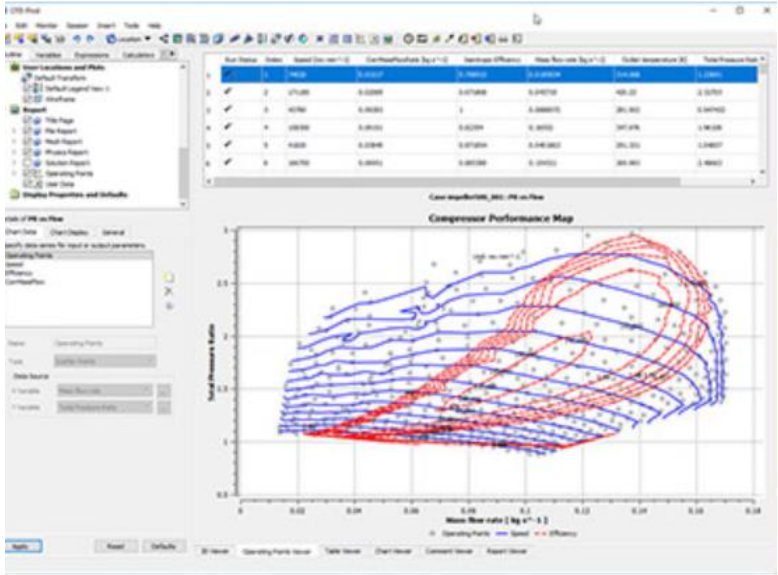
CFD for Turbomachinery

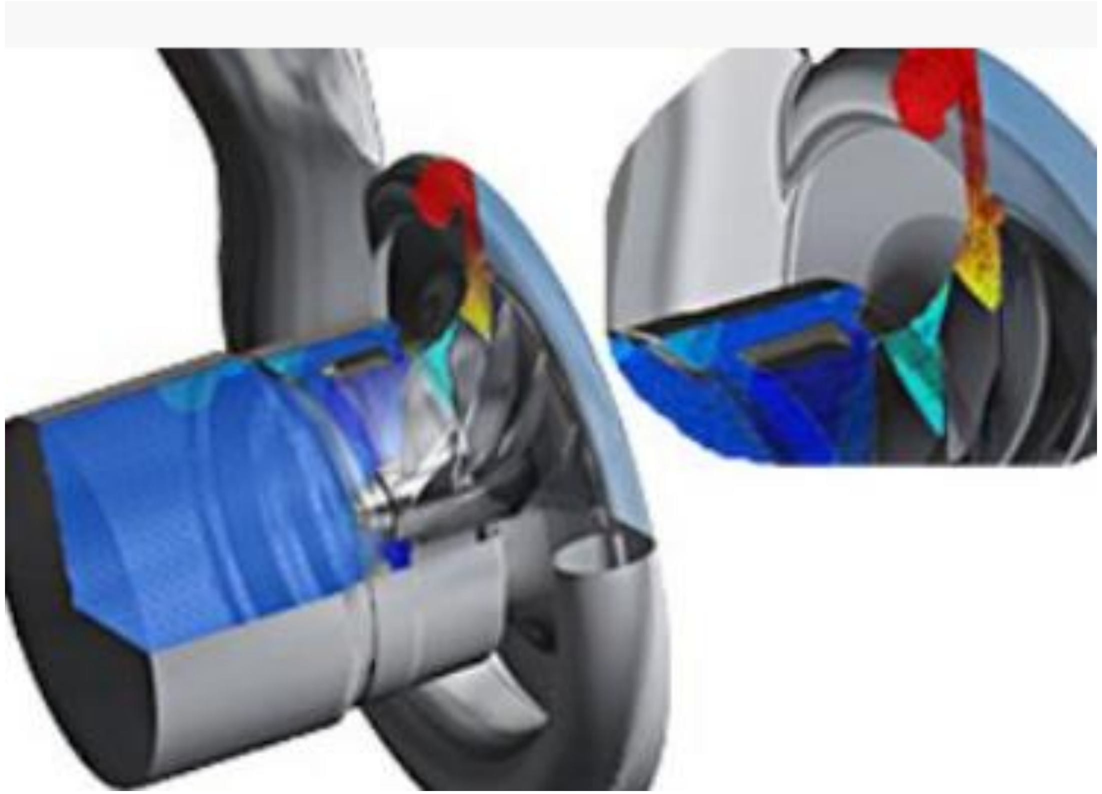
Joshua Keep

School of Mechanical and Mining Engineering

CFD for turbomachinery applications – the marketed view

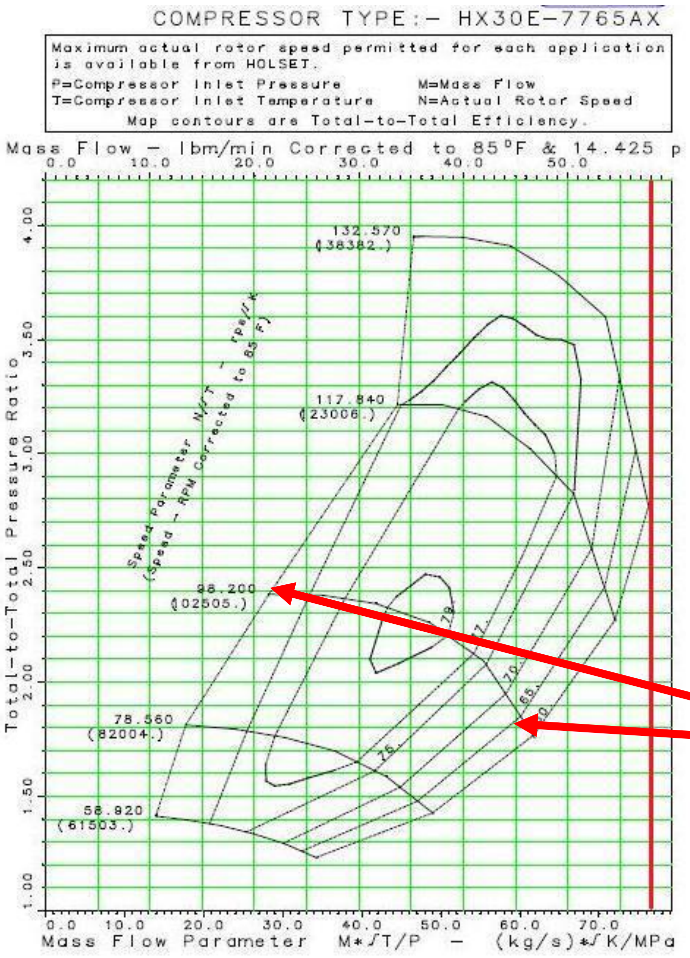
- There is a big divide between what is marketed and what is time efficient / readily achievable / trustworthy.





