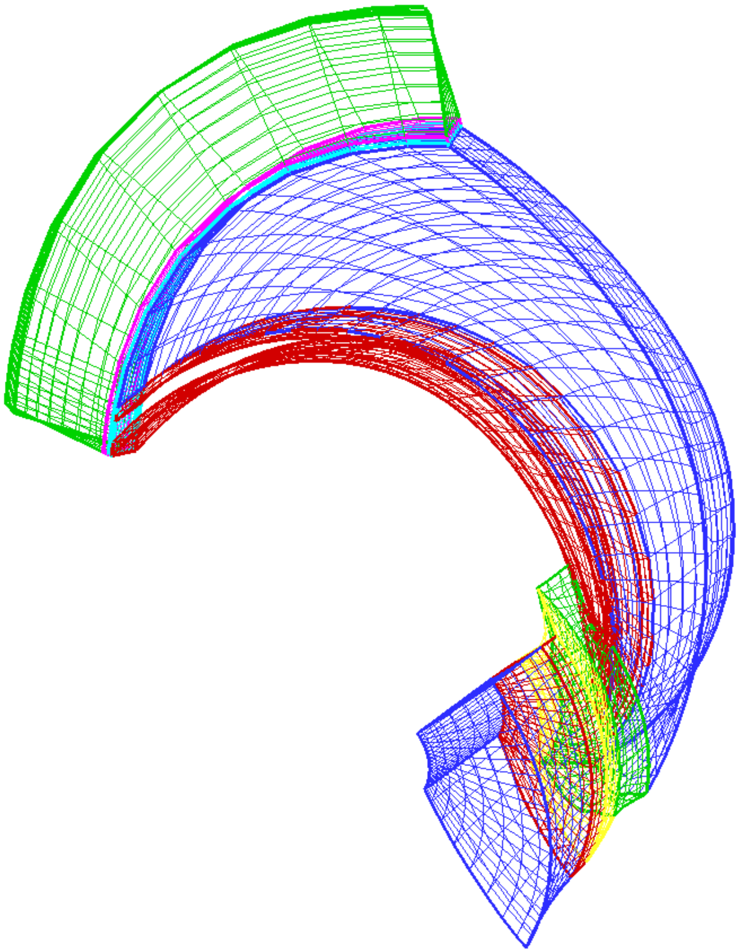
-no turbulence model

-adjusted viscosity

-fast and simple calculation

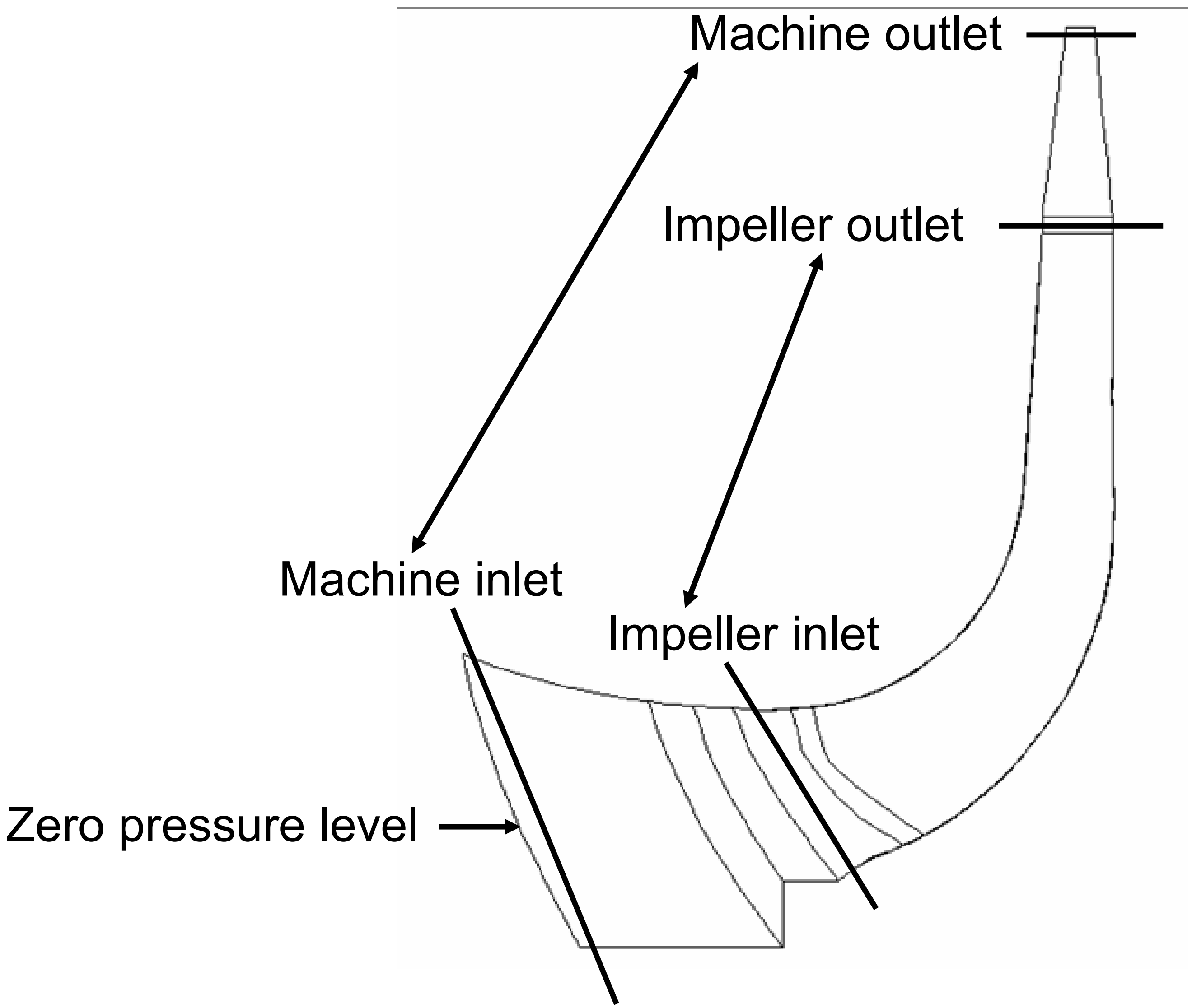
-rough result

-first approximation of performance

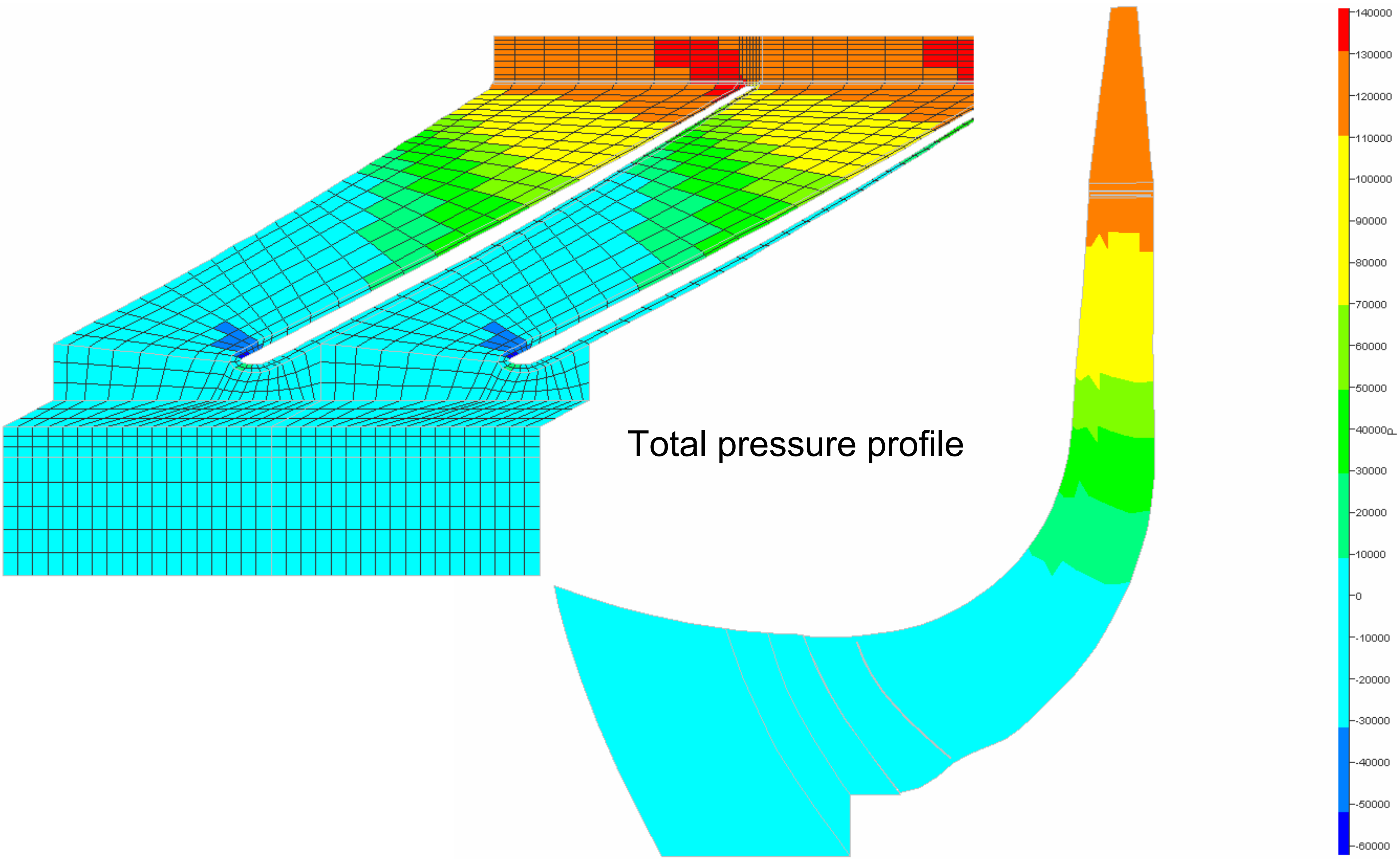


9522 Nodes

Measurement positions, balancing



Result of a quasi Euler calculation



RANS calculation with wall friction

-fine grid ⇨ increased number of nodes

-simplified outlet model

-k, ε – turbulence model

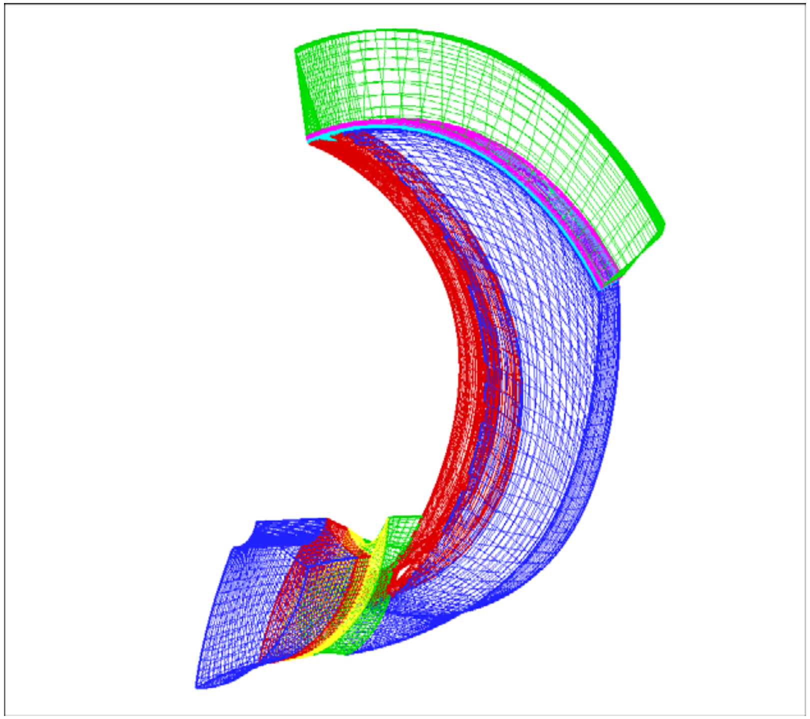
-precise result

-no cavitation effects

-wall friction

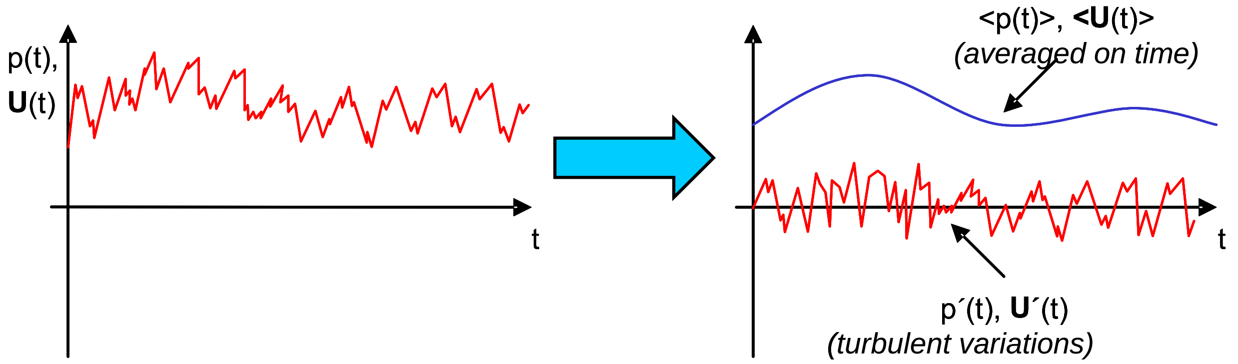
