Training Outline

- First Touch = First Success
- TCFD® Introduction
- TCFD® Inputs Requirements
- TCFD® GUI - TCFD Source
- TCFD® GUI - GENERAL + PHYSICS
- TCFD® Source - COMPONENTS
- TCFD® Source - MESH
- TCFD® Source - SIMULATION
- TCFD® Source - Boundary Conditions
- TCFD® Source - Post Processing
- TCFD® Manager - Simulation
- TCFD® Results
- Workshop
First Touch = First Success

- Both Windows and Linux
- Installation: (manual - Installation & First run)
  - Linux: > ./TCFD-19.10v1-r1941-linux64.sh -install
  - Windows: TCFD-19.10v1-r1941-win64-install.exe
  - Put the licence file into the installation directory
- Run the Axial Fan Simple case
  - Linux: .../TCFD-19.10/TurbomachineryCFD/tutorials/CFDProcessor/axial-fan-simple
  - Windows: C:\TCFD\19.10\tutorials\axial-fan-simple
- Have your first TCFD success
● Unique CFD simulation tool
● Data in - Data out
● Merges the best from open-source and commercial codes
  ○ Can be used both as “black box” and “white box”
Benefits of TCFD®

- Smart, focused, easy-to-use and affordable CFD Simulation tool
- For all rotating machinery (pumps, fans, compressors, turbines, ...)
- Both internal and external flows (virtual tunnel, piping, valves, buoyant flow, ...)
- Unique values:
  - Unlimited number of jobs or cores
  - Focused
  - Fully Automated
  - Tailored for optimization loops
TCFD® Deliverables

- TCFD® includes several components:
  - OpenFOAM® based software
  - CFD Processor® software - CFD Support - C++ (licensed)
  - TCFD® GUI filters - implemented in Paraview

- Extensive technical support

- Learning materials
Learning Materials

- TCFD web page:
- Webinars
  - https://cfdsupport.com/webinars.html
- Video tutorials
  - https://cfdsupport.com/tcfd-video-tutorials.html
- Ready-to-run tutorials with best practice setup
  - https://cfdsupport.com/download-cases.html
- PDF manual
TCFD® Workflow

• Inputs
  ○ **STL** (*triangulated surface*) *geometry* - mesh is generated by TCFD®
  ○ Ready-to-simulate meshes in: MSH (Fluent), CGNS or OpenFOAM format
  ○ Machine parameters - dimensions, axis of rotation, working conditions, …
  ○ Beware of “Garbage in, garbage out” !!!

○ TCFD® is not (yet) a tool for CAD-to-STL data conversion
TCFD® Workflow

- TCFD®
  - CFD Simulation setup - TCFD Source / configuration file *.tcfd
    - Simulation type, rotation speed, BCs, physics, ...
  - Processing - TCFD Manager / CFD Processor
    - Write case -> Mesh generation -> Run simulation -> Report generation
  - Reporting
    - Automatically generated html report = All at one place
    - Compare reports feature = One report for more simulations
TCFD® Inputs Requirements
TCFD® Inputs Requirements

- Important properties of **STL geometry** (STL) and **External meshes** (EXT)
  - Flow path geometry - STL, EXT
  - As simple as possible - STL, EXT
  - Component thinking - STL, EXT
  - Physical boundaries - STL, EXT
  - Watertight property - STL
  - Triangulation refinement - STL
- Clear geometry describing a flow path - “Wet Surface”
TCFD® Inputs - Simplicity

- Geometry should include important details only
- Any detail **may not** affect the overall machine parameters (efficiency,...)
- Any detail **do** increase mesh requirements and simulation cost

“Everything should be made as simple as possible, but not simpler.”