Full stage compressor simulation, with automated map generation (ANSYS)

CFD for turbomachinery applications – reality



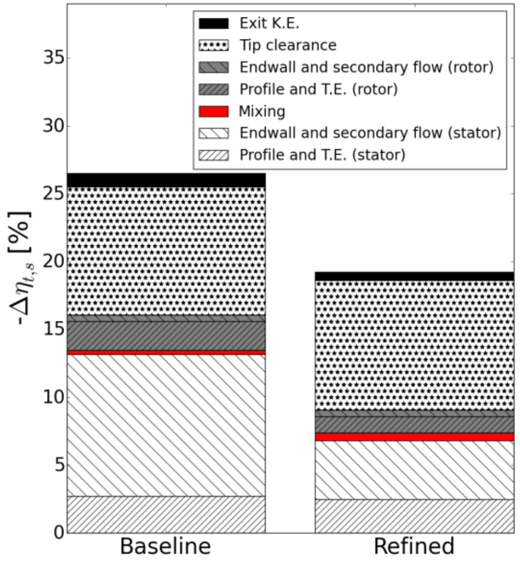
Turbocharger compressor map (Holset)

- Validation and verification
- Calibrated workflow
  - i.e. discrepancies are know, may be accounted for
  - Component / functional segregation with appropriate boundary conditions.

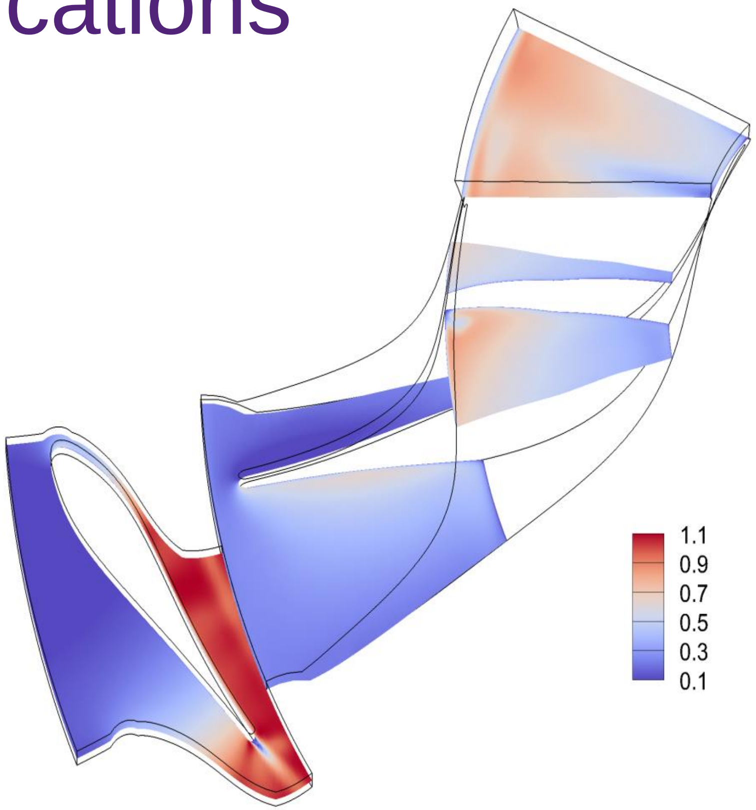
Required boundary conditions are different

Utility of CFD for turbomachinery applications

- Easy to conduct parametric studies
- Direct visualisation of variables
- Can focus on components not readily accessible in experiment (i.e results are available from areas where instruments cannot access / survive)



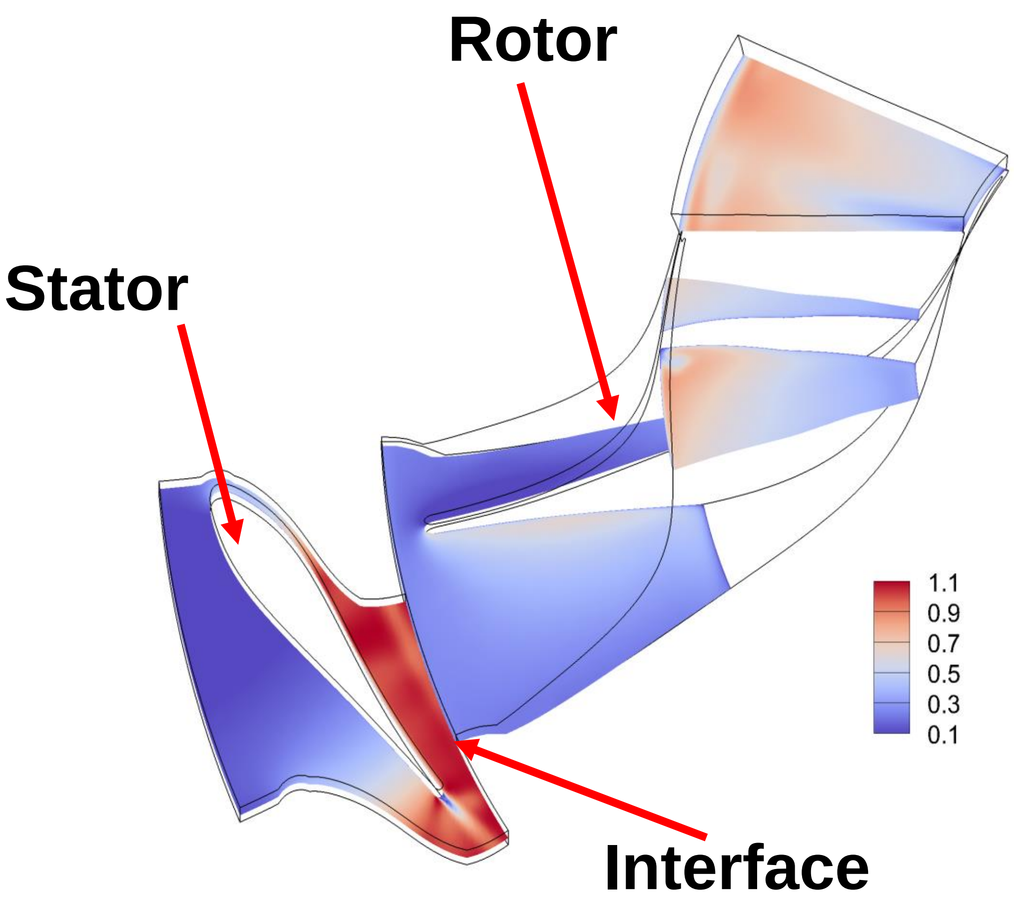
Loss breakdown comparison for two turbine designs