-calculation with terms of first and second order discretisation

-final blade grid



about 54000 nodes

Reynolds averaged turbulence model



real stream value

Reynolds averaged stream value
$\Delta t_{Mittelung} >> \Delta t_{Turbulenz}$

$$U(t) = \bar{u}_1 + \bar{u}_2 + \bar{u}_3$$

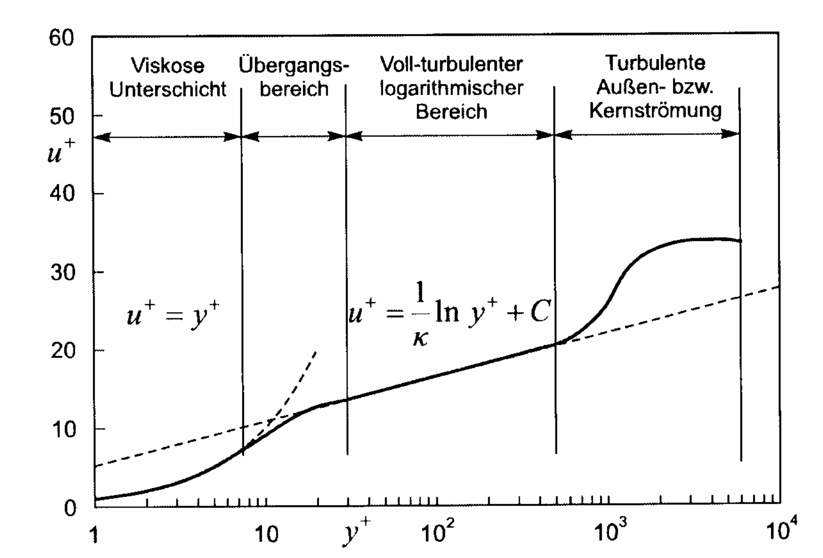
$$k = \frac{1}{2}\left\{\bar{u}_1^2 + \bar{u}_2^2 + \bar{u}_3^2\right\}$$

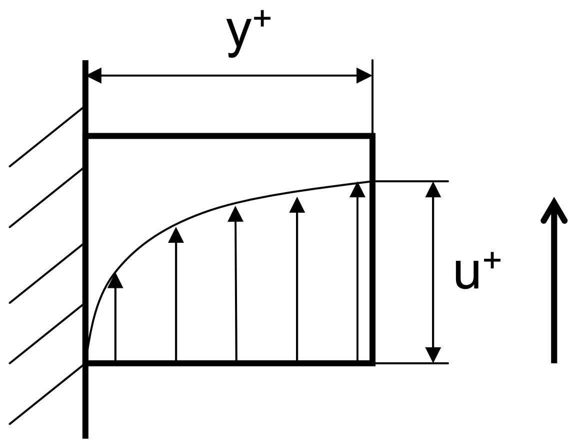
$$\nu_t = c_\mu \cdot \frac{k^2}{\varepsilon}$$

$$c_\mu = 0.09$$

u =velocity [m/s]
ν_t=turbulent viscosity $[m^3/s^2]$
ε =dissipation rate $[m/s^2]$
k =turbulent kinetic energy $[m^2/s^2]$