Albert Einstein
TCFD® Inputs - Component Thinking

- TCFD® fully uses a beauty of component approach
- Any design can be split into meaningful parts/volumes (components)
- Components of rotating geometries can be grouped into:
  - **Stators** - no shape restrictions
    - Pipes
    - Volutes
    - Diffusers
    - Vaned stators
    - Leakages
    - Virtual tunnels
    - Rooms
  - **Rotors** - solids of revolution
    - Impellers
    - Propellers
    - Wheels
    - Rims
    - Blades
- TCFD® automatically evaluates integral quantities at each interface between two components
- Splitting the geometry into more components could be convenient
● Radial pump example:
TCFD® Inputs - Component Thinking

- Radial pump can be disassembled into 4 components
TCFD® Inputs - Component Thinking

- Radial pump - inlet pipe, rotor, volute, outlet pipe
Each component holds a separate computational mesh or mesh region (imported or generated by TCFD)
TCFD® Inputs - Physical Boundaries

- Partitioning component boundary into physical boundaries
TCFD® Inputs - Physical Boundaries

- Partitioning component boundary into physical boundaries
  - Inlet, outlet, blade (pressure side, suction side, trailing edge, ...), hub, interfaces, ...
  - Detailed splitting allows finer tuning of mesh parameters
**TCFD® Inputs - Interfaces**

- **Component connection**: interface alignment
  - Important for connecting neighbouring components
TCFD® Inputs - Interfaces

- Interface is a part of boundary which connects two components
- Each component has its own interface
- Interfaces alignment of neighbouring components must be accurate
TCFD® Inputs - Interfaces

- TCFD® can simulate rotor/stator segment geometries
- Here, virtual full wheel interfaces must be accurately aligned
**TCFD®** Inputs - Watertight Property - STLs

- Extremely important for meshing (snappyHexMesh)
  - Watertight property or point-to-point correspondence
  - Tiny holes (much smaller than resulting cells) are allowed
TCFD® Inputs - Triangulation - STLs

- A surface triangulation has to be **fine-enough** to capture all the details
- Surface triangulation must be always finer than the resulting mesh
TCFD® Inputs - Triangulation Fineness

- Finer triangulation captures better all the curved surfaces and ensures accurate interface alignment.
TCFD® Inputs - Triangulation Fineness

- Finer triangulation (left image) ensures accurate alignment of interfaces

- Rotor outlet
- Volute inlet
● TCFD® GUI is implemented in ParaView 5.6.0 64-bit - https://www.paraview.org/
TCFD Source

TCFD Source holds settings of TCFD parameters and input geometry (components).

New TCFD Source can be created by:
- Clicking on the icon
- Sources -> TCFD Source

Menus for TCFD setup
TCFD Source

- Advanced settings button

This button enables advanced options

Advanced Meshing options menu
TCFD® Source - GENERAL

TCFD setup file (*.tcfd) in use

Button for checking possible errors in TCFD setup

Output Messages
Case setup OK!

Each type holds:
- specific setup parameters
- evaluation methods
- efficiency formulas

The solution is focused!
● **steady state** - steady state simulation = no physical time = looking for steady and averaged solution

● **transient**
  ○ *Dynamic mesh* - full transient simulation - real mesh motion - rotors are physically rotating
  ○ *MRF* - semi-transient simulation - no real mesh motion - mesh is static

**NOTE:** steady state + transient means that transient simulation is initialized by steady state simulation
Number of speedlines

Number of points per speedline

Number of iteration per point and given speedline
  ○ **Iteration** = single computation loop connected to the steady-state solver

Transient time to be simulated for each point and each speedline
  ○ **Transient time** = physical (real) time which is simulated by transient solver
TCFD® Source - PHYSICS - What is Speedline?

- **speedline** = group of points with common rotation speed
- **point** = one CFD simulation for particular setup (in/out BC, iteration / time, ...)

3 speedlines, 6 points each

1 speedline, 5 points
Physical model sets a general physics of the flow:
- Incompressible
- Compressible
- Heat Transfer

Gravitational acceleration is mainly applicable for:
- Heat Transfer model
**TCFD® Source - Fluid Properties - Incompressible**

<table>
<thead>
<tr>
<th>Physical model</th>
<th>Incompressible</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid name</td>
<td>air</td>
</tr>
<tr>
<td>Use fluid defaults</td>
<td></td>
</tr>
<tr>
<td>Dynamic viscosity</td>
<td>1.8e-5</td>
</tr>
<tr>
<td>Reference density</td>
<td>1.2</td>
</tr>
<tr>
<td>Reference pressure</td>
<td>101325</td>
</tr>
</tbody>
</table>

- **Type of a fluid**
  - For **air** and **water** user can check **Use fluid defaults** or set custom values