Visualisation of Mach number for a single Stator / rotor passage

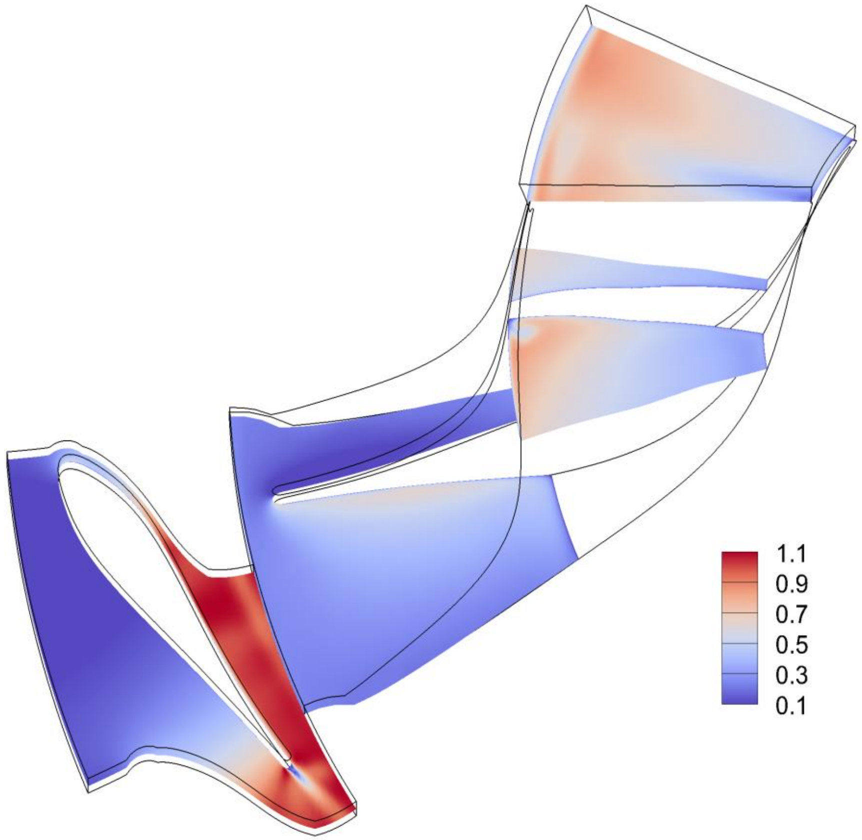
Solver pre-requisites

- Cylindrical coordinates
- Frame change
  - Many simulations will involve both stationary and rotating domains
- Compressible flow
  - Mach number is almost always above 0.3
- Component level simulations
  - Steady / Unsteady RANS formulations
  - Wall functions / implicit time stepping
- In-built capability for parametrization / optimization
- **Other local users!**



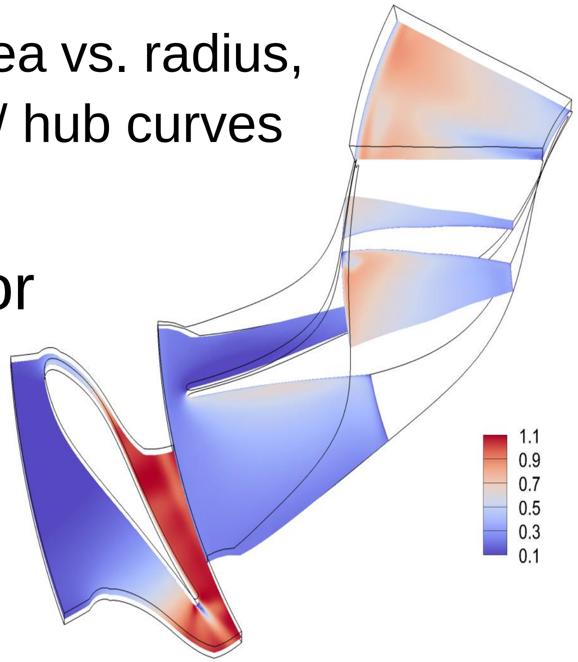
Functional segregation of machine

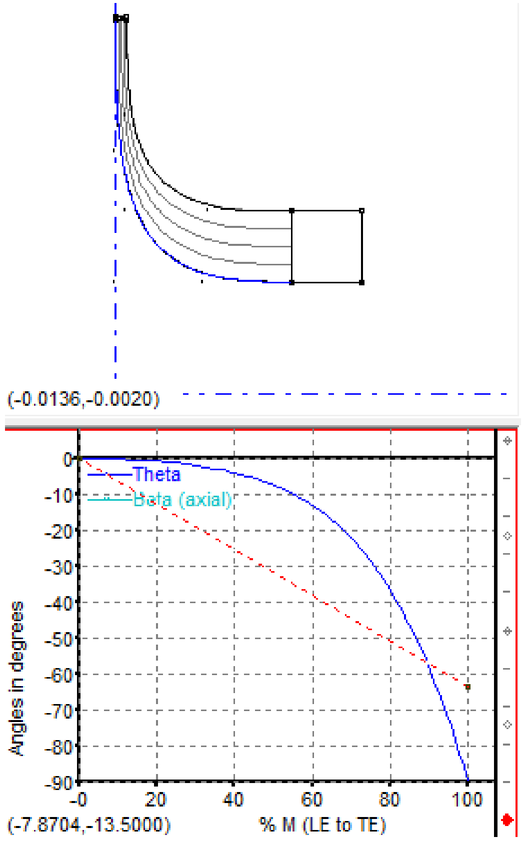
- Eliminate low loss regions (inlet and outlet)
  - Inlet is momentum dominated – good correlations for perofmance.
  - Diffuser is complicated to simulated due to separation – performance impact can be accounted for
- Reduction of domain through symmetry (periodic)
  - Appropriate interfaces are required (to account for different pitch).
  - Reduction in domain is at the price of fidelity – may be minor, but should be quantified.