Wall friction model

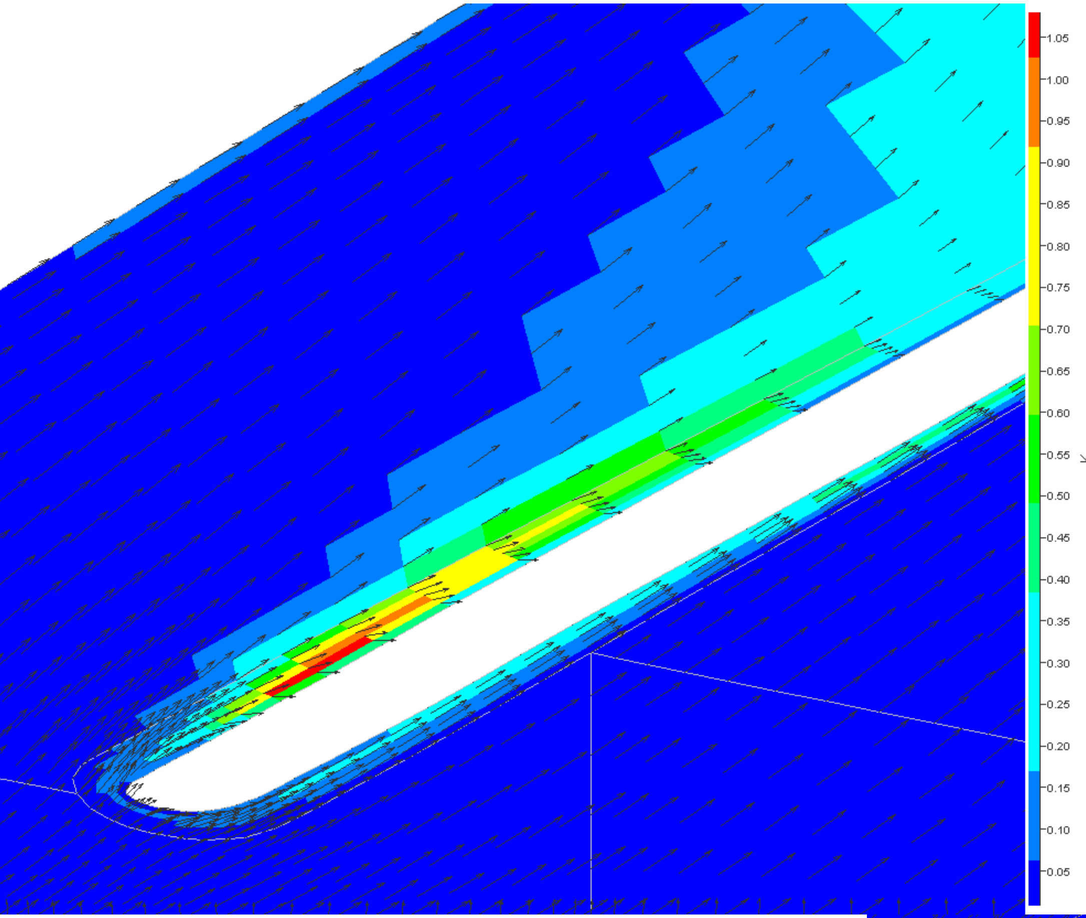


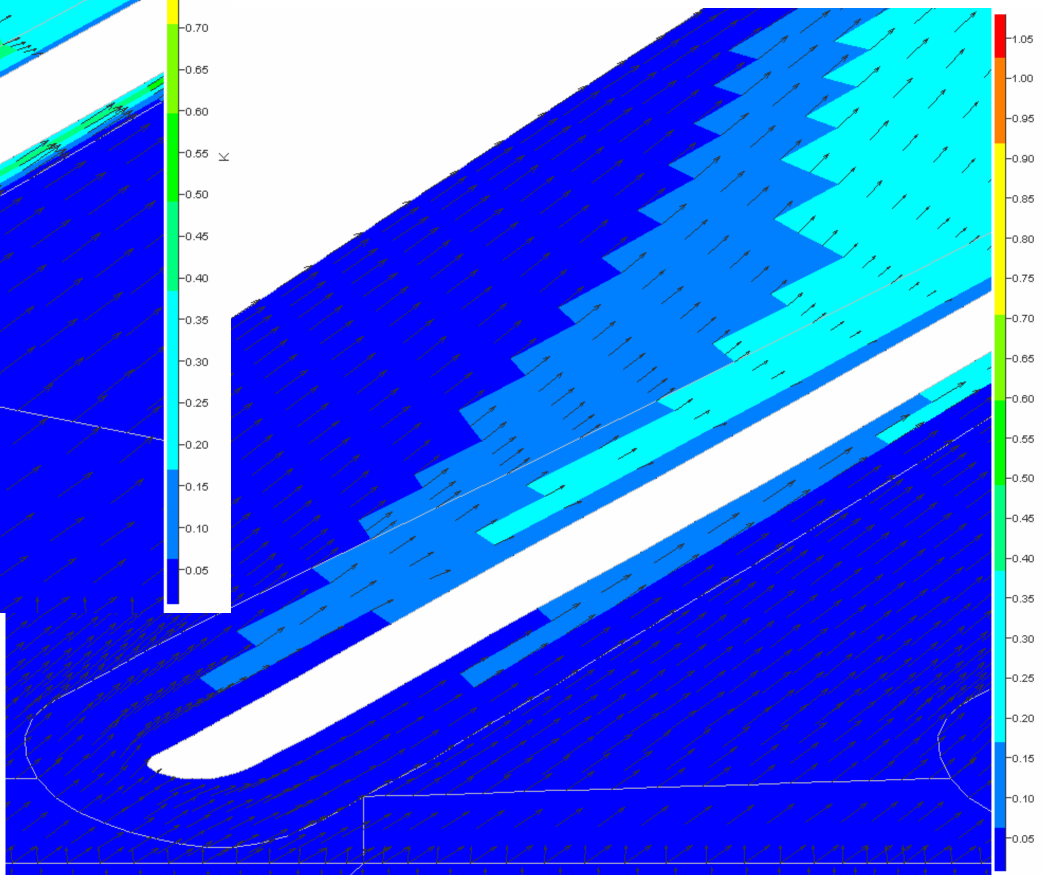


Velocity → Wall shearing stress → Wall Friction force

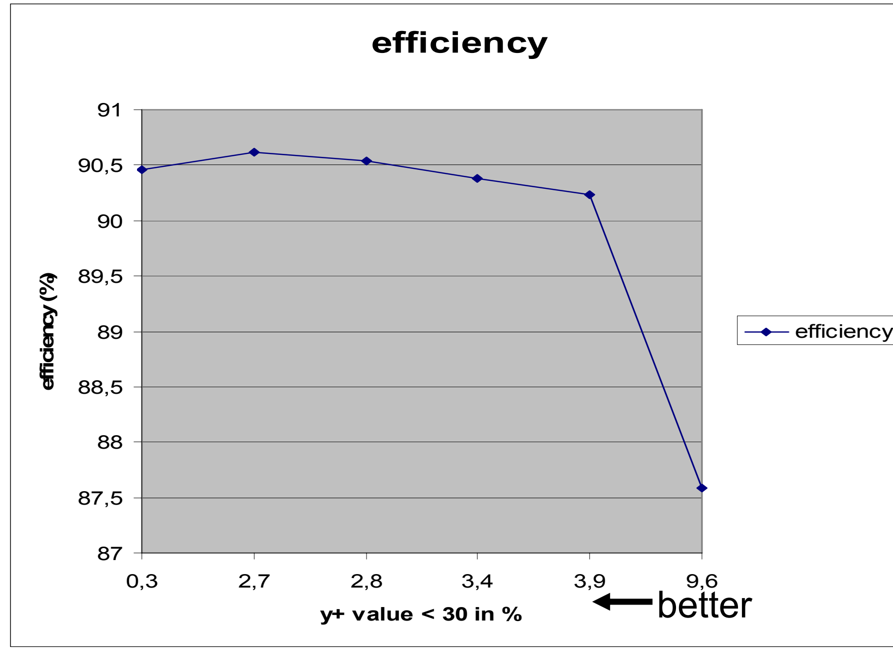
Evaluation of wall function

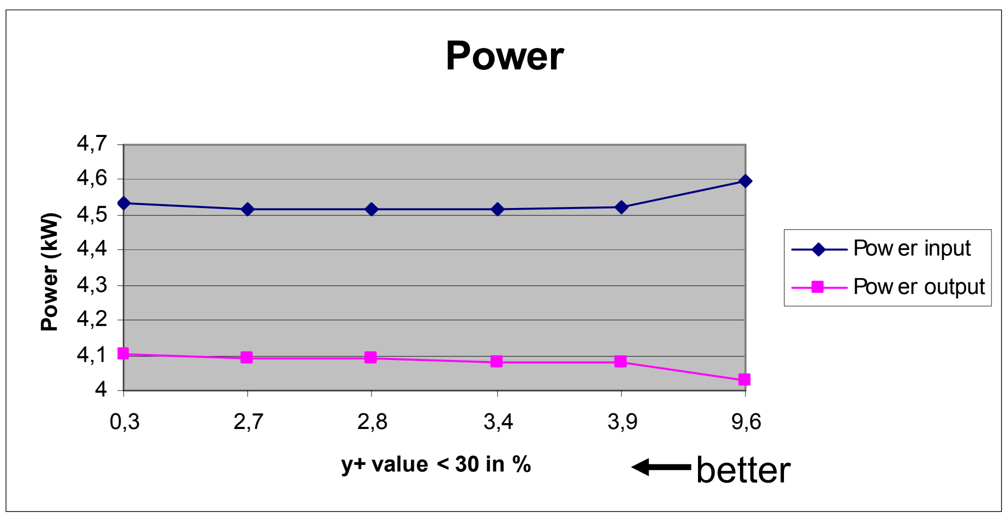
Turbulent kinetic energy



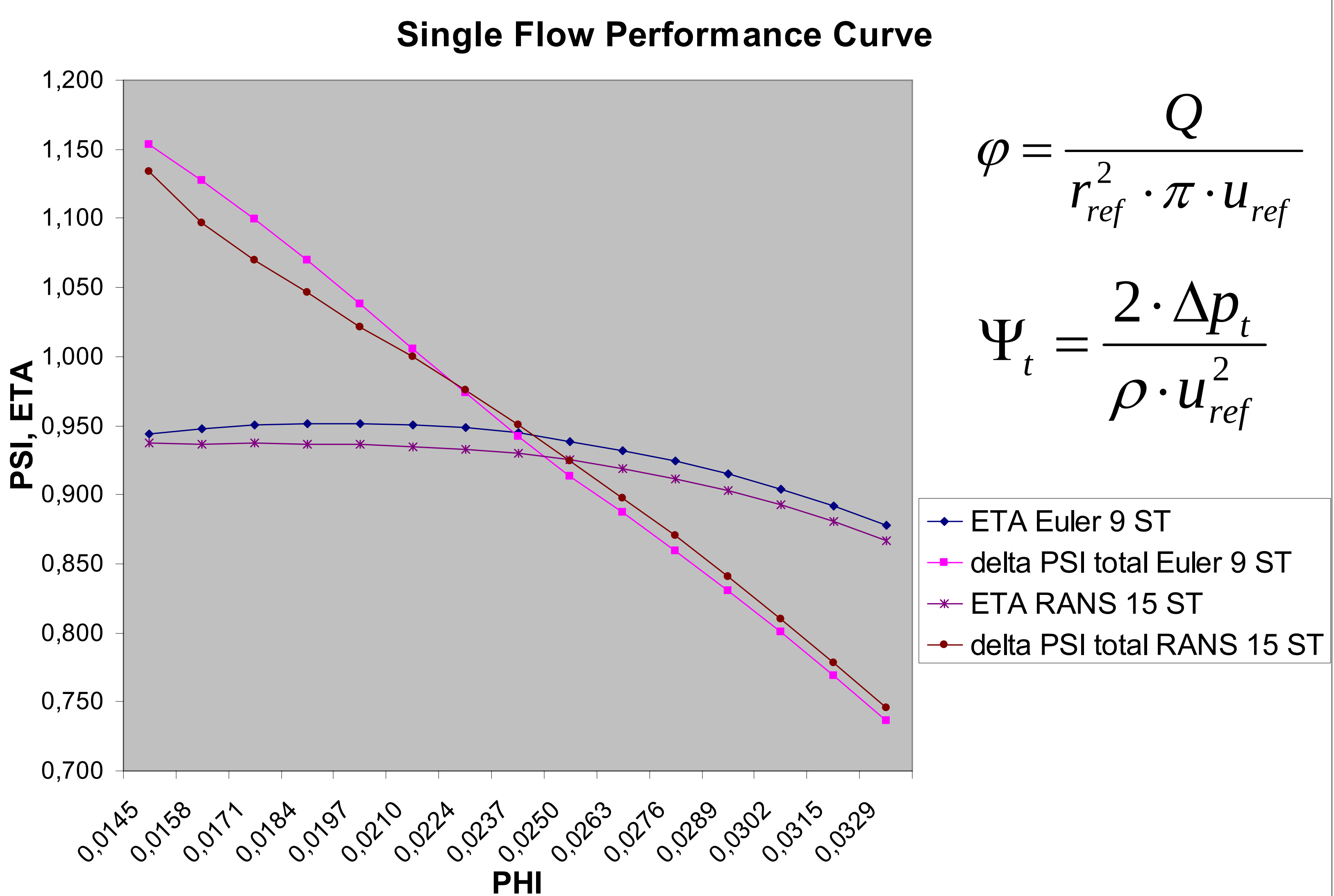


Evaluation of wall function





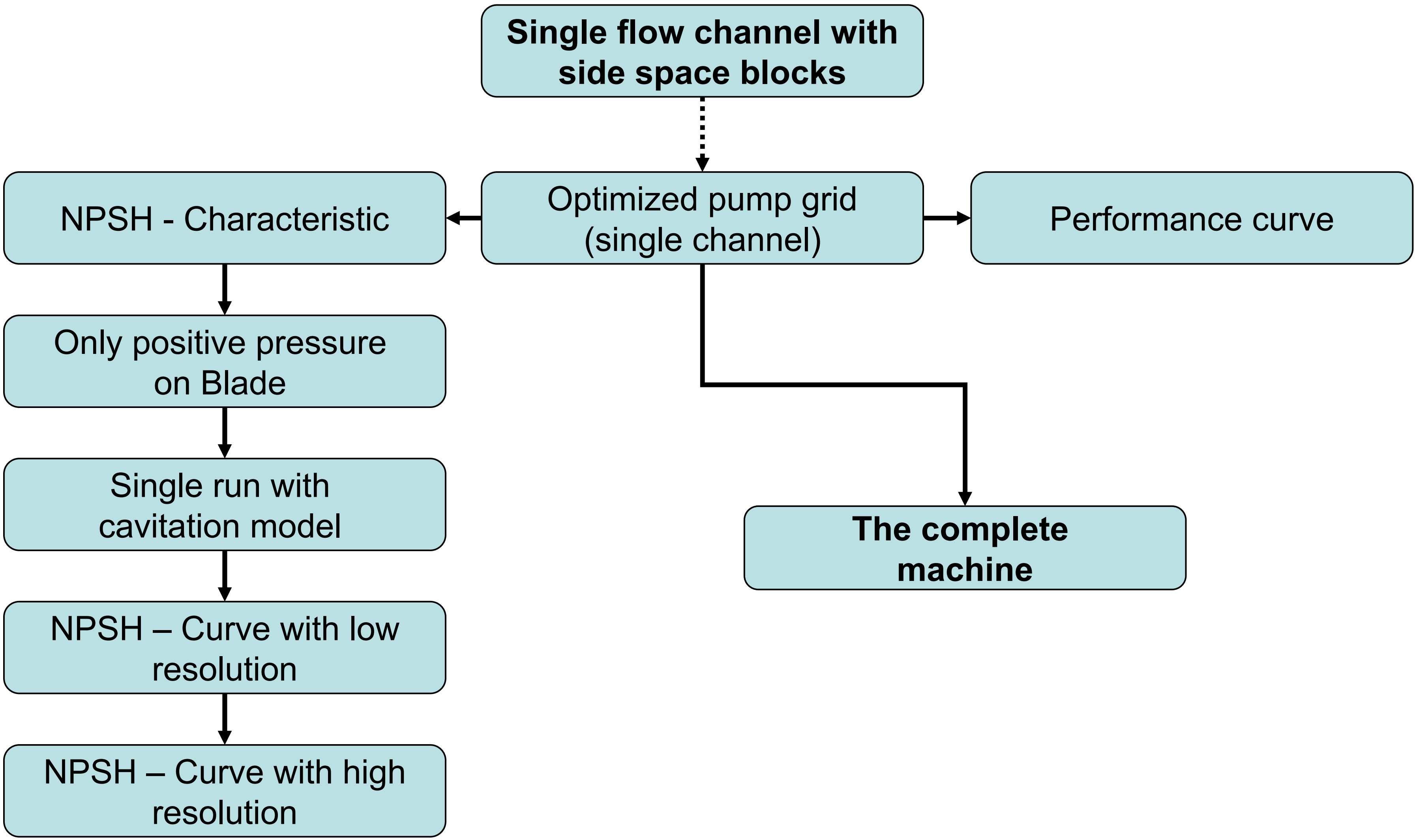
Comparison of results



$$\varphi = \frac{Q}{r_{ref}^2 \cdot \pi \cdot u_{ref}}$$