- **Dynamic viscosity** is the only parameter influencing the simulation

- **Reference density** and **Reference pressure** are used for adjusting the kinematic pressure \([m^2/s^2](p/\rho)\) back to pressure in Pascals
  - **NOTE:** Governing equation for incompressible solvers uses so called kinematic pressure
● **Compressible model** is applied for cases where compressible effects has to be taken into account, e.g., Mach > 0.3

● **Transonic** should be applied for transonic flows, i.e., Mach ~ 1
  ○ For air and water user can check **Use fluid defaults** or set custom values

● **Transport model** - user can choose two models for viscosity:
  ○ Constant
  ○ Sutherland

\[ \mu = \frac{A_s T^3}{T_s + T} \]
Heat Transfer can be used for compressible, subsonic flows, where heat transfer is a major subject of investigation.

Boussinesq approximation can be enabled for modeling heat transfer including buoyancy in flows with low temperature difference.
Cavitation risk can be used to request estimation of the cavitation, it is available for "water" machines only [postprocessing].

Multiphase cavitation enables use of a specialized cavitation solver for the transient phase of the calculation [water machines, transient simulation].
TCFD® Source - Fluid Properties - Age, Passive Scalars

- **Add passive scalar** allows user to add arbitrary number of passive scalars to the simulation. The scalar is computed by means of convection-diffusion equation.

- **Diffusivity type:**
  - Constant diffusivity $D$
  - Turbulent $D = \alpha_D \cdot \nu + \alpha_Dt \cdot \nu_t$

- **Calculate age** switches on calculation of the fluid age
TCFD® Source - Fluid Properties - Calculate comfort

- **PPD** (Predicted Percentage of Dissatisfied)
  - 5% is the lowest (best) value ;-
  - 100% max value

- **age** - gives time taken for a particle to travel from an inlet to the position in the domain

- **PMV** (Predicted Mean Vote)
  - Index predicting the mean value of the thermal sensation
    - -3: Cold
    - -2: Cool
    - -1: Slightly Cool
    - 0: Neutral (Comfort)
    - 1: Slightly Warm
    - 2: Warm
    - 3: Hot

- **Available for**: Physical model - Heat Transfer

<table>
<thead>
<tr>
<th>PHYSICS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time management</td>
</tr>
<tr>
<td>Fluid properties</td>
</tr>
<tr>
<td>✓ Calculate comfort (PMV, PPD)</td>
</tr>
</tbody>
</table>

- **Comfort clothing**: 0.5 clo (summer); 1.0 clo (winter) [m²k/W]
- **Comfort metabolic rate**: 1.0 met ~ 58 W/m²
- **Comfort external work**: usually 0 W
- **Comfort relative humidity**: 60 %
● **Turbulence model** to be used within the simulation
  ○ *kOmegaSST* - choice number one for turbomachinery
  ○ Other models can be implemented (scripting option)

● **Wall treatment**
  ○ *Standard wall functions* - for most industrial application (30 < y+ < 300)
  ○ *Low-Reynolds wall functions* - for very fine mesh (y+~1)
  ○ *Rough walls* - for rough wall modelling
TCFD® allows rotation of several disjoint regions
- Each frame holds axis of rotation
  - or axis of transformation for segment stators
- Rotating regions additionally holds rotation speed
TCFD® operates with a standard SI units, therefore the input geometry has to be always scaled to meters using a correct *Scale factor* parameter value.

In other words, *Scale factor* defines length units of input geometry.

*Number of components* parameter defines how many components builds the whole geometry.
TCFD® can create computational mesh from STL files

TCFD® can read meshes in MSH (Fluent), CGNS and OpenFOAM format
**Bounding box** enables definition of box-shaped computational domain

- Available for following simulation types:
  - stator, propeller, virtualTunnel
**TCFD® Source - Components - Geometry source**

- **STL geometry**
  - Directory with STL files
  - STL directory
  - Component name
  - Reference frame
  - No. periodic segments
  - Patches