Example CFD application – design of bladed components

- Analytical techniques generally not possible for passage design
- Geometry definition is indirect
  - i.e. may be interested in passage area vs. radius, however this is a function of shroud / hub curves and angle distribution.
- Lends to optimization (manual or automated)



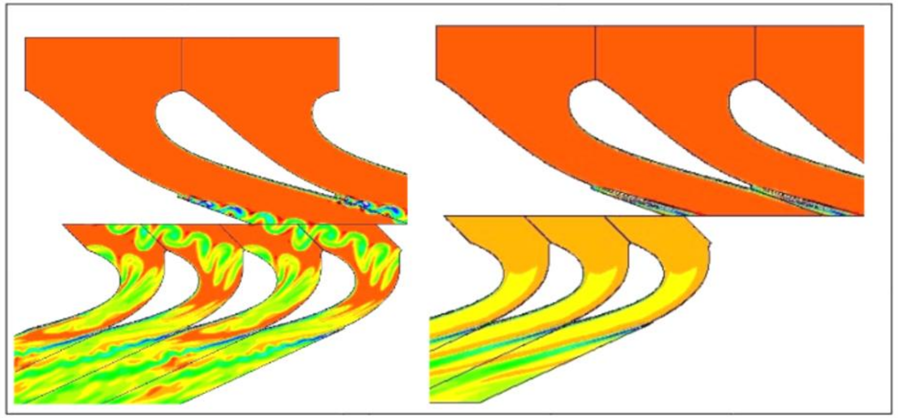


Radial inflow turbine passage definition (ANSYS BladeGen)

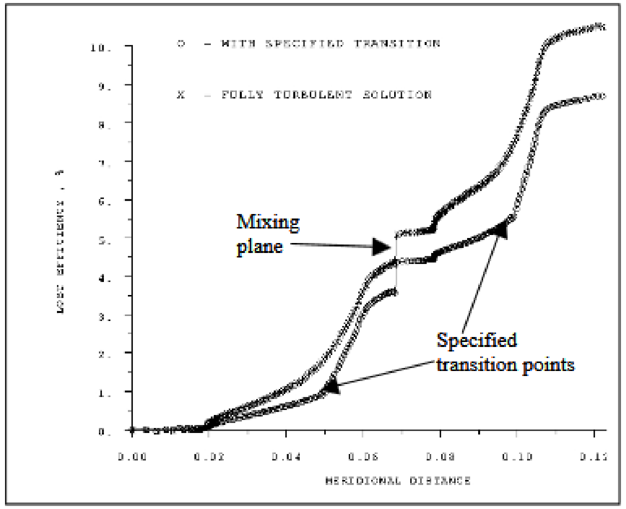
Steady / unsteady

- Increasing simulation fidelity generally increases accuracy
- Relative error in lower order simulations depends on type of turbomachine – <u>discrepancies should be quantified</u>

Interface	Stator / rotor passages	Solver type	# cells
Mixing Plane	1/1	RANS	$2x10^6$
Frozen rotor	1/1	RANS	$2x10^6$
Unsteady phase transform	1/1	URANS	$2x10^6$
None	5/4	URANS	$8.5x10^6$
None	19/16 (360^0)	URANS	$3.5x10^7$



Visualisation of entropy contours in stator / rotor; Unsteady vs. Mixing plane (Denton, 2010)



Plot of entropy along a stator / rotor passage (Denton, 2010)

Geometry representation and meshing

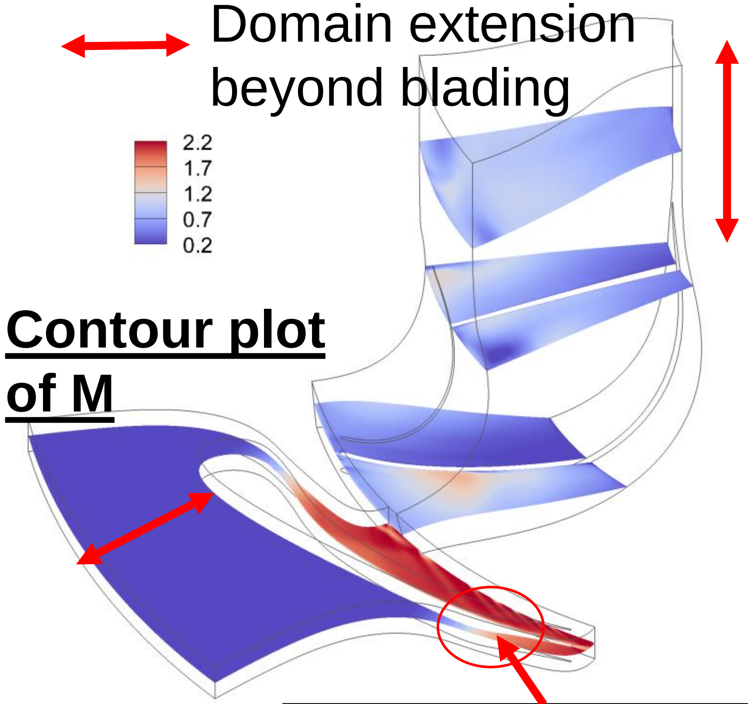
<u>A good mesh is a pre-requisite to a trustworthy solution</u>

- Geometry representation
  - The influence of boundary conditions needs to be appropriately accounted for (e.g. domain extension, placement of interfaces)
  - Trailing edge wakes are <u>intrinsically unsteady</u> – this can impact simulation convergence / blade loading predictions. Can be addressed through “cutting off” blade trailing edges / coarsening
  - <u>Reproduction</u>

<u>Example – 10 kw ORC turbine</u>

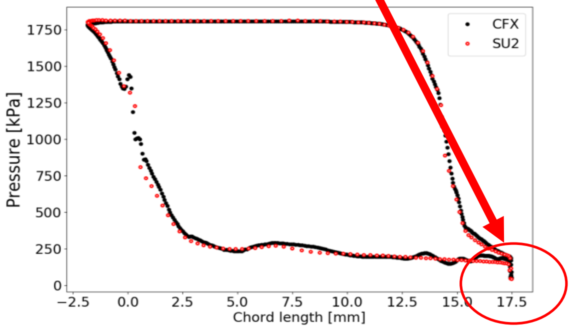
Domain extension beyond blading

<u>Contour plot of M</u>



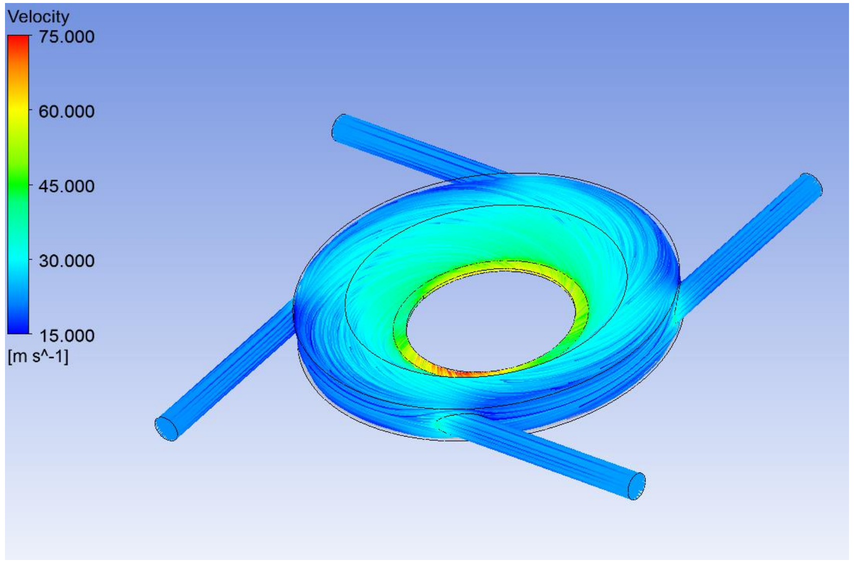
Incorrectly predicted!