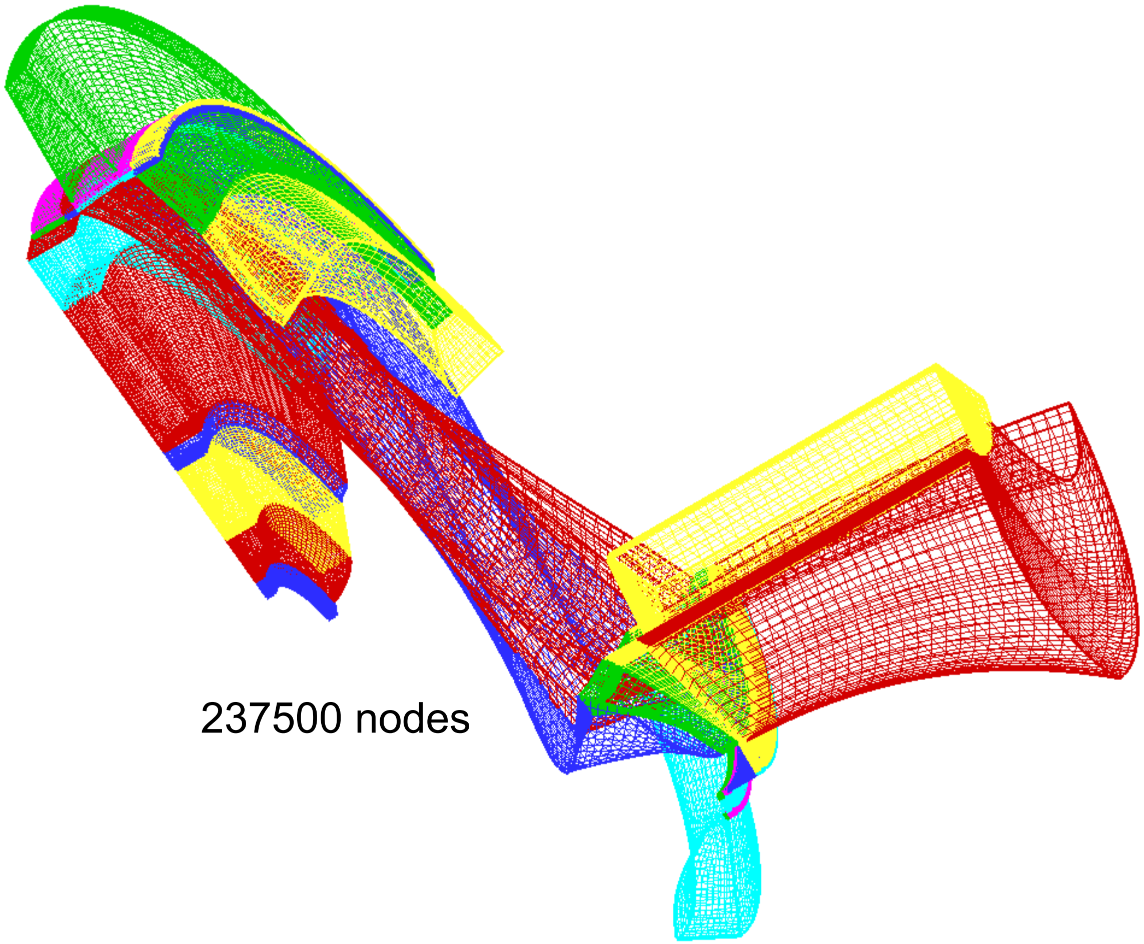
$$\Psi_t = \frac{2 \cdot \Delta p_t}{\rho \cdot u_{ref}^2}$$

Single flow channel with side space



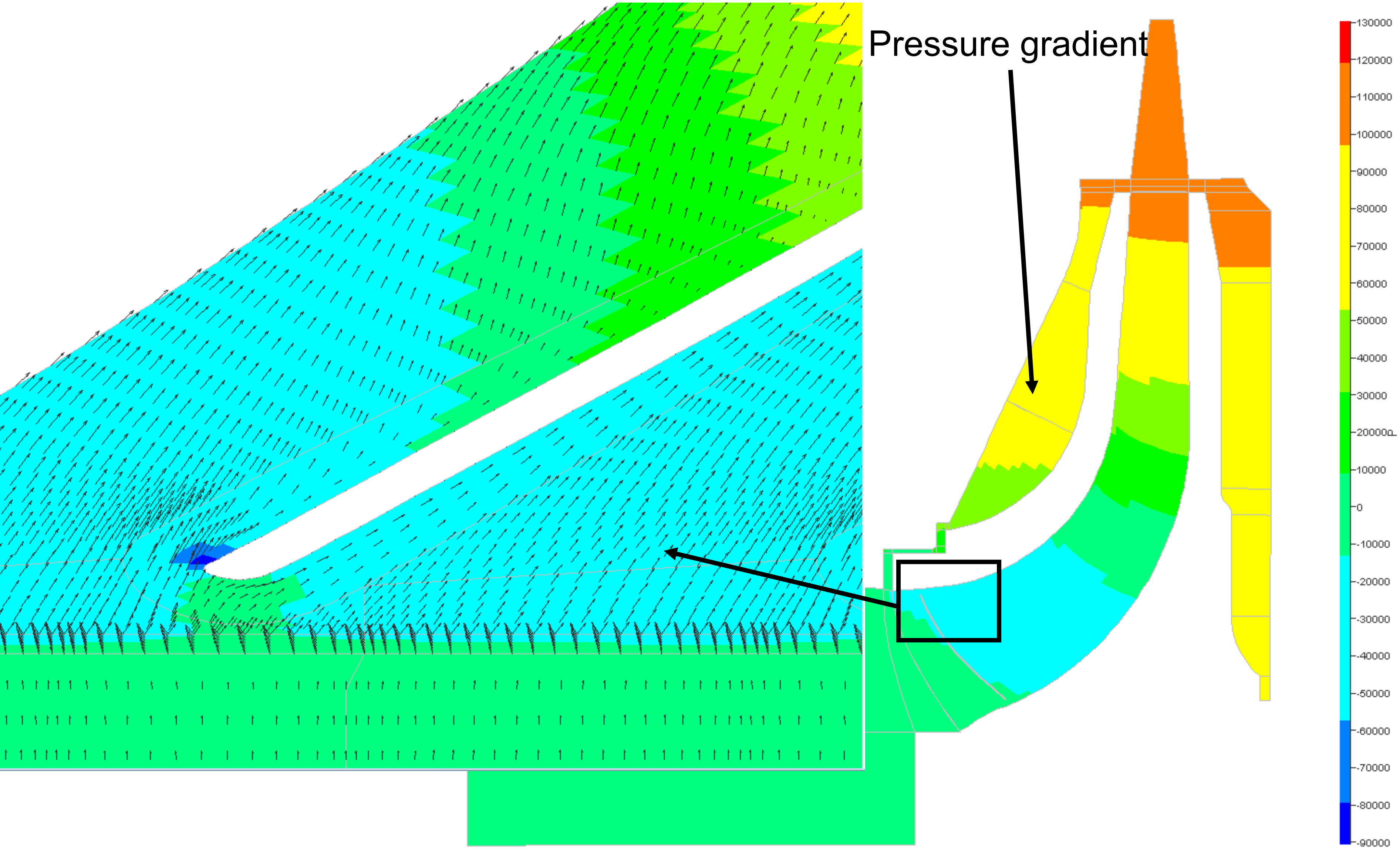
Single flow channel with side space blocks

- -fine grid ⇨ large number of nodes
- -simple outlet
- -k, ε – turbulence model
- -precise result
- -no cavitation
- -wall friction
- -calculation with terms of first and second order discretisation
- -optimized side space grid
- -reference for NPSH curve