<table>
<thead>
<tr>
<th>name</th>
<th>type</th>
<th>frame</th>
<th>grp</th>
<th>min ref</th>
<th>max ref</th>
<th>layers</th>
<th>mxp</th>
</tr>
</thead>
<tbody>
<tr>
<td>impeller-blades</td>
<td>blade</td>
<td>rotating</td>
<td>2</td>
<td>2</td>
<td>0</td>
<td>7</td>
<td></td>
</tr>
<tr>
<td>impeller-hub</td>
<td>hub</td>
<td>rotating</td>
<td>1</td>
<td>2</td>
<td>0</td>
<td>7</td>
<td></td>
</tr>
<tr>
<td>impeller-inflow</td>
<td>inlet</td>
<td>static</td>
<td>0</td>
<td>2</td>
<td>0</td>
<td>7</td>
<td></td>
</tr>
<tr>
<td>impeller-outflow</td>
<td>outletInterface</td>
<td>static</td>
<td>1</td>
<td>2</td>
<td>0</td>
<td>10</td>
<td></td>
</tr>
<tr>
<td>impeller-perA</td>
<td>rotationAMI</td>
<td>static</td>
<td>1</td>
<td>2</td>
<td>0</td>
<td>7</td>
<td></td>
</tr>
<tr>
<td>impeller-perB</td>
<td>rotationAMI</td>
<td>static</td>
<td>1</td>
<td>2</td>
<td>0</td>
<td>7</td>
<td></td>
</tr>
<tr>
<td>impeller-shroud</td>
<td>shroud</td>
<td>rotating</td>
<td>1</td>
<td>2</td>
<td>0</td>
<td>7</td>
<td></td>
</tr>
<tr>
<td>variedGate-blades</td>
<td>(not used)</td>
<td>static</td>
<td>?</td>
<td>?</td>
<td>0</td>
<td>7</td>
<td></td>
</tr>
<tr>
<td>variedGate-hub</td>
<td>(not used)</td>
<td>static</td>
<td>?</td>
<td>?</td>
<td>0</td>
<td>7</td>
<td></td>
</tr>
</tbody>
</table>

- **External mesh**
  - External Fluent mesh
  - MSH file
  - Mesh region
  - Component name
  - Reference frame
  - No. periodic segments
  - Patches

<table>
<thead>
<tr>
<th>name</th>
<th>type</th>
<th>frame</th>
<th>mxp</th>
</tr>
</thead>
<tbody>
<tr>
<td>BLADE_STAY</td>
<td>blade</td>
<td>static</td>
<td>?</td>
</tr>
<tr>
<td>HUB_STAY</td>
<td>hub</td>
<td>static</td>
<td>?</td>
</tr>
<tr>
<td>INFLOW</td>
<td>inlet</td>
<td>static</td>
<td>?</td>
</tr>
<tr>
<td>PER1</td>
<td>rotationAMI</td>
<td>static</td>
<td>?</td>
</tr>
<tr>
<td>PER2</td>
<td>rotationAMI</td>
<td>static</td>
<td>?</td>
</tr>
<tr>
<td>SHROUD_STAY</td>
<td>shroud</td>
<td>static</td>
<td>?</td>
</tr>
<tr>
<td>STAY_GUIDE</td>
<td>outletInterface</td>
<td>static</td>
<td>10</td>
</tr>
</tbody>
</table>

Parameters exclusive for mesh generation by snappyHexMesh
TCFD® Source - Components - Patches type

- **Physical inlets/outlets into/out of the whole geometry**
- **Couple of patches within one component to be connected (right click) by a given transformation:**
  - `internalAMI` - no transformation (overlapping patches)
  - `translationAMI` - translation (2D periodic cases)
  - `rotationAMI` - rotation (3D vaned rotors and stators)
  - `empty` - for 2D meshes only (front and back patches)
  - `symmetry` - for symmetry planes (“half a car”)
  - `wall` - physical wall
  - `wallSlip` - wall with no friction

- **Exclusive names for the type wall**
  - Useful for postprocessing, mesh refinement, etc.