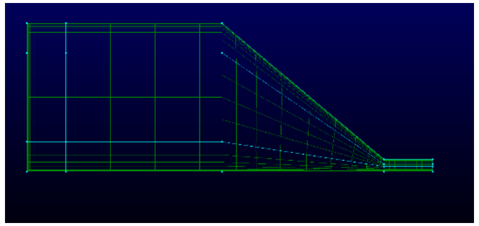
<u>Blade pressure loading</u>

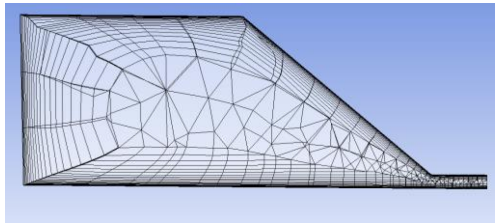


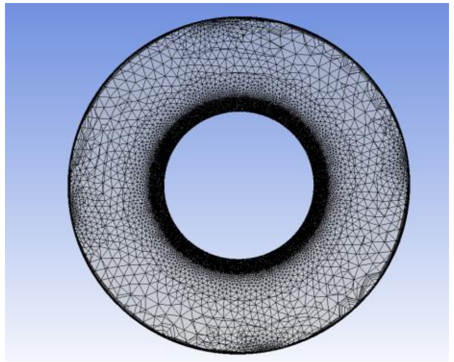
Geometry representation and meshing

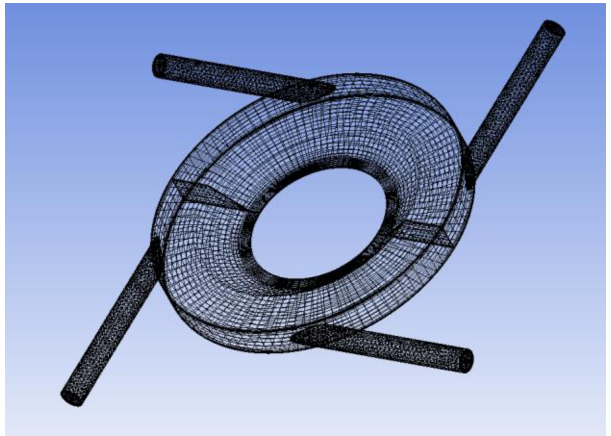
- ***<u>A good mesh is a pre-requisite to a trustworthy solution</u>***
- Structured grids are almost always favorable
  - May be time intensive to generate initial geometries
  - Consistent for comparative studies, grid comparison, and optimization









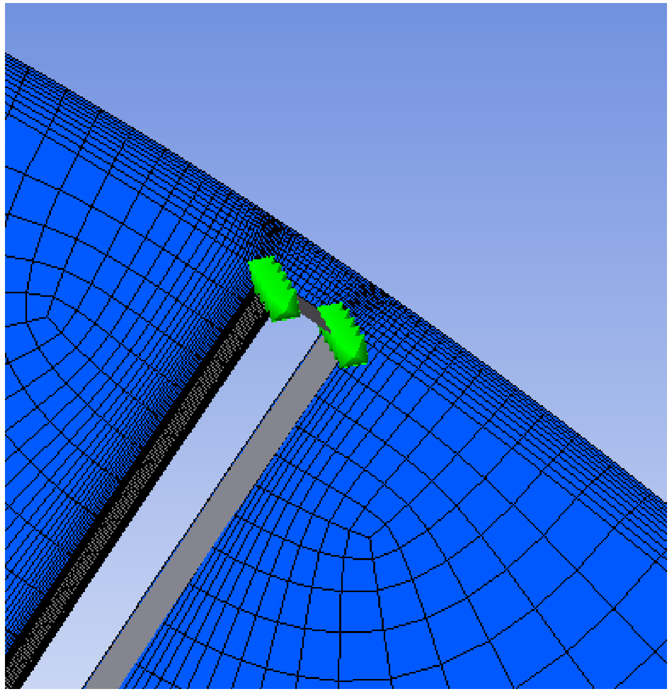


<u>Example – Structurally optimised inlet</u>

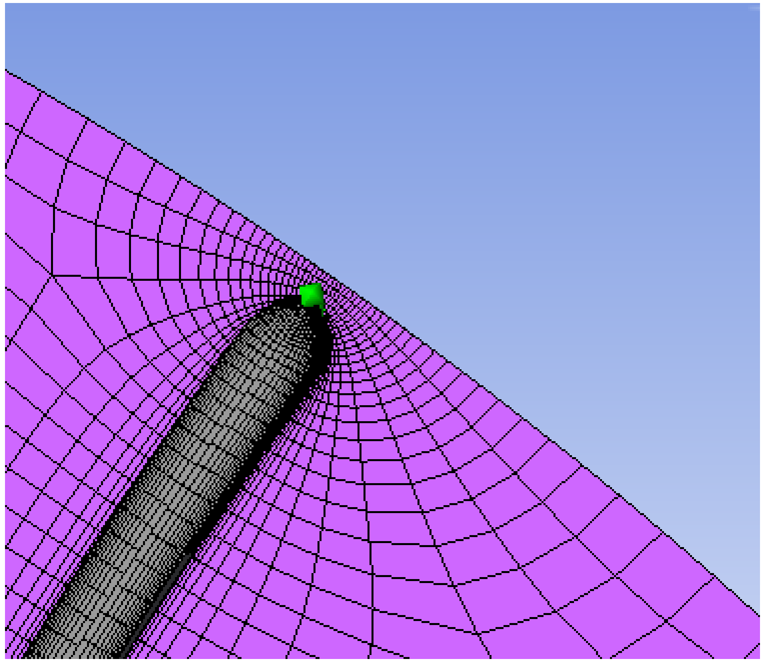
Geometry representation and meshing

- ***A good mesh is a pre-requisite to a trustworthy solution***
- Different geometries require different topologies
  - Topologies may have other un-intentional consequences / constraints