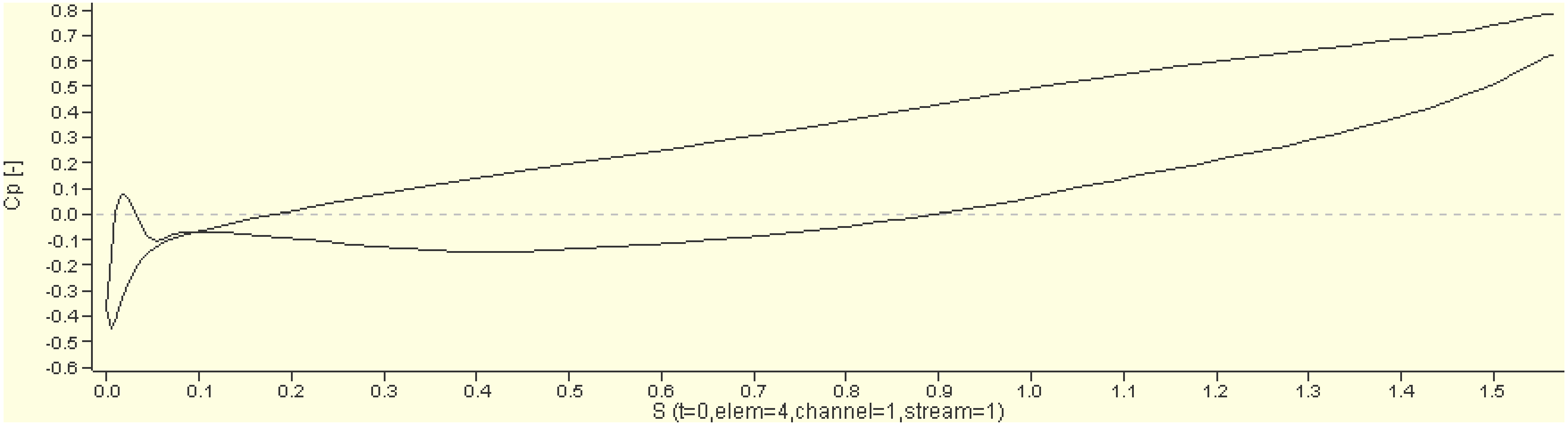
237500 nodes

Total pressure meridian contour

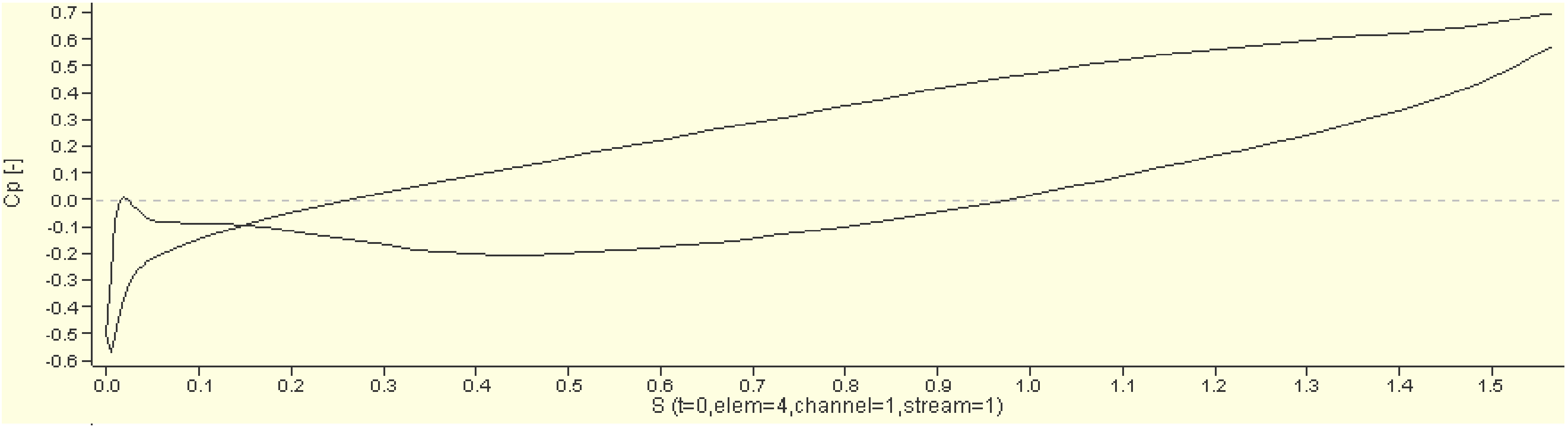


Pressure profile on blade

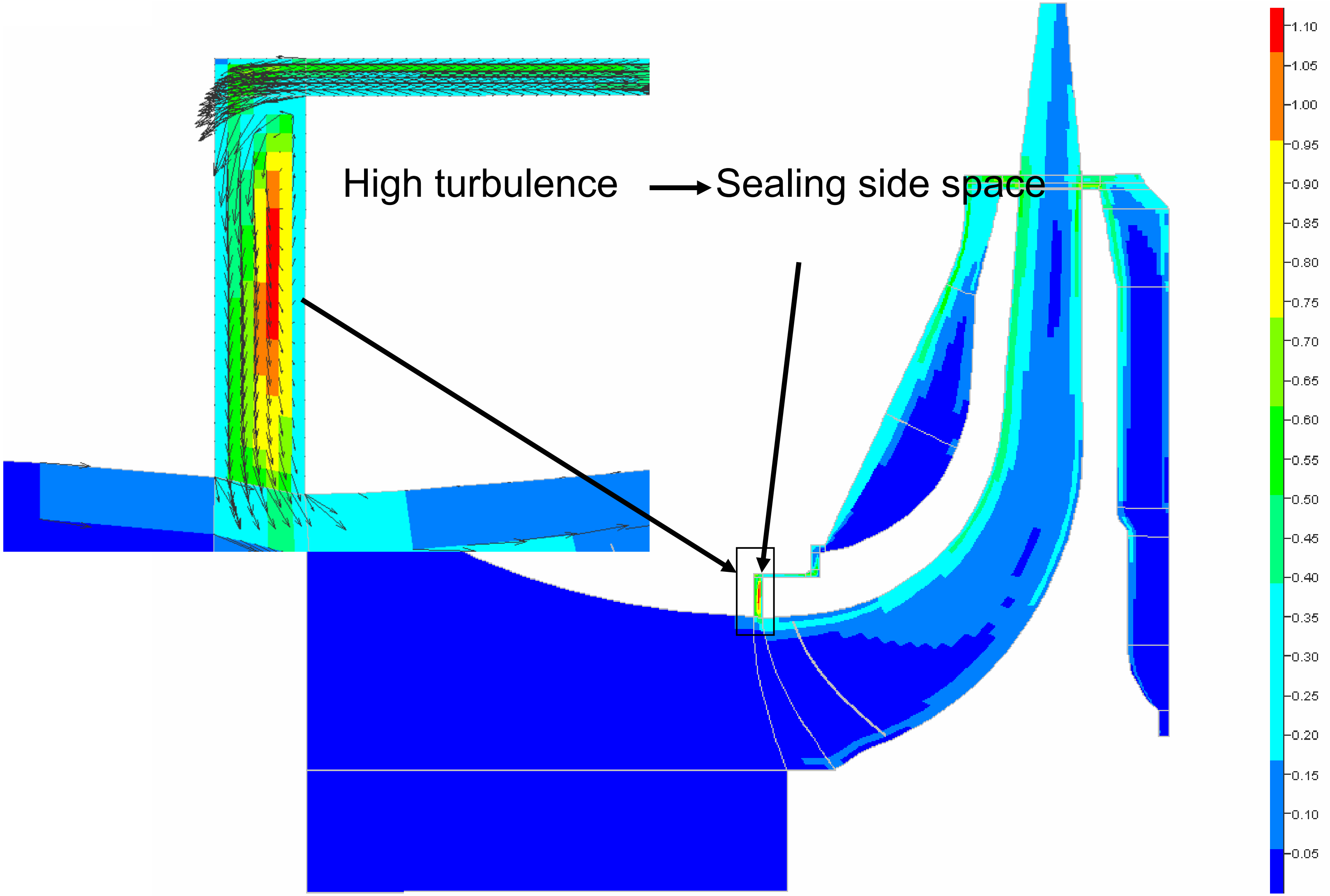
Pressure profile on blade without side space



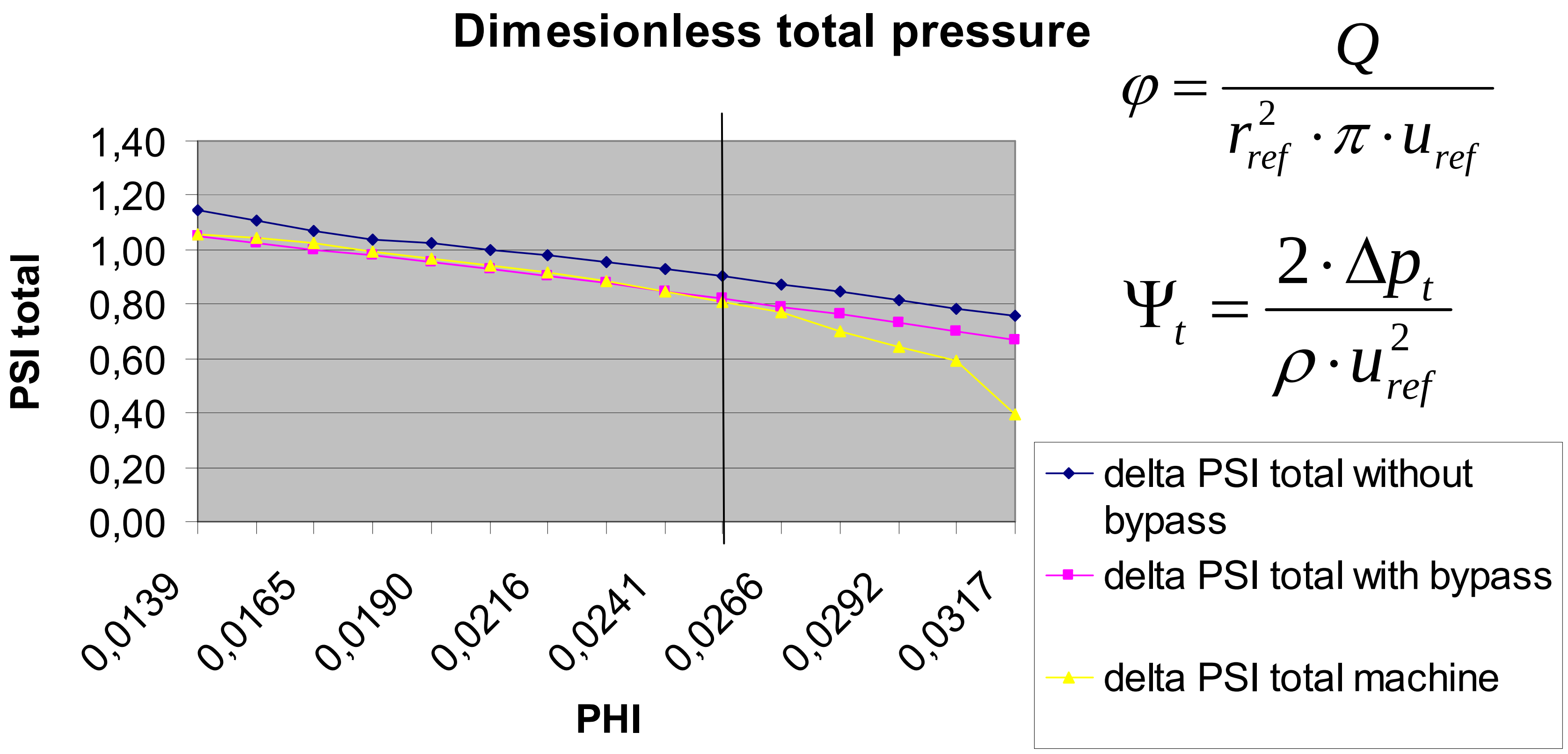
Pressure profile on blade with side space