- Types for connecting (right click) two neighbouring components:
  - `inletInterface/outletInterface` - couple of patches, where flow direction is known
  - `freestreamInterface` - couple of patches where flow direction is not known or heavy recirculation appears

<table>
<thead>
<tr>
<th>name</th>
<th>type</th>
<th>rot</th>
<th>mxp</th>
</tr>
</thead>
<tbody>
<tr>
<td>periodicA</td>
<td>inlet</td>
<td></td>
<td></td>
</tr>
<tr>
<td>periodicB</td>
<td>inlet</td>
<td></td>
<td></td>
</tr>
<tr>
<td>rotor-inlet</td>
<td>internalAMI</td>
<td></td>
<td></td>
</tr>
<tr>
<td>rotor-outlet</td>
<td>translationAMI rotationAMI</td>
<td></td>
<td></td>
</tr>
<tr>
<td>rotor-inlet-hub</td>
<td>empty</td>
<td></td>
<td></td>
</tr>
<tr>
<td>rotor-blade</td>
<td>wall</td>
<td></td>
<td></td>
</tr>
<tr>
<td>rotor-shroud</td>
<td>shroud</td>
<td></td>
<td></td>
</tr>
<tr>
<td>rotor-inlet-shroud</td>
<td>blade</td>
<td></td>
<td></td>
</tr>
<tr>
<td>rotor-hub</td>
<td>wallPressureSide</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>bladeSuctionSide</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>bladeLeadingEdge</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>bladeTrailingEdge</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>bladeHubFillets</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>bladeShroudFillets</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>bladeCap</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>cutWater</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>inletInterface</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>outletInterface</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>freestreamInterface</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
TCFD® Source - Components - ROT and MXP

- Reference frame assignment for the whole component (rotor vs stator)
- Reference frame assignment - definition of static or rotating frame
- Exclusive for *Interface patch types
  - mxp = 0 - frozen rotor
  - General usage: Connected components are not rotational symmetric and/or flow recirculation occurs at the interface
  - mxp > 0 - mixing plane approach
  - mxp = number of averaging planes
  - General usage: For rotational symmetric components with smooth flow and/or segment geometry
TCFD® Source - Components - Patch light

- Each boundary can be highlighted by double-click on boundary name.
**TCFD® Source - Components - Mesh parameters**

- **minRef** - minimum level of refinement which is applied on the patch
- **maxRef** - maximum level of refinement which is applied on the patch
- **layers** - number of boundary layers to be added, layer addition has to be enabled by the switch
- **Background mesh size** - size of a default (largest) cell (level 0)
- **Internal point** - geometry part to be meshed

**Background mesh size**
- Level 0: (~0.005³ m)
- Level 1: (~0.0025³ m)
- Level 2: (~0.00125³ m)

**Internal point**
- Pipe inlet: 0.00897732
- Pipe outlet: 0.199624
- Pipe wall: 0.958961

**Components**
- Pipe
- Reference frame: static
- No. periodic segments: 1
- Scale factor:
  - pipe_inlet: 1
  - pipe_outlet: 1
  - pipe_wall: 1
- Number of components: 1
- Geometry source: One multi-solid STL file
- STL file: tutorial/CFDSupportFOAM/trunk/tutorials/CFDProcessor/pipe/STL/pipe.stl...
TCFD® - Mesh parameters - Minimalistic setup

<table>
<thead>
<tr>
<th>name</th>
<th>type</th>
<th>frame</th>
<th>grp</th>
<th>min ref</th>
<th>max ref</th>
</tr>
</thead>
<tbody>
<tr>
<td>pipe_inlet</td>
<td>inlet</td>
<td>static</td>
<td></td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>pipe_outlet</td>
<td>outlet</td>
<td>static</td>
<td></td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>pipe_wall</td>
<td>wall</td>
<td>static</td>
<td></td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

Background mesh size: 0.0025

687983 cells

<table>
<thead>
<tr>
<th>name</th>
<th>type</th>
<th>frame</th>
<th>grp</th>
<th>min ref</th>
<th>max ref</th>
</tr>
</thead>
<tbody>
<tr>
<td>pipe_inlet</td>
<td>inlet</td>
<td>static</td>
<td></td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>pipe_outlet</td>
<td>outlet</td>
<td>static</td>
<td></td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>pipe_wall</td>
<td>wall</td>
<td>static</td>
<td></td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

Background mesh size: 0.005

186171 cells

<table>
<thead>
<tr>
<th>name</th>
<th>type</th>
<th>frame</th>
<th>grp</th>
<th>min ref</th>
<th>max ref</th>
</tr>
</thead>
<tbody>
<tr>
<td>pipe_inlet</td>
<td>inlet</td>
<td>static</td>
<td></td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>pipe_outlet</td>
<td>outlet</td>
<td>static</td>
<td></td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>pipe_wall</td>
<td>wall</td>
<td>static</td>
<td></td>
<td>2</td>
<td>2</td>
</tr>
</tbody>
</table>