Example – Rotor blade inlet



Cut-off blade



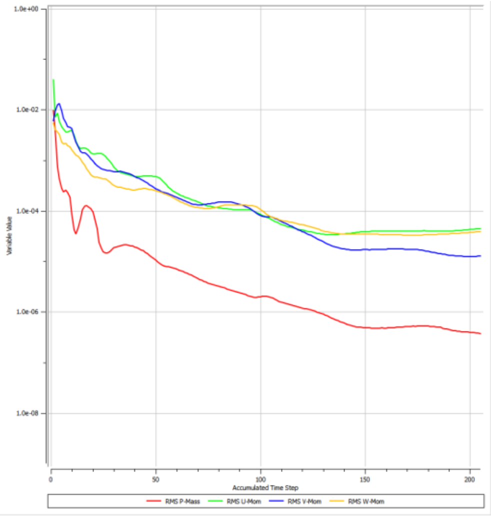
rounded blade

When is my simulation done?

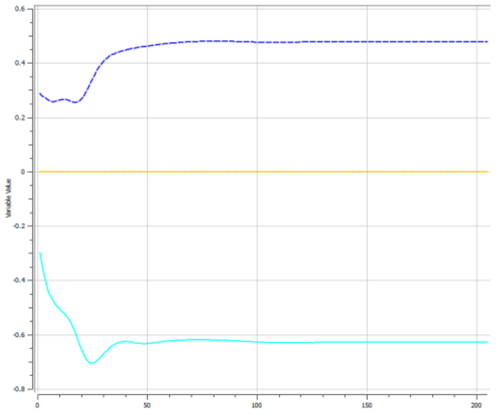
.. Depends on what you need from it

- Momentum and energy balances have different requirements
- Not usually indicated by residuals
  - Many commercial solvers have internal ramps (Pressure / shaft speed)
- Mass flow rate, or mass imbalance is usually a more reliable indicator *
  - Additionally provides an indicator of solution steadiness
- Appropriate performance parameters are also appropriate (e.g. power output / efficiency / rothalpy)

* When starting from scratch

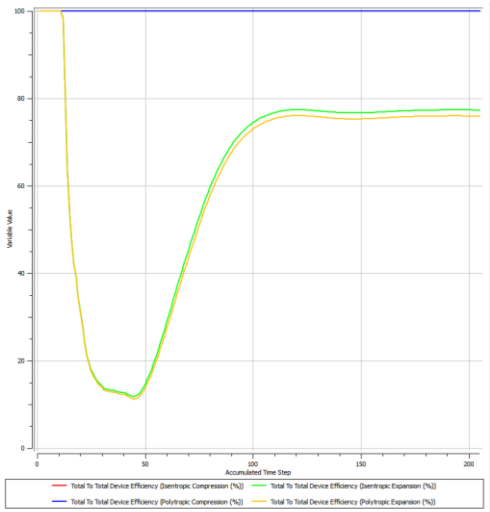


RMS residuals



Mass

Stage efficiency (energy)



<u>Example – re-started solution with change in numerical schemes – RANS</u> (courtesy G. Yagenegi)

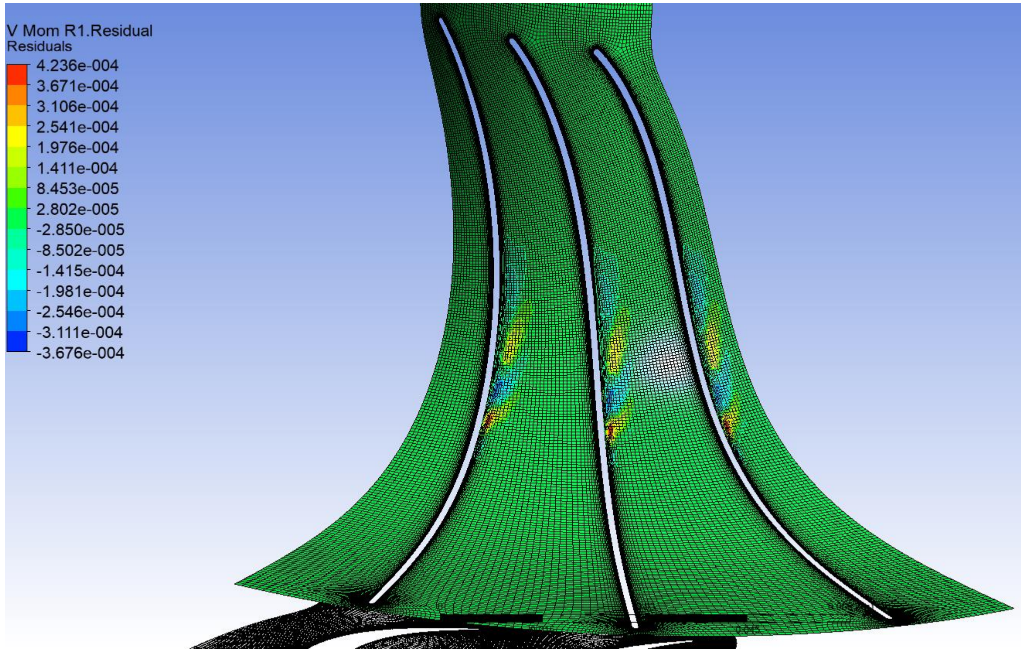
Why wont my simulation converge?

Potential problems

- Grid
  - Poor geometric representation
  - Poor geometry
  - Poor meshing
- Solver settings
- Flow may be intrinsically unsteady

Solutions

- Check solution against geometry
- Solver settings against known case
  - Is it rotating the correct way?
- Plot performance parameters vs time
- Try solving unsteady