Turbulent kinetic energy meridian contour



Comparison of results



$$\varphi = \frac{Q}{r_{ref}^{2} \cdot \pi \cdot u_{ref}}$$

$$\Psi_t = \frac{2 \cdot \Delta p_t}{\rho \cdot u_{ref}^{2}}$$

NPSH - Characteristic

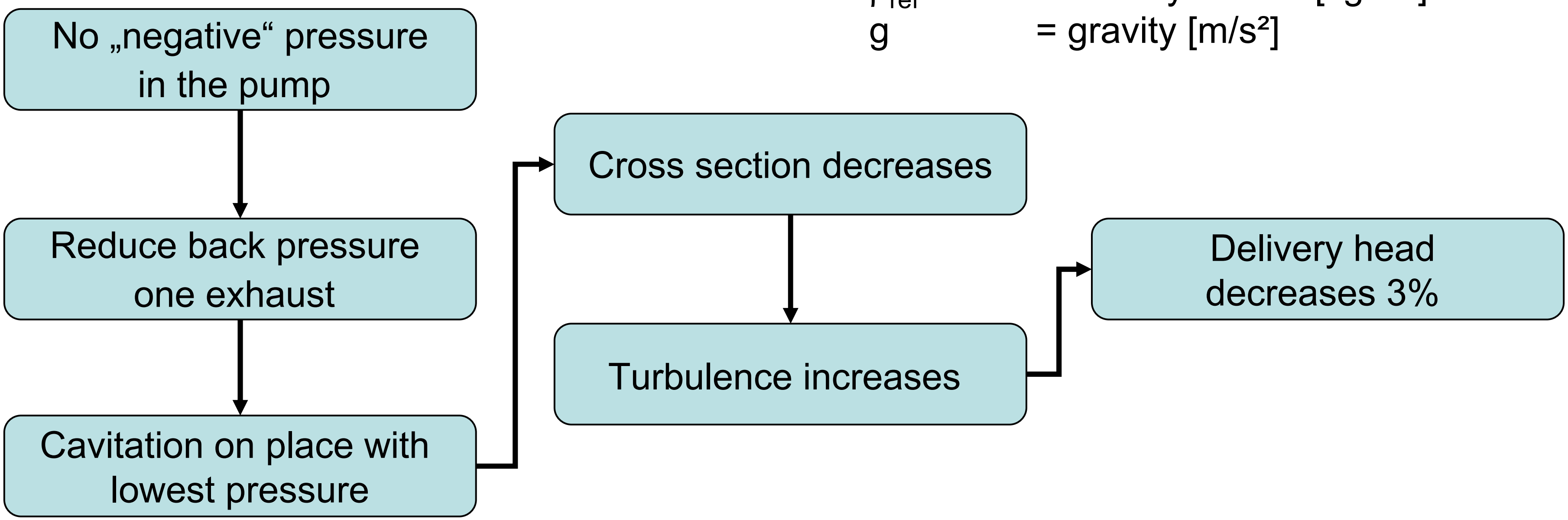
NPSH **N**et **P**ositive **S**uction **H**ead

-Single flow channel with side space
-Cavitation model (volume fraction based)
-Back pressure on exhaust

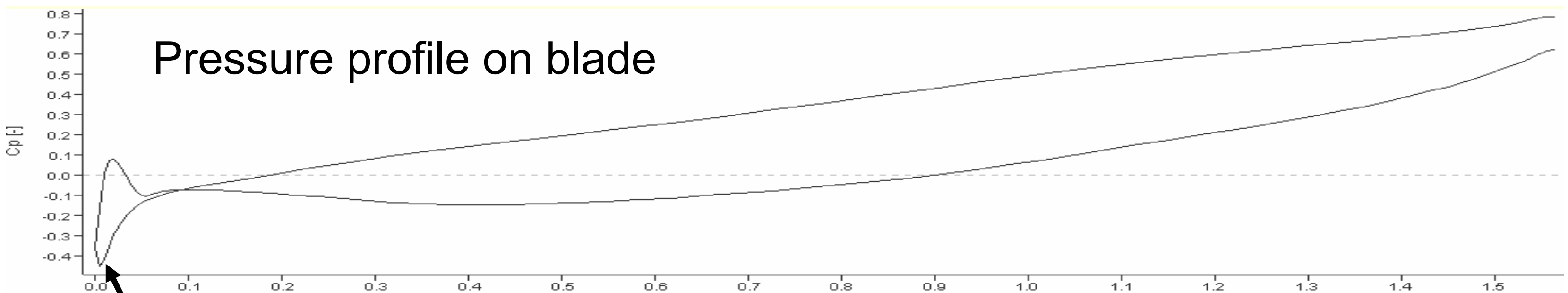
$$NPSH = \frac{p_{t,inlet} - p_{vapor}}{\rho_{ref} \cdot g}$$

NPSH-Value [m]

$p_{t,inlet}$ = total pressure inlet [Pa]
p_{vapor} = vaporizing pressure [Pa]
ρ_{ref} = density of fluid [kg/m³]
g = gravity [m/s²]



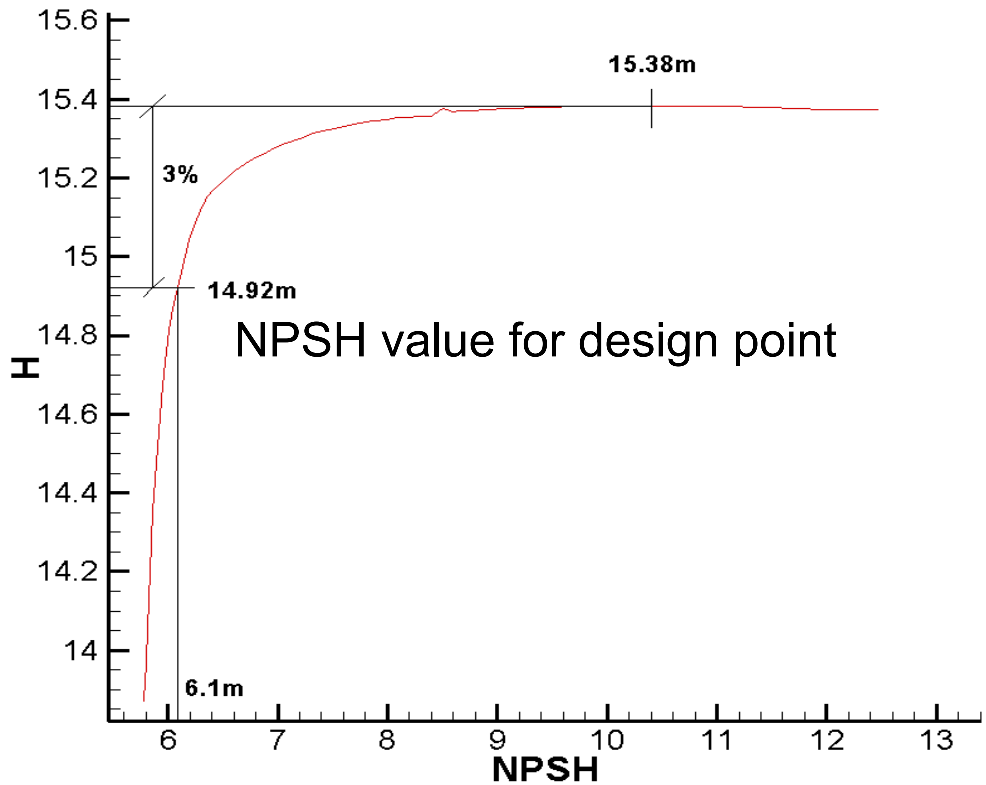
NPSH Curve



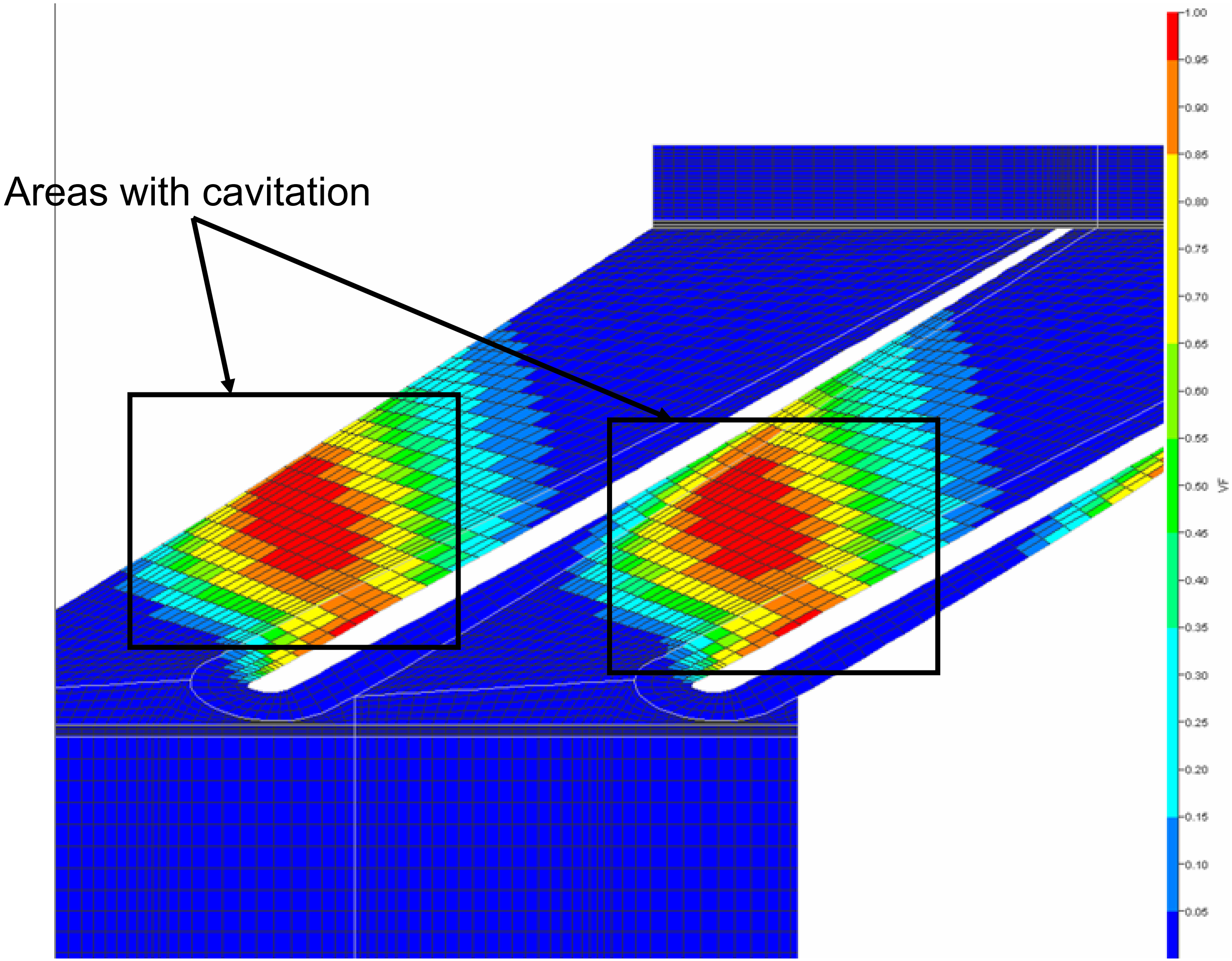
lowest pressure

⇩

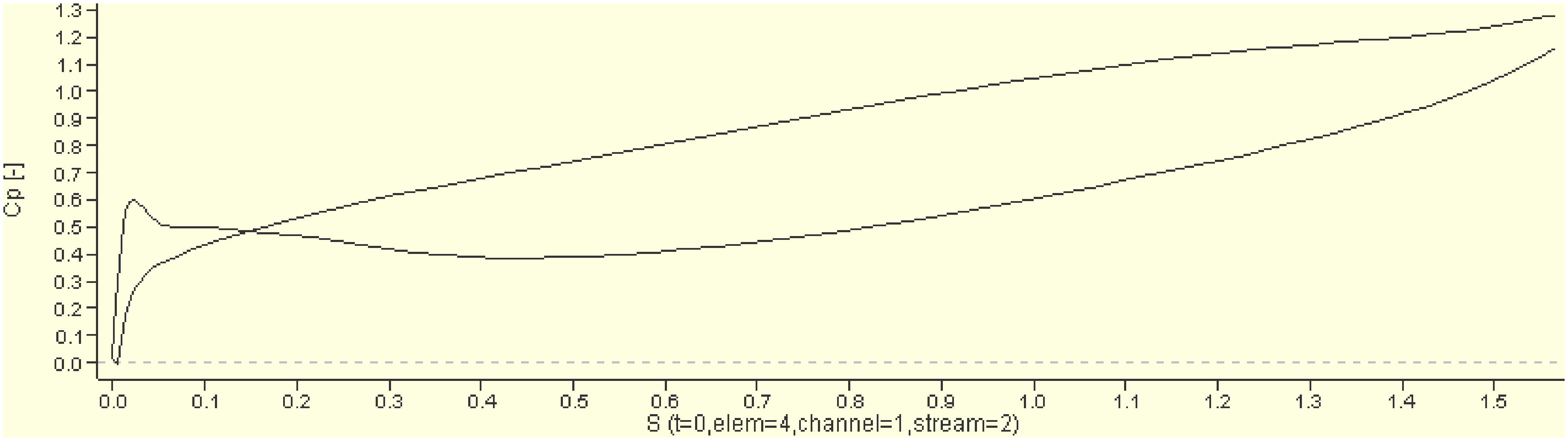
cavitate at first

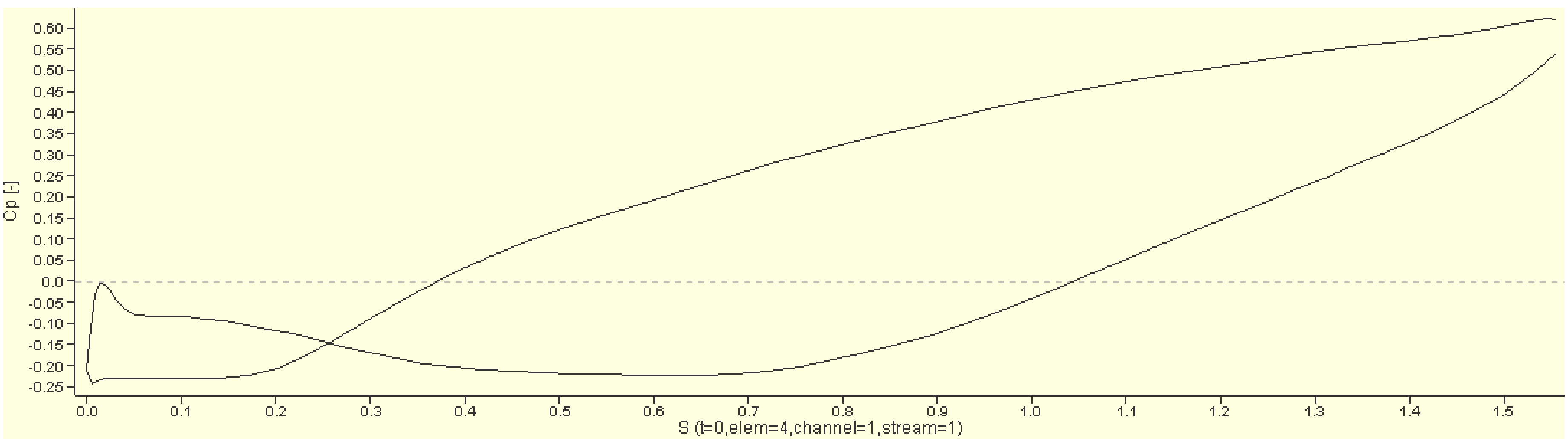


Cavitation on blade

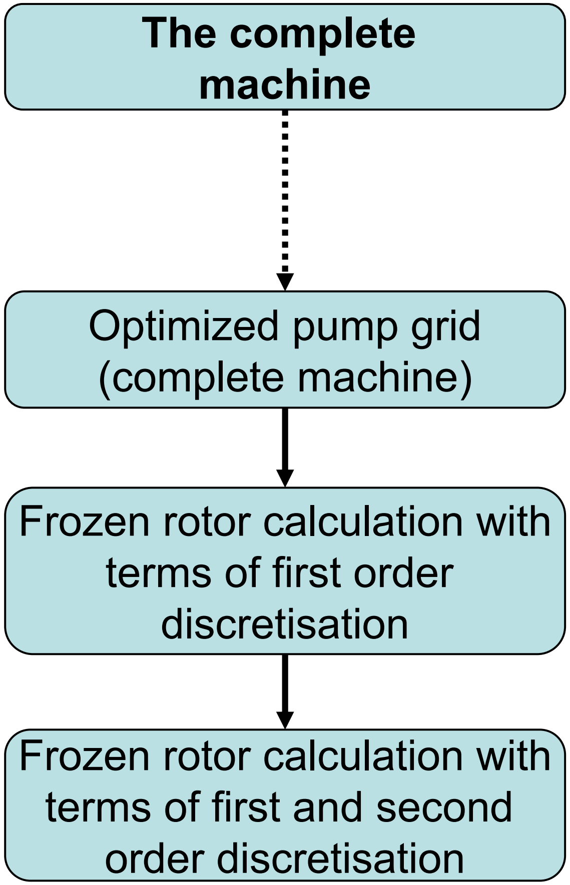


Pressure profile on blade





Complete machine



The complete machine

-rotate grid with side space

-very resource intensive calculation

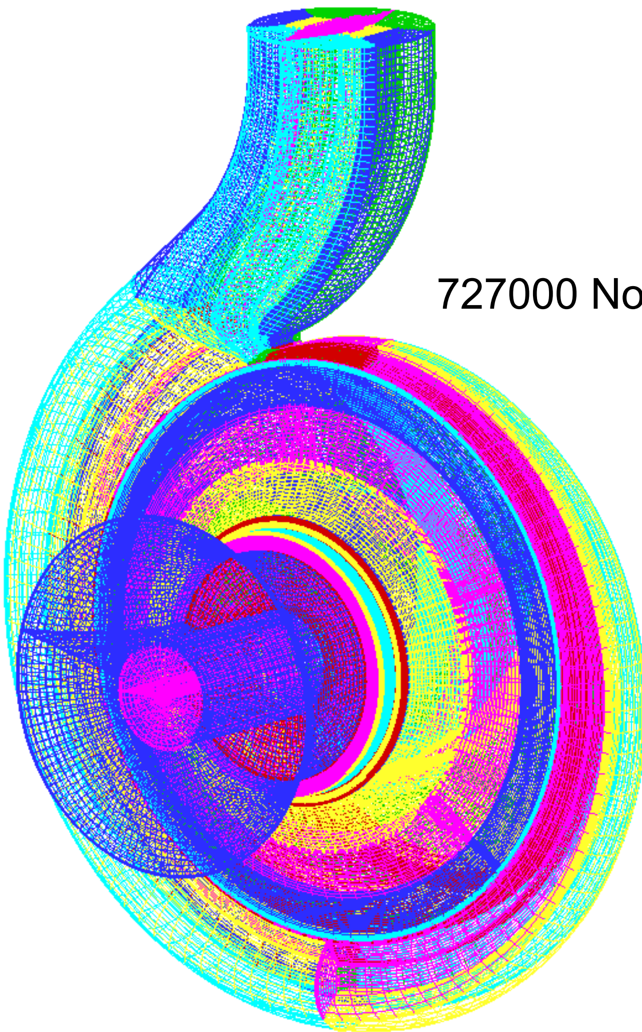
-connect spiral with rotated grid

-pump characteristic at design point

-frozen rotor calculation

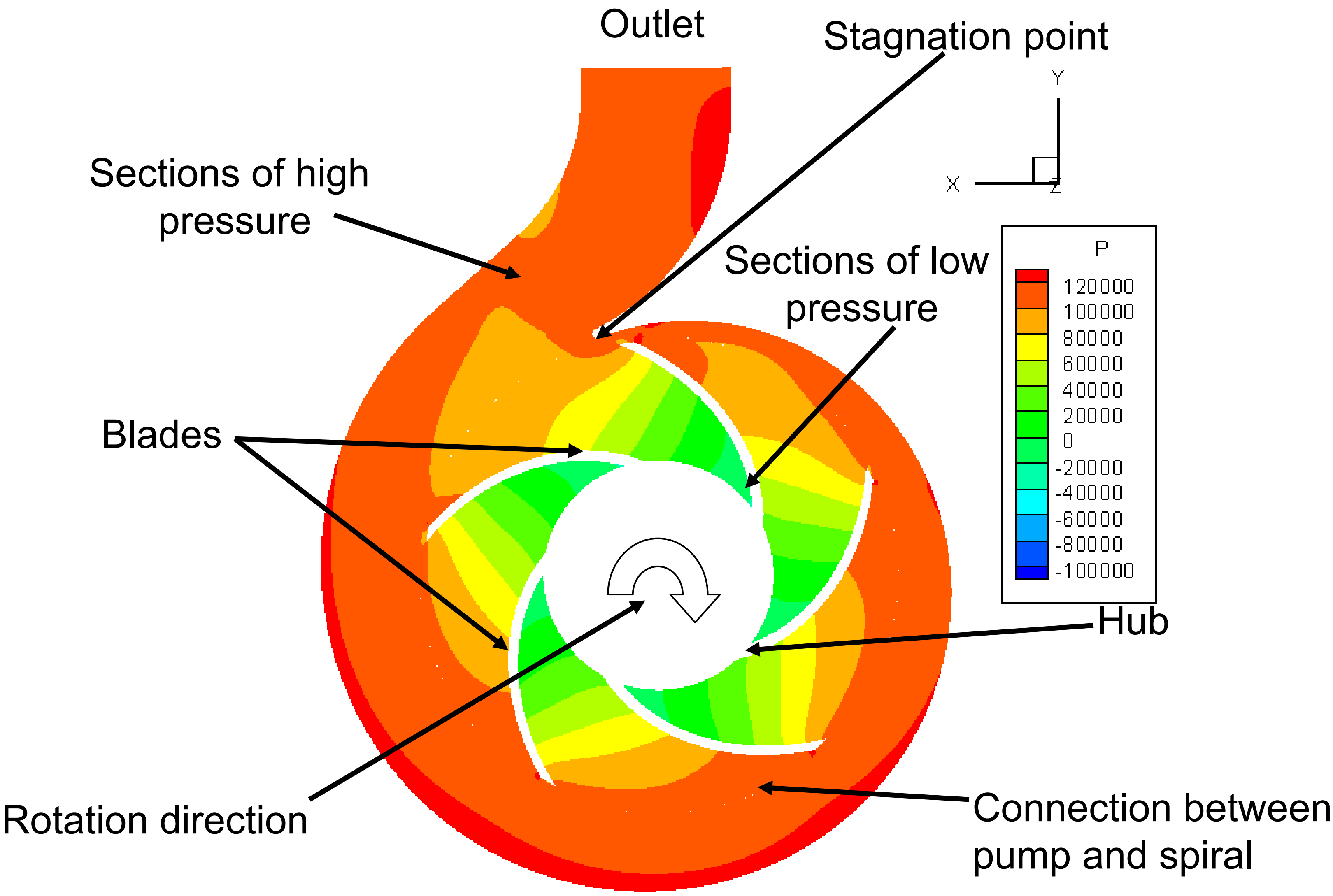
-rotor change position in steps of 9 degree