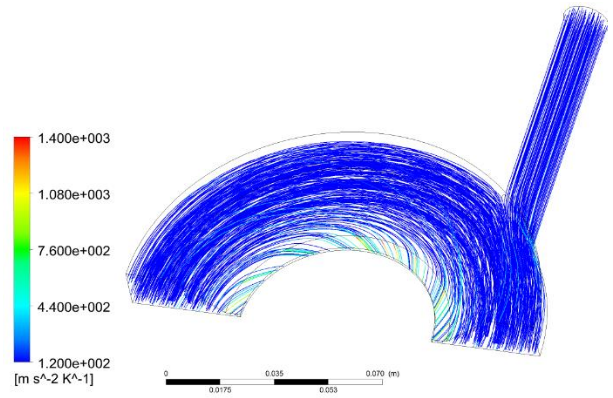


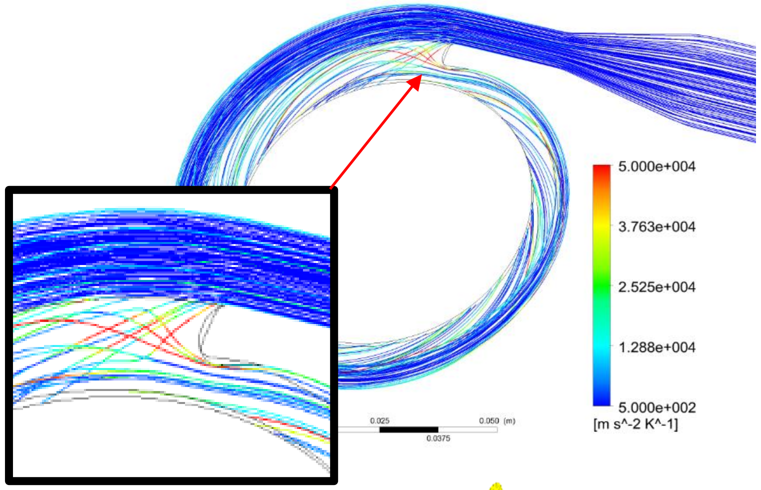
Example – plot of residuals on grid (courtesy G. Yagenegi)

Post processing – qualitative

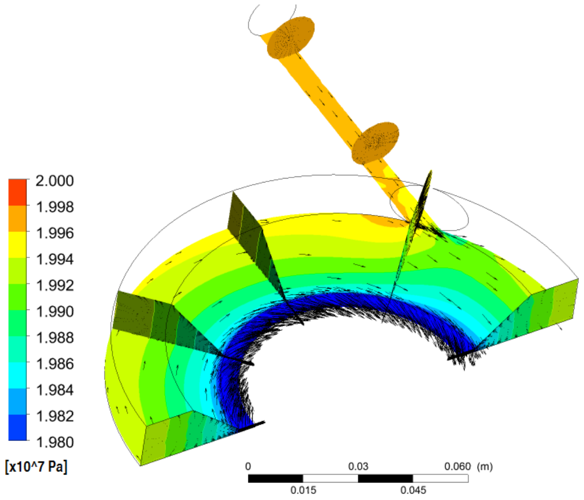
Example – Inlet delivery system

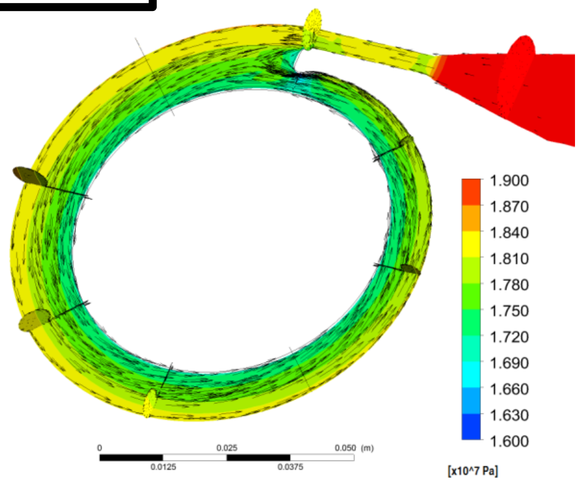
Streamline coloured by entropy gradient





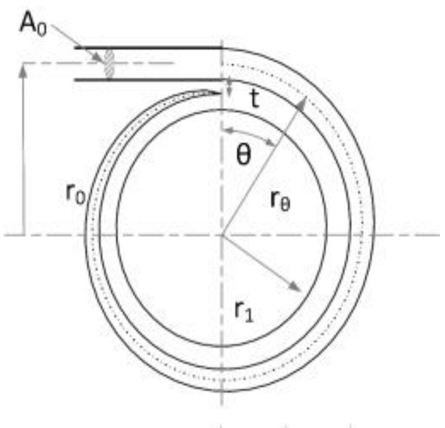
Static pressure contour and vector plots

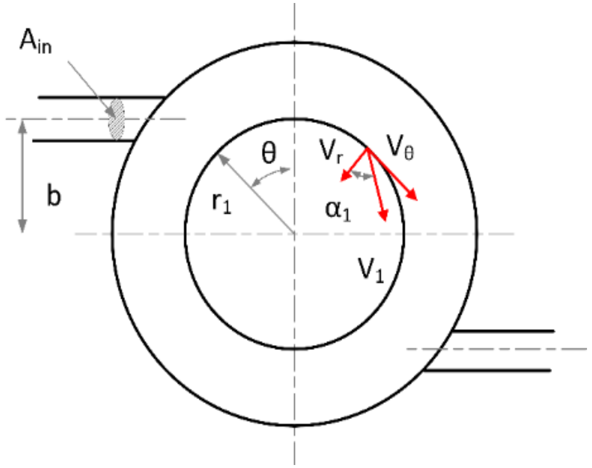




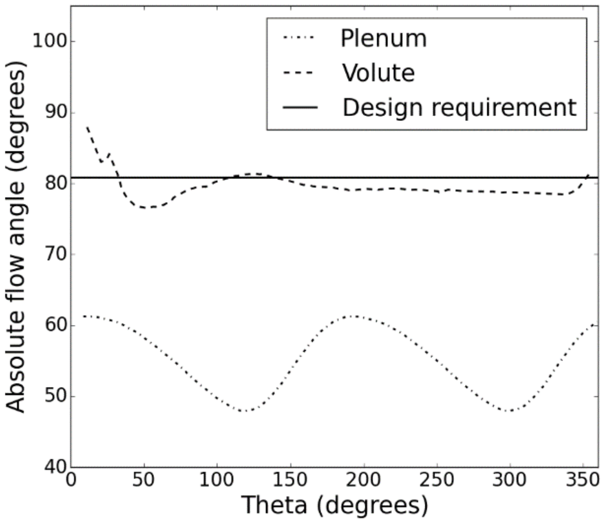
Post processing – quantitative

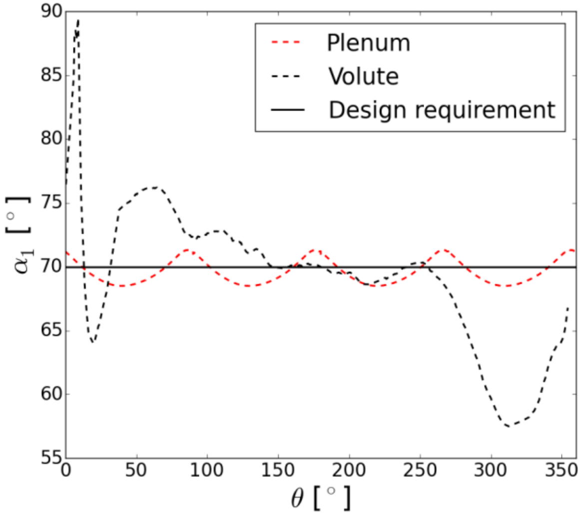
Example – Inlet delivery system





Flow angle variation with azimuthal angle (baseline vs refined)