-characteristic through different rotor settings

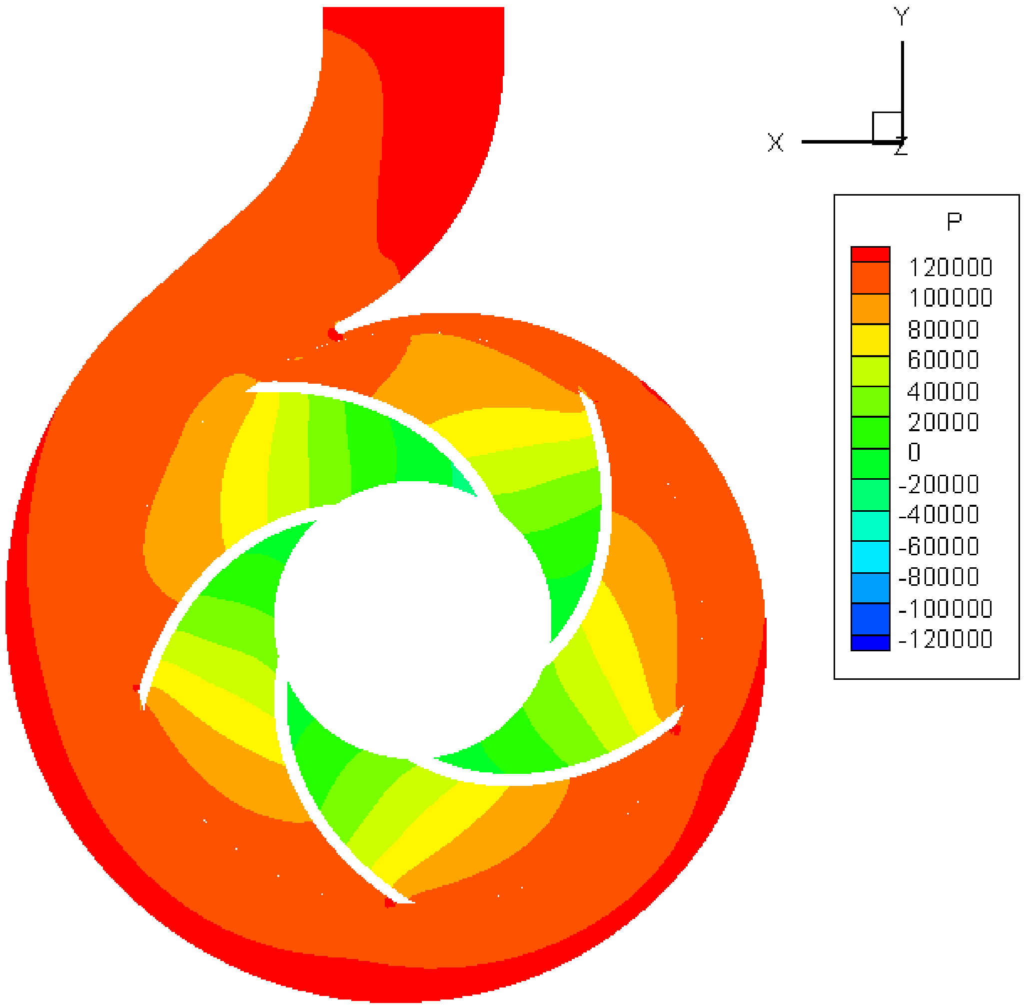


727000 Nodes

Pressure profile 0°

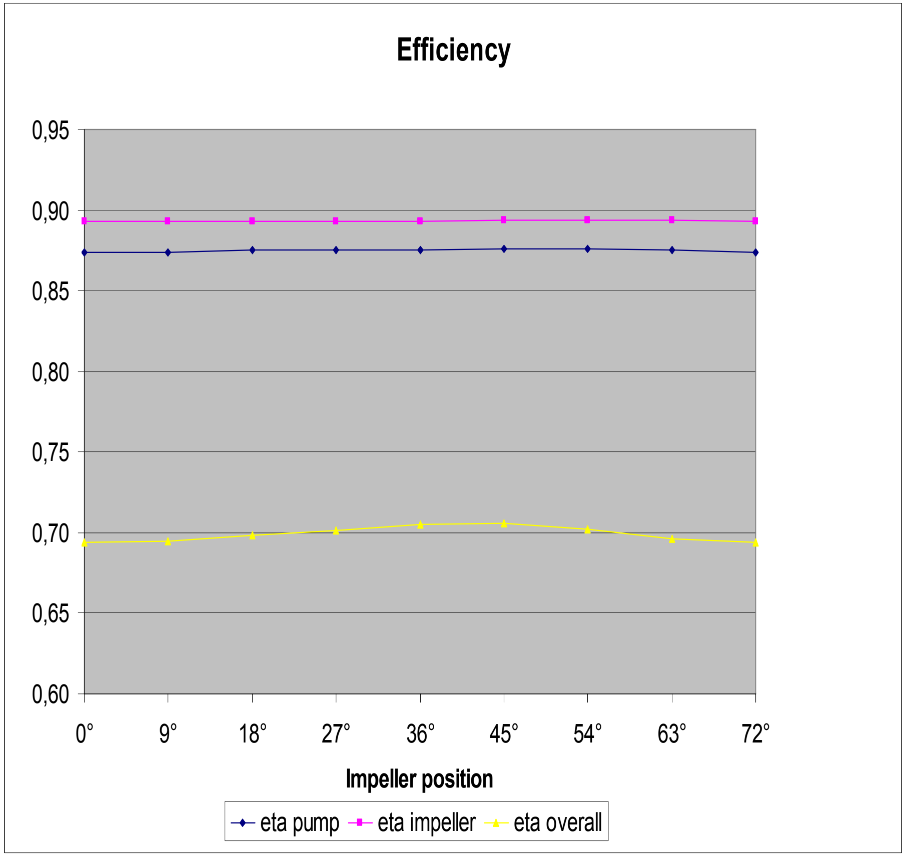


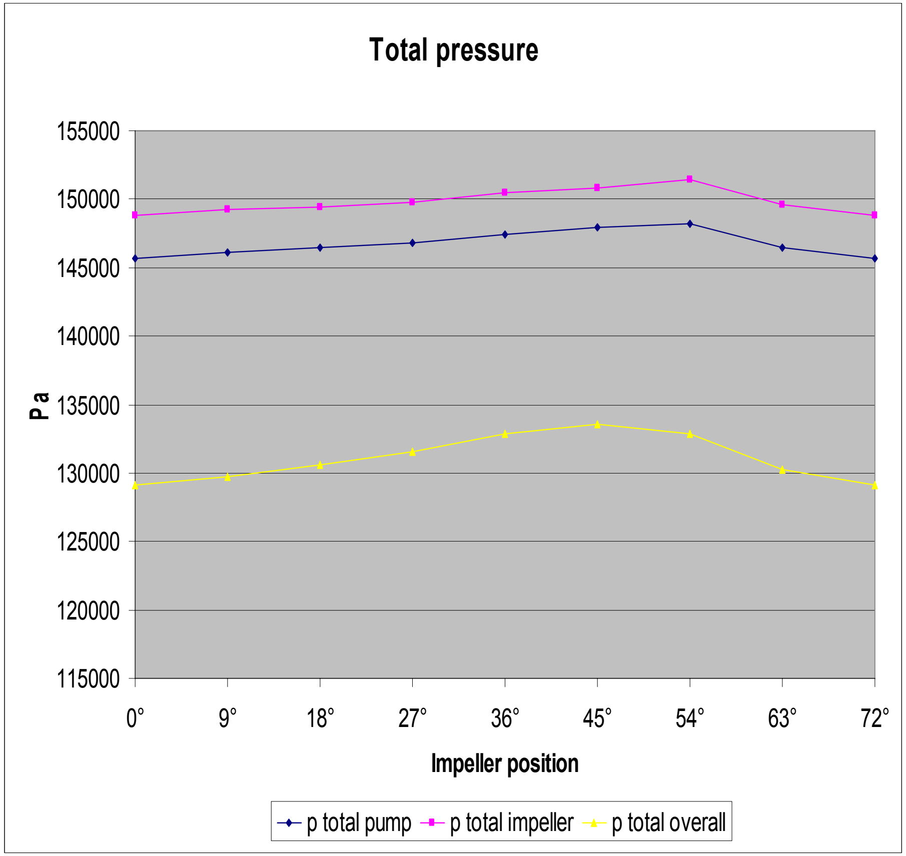
Pressure profile 45°



Comparison of results

Total pressure and efficiency depend on rotor position





Conclusions

- It is possible to mesh pump geometry with the given mashing tools.
- An iterative procedure is the best procedure to get the results.
- It is possible to get results only with the given inputs.
- The results of the IDS-Suite matching with the results of the industrial test rig.
- It is possible to design a centrifugal pump with CAE

Thank you

Discussion