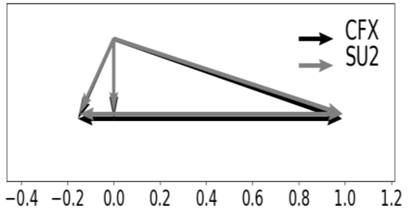


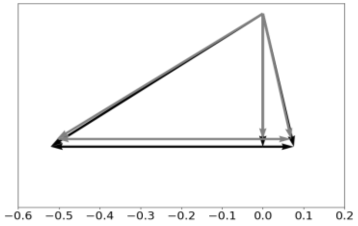


Parameter	Volute	Plenum
$h_{1,total}$ (kJ kg^{-1})	8385.11	8385.11
Δs (J kg^{-1} K^{-1})	8.06	0.49
P_1 (MPa)	17.15	19.72
$\Delta P_{1,total}$ (MPa)	0.83	0.05
$\dot{m}$ (kg s^{-1})	1.11	1.11
$\eta_{isentropic}$	73 %	80 %

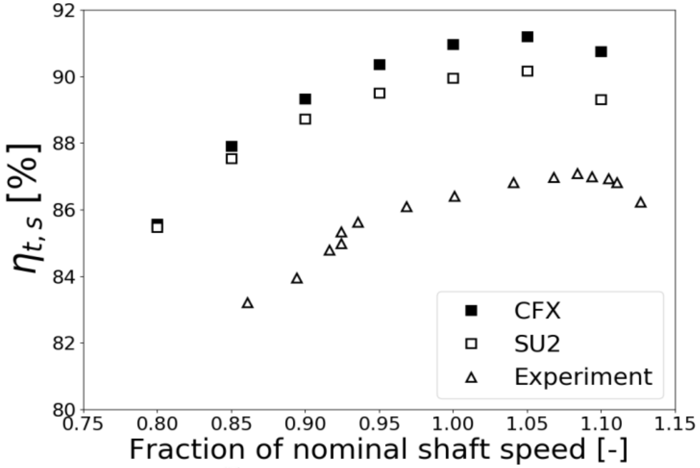
Solver comparison - 100kw turbine stage

Inlet and outlet Mach numbers

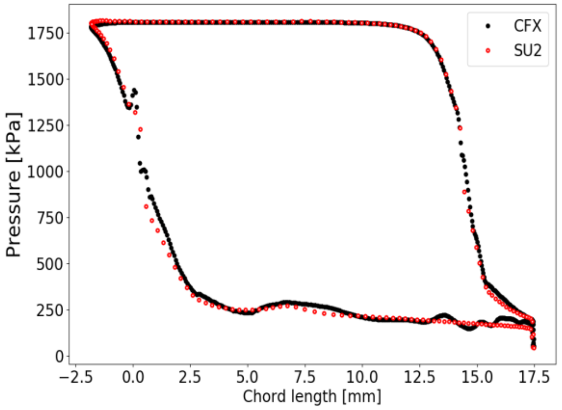




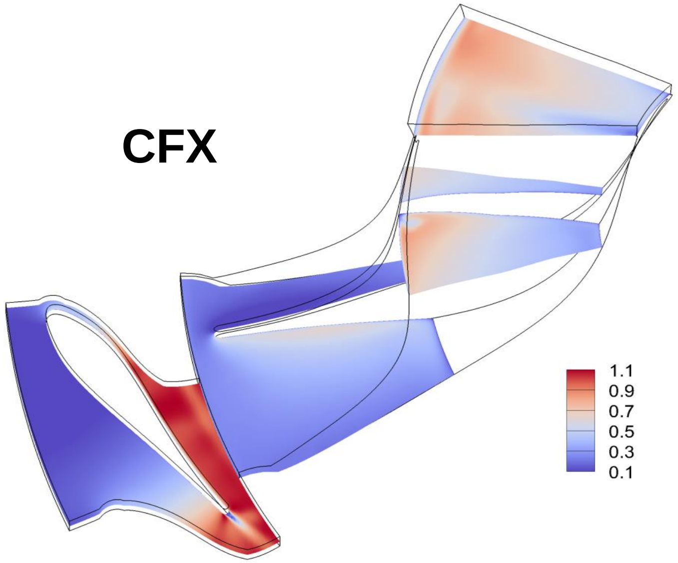
Stage performance



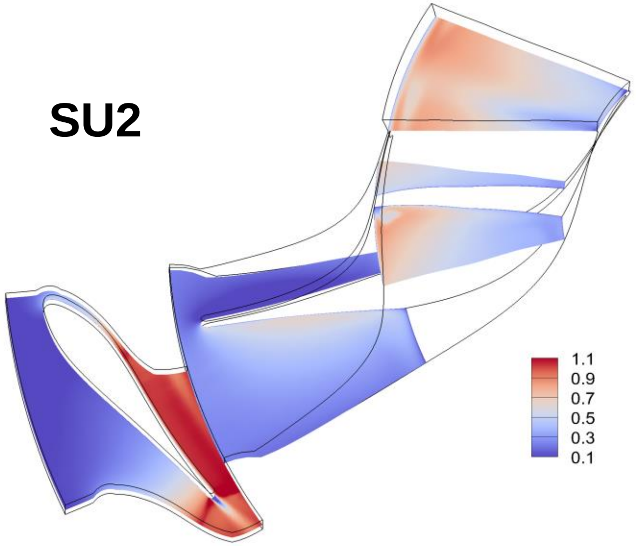
Stator blade loading



CFX



SU2



Postprocessing - summary

- Be quantitative
  - Utilise appropriate figures of merit
  - Extract trends
- Use qualitative post processing as a diagnostic tool
- Save templates! (.cst for qualitative / external scripts for quantitative)

CFD for turbomachinery - summary

- Invest the time in a good quality mesh
- Know the limitations of your model
  - Know the theory
  - Find the limits of your model
  - Experimentally validated if you can
- Use qualitative and quantitative post to their respective strengths
- Good results take time