Background mesh size: 0.01

142568 cells
TCFD® - Components - Advanced options

- Background mesh positioning

| Euler alpha | 0 |
| Euler beta  | 0 |
| Euler gamma | 0 |

| Euler alpha | 90 |
| Euler beta  | 50 |
| Euler gamma | 0 |
TCFD® - Components - Advanced options

- **Cylindrical background mesh**

<table>
<thead>
<tr>
<th>Cylindrical mesh</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Background mesh size</td>
<td>2</td>
</tr>
<tr>
<td>Cylindrical radii</td>
<td>140 $r_0$</td>
</tr>
<tr>
<td>Cylindrical grading</td>
<td>$g_0$</td>
</tr>
<tr>
<td>Cylindrical warp</td>
<td>1</td>
</tr>
</tbody>
</table>

  $w=1 \quad d=r_0$ (cylinder)
  $w=0 \quad d=r_0/\sqrt{2}$ (square)

- **Enables cylindrical background mesh**
TCFD® - Components - Advanced options

- Advanced meshing parameters - meshing small gaps

<table>
<thead>
<tr>
<th>Leading edge gap</th>
<th>0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Trailing edge gap</td>
<td>0.5</td>
</tr>
<tr>
<td>Blade cap gap</td>
<td>0.2</td>
</tr>
</tbody>
</table>

- Distance between:
  - bladeLeadingEdge and inlet(Interface)
  - bladeTrailingEdge and outlet(Interface)
  - bladeCap and shroud (clearance gap)

- If set, TCFD automatically increases level of refinement to impose fine-enough meshing at these blade* parts

<table>
<thead>
<tr>
<th>name</th>
<th>type</th>
<th>frame</th>
<th>grp</th>
<th>min ref</th>
<th>max ref</th>
<th>layers</th>
<th>map</th>
</tr>
</thead>
<tbody>
<tr>
<td>CCTurbocharger_Co2_BladeFillet</td>
<td>bladeHubFillets</td>
<td>rotating</td>
<td>2</td>
<td>2</td>
<td>3</td>
<td>?</td>
<td></td>
</tr>
<tr>
<td>CCTurbocharger_Co2_BladeLE</td>
<td>bladeLeadingEdge</td>
<td>rotating</td>
<td>2</td>
<td>2</td>
<td>3</td>
<td>?</td>
<td></td>
</tr>
<tr>
<td>CCTurbocharger_Co2_BladePS</td>
<td>bladePressureSide</td>
<td>rotating</td>
<td>2</td>
<td>2</td>
<td>3</td>
<td>?</td>
<td></td>
</tr>
<tr>
<td>CCTurbocharger_Co2_BladeSS</td>
<td>bladeSuctionSide</td>
<td>rotating</td>
<td>2</td>
<td>2</td>
<td>3</td>
<td>?</td>
<td></td>
</tr>
<tr>
<td>CCTurbocharger_Co2_BladeTE</td>
<td>bladeTrailingEdge</td>
<td>rotating</td>
<td>2</td>
<td>2</td>
<td>3</td>
<td>?</td>
<td></td>
</tr>
<tr>
<td>CCTurbocharger_Co2_BladeTip</td>
<td>bladeCap</td>
<td>rotating</td>
<td>2</td>
<td>2</td>
<td>3</td>
<td>?</td>
<td></td>
</tr>
<tr>
<td>CCTurbocharger_Co2_Hub</td>
<td>hub</td>
<td>rotating</td>
<td>1</td>
<td>1</td>
<td>3</td>
<td>?</td>
<td></td>
</tr>
<tr>
<td>CCTurbocharger_Co2_Inflow</td>
<td>inlet</td>
<td>static</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>?</td>
<td></td>
</tr>
<tr>
<td>CCTurbocharger_Co2_Outflow</td>
<td>outlet</td>
<td>static</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>?</td>
<td></td>
</tr>
<tr>
<td>CCTurbocharger_Co2_PeriodicPS</td>
<td>rotationAMI</td>
<td>static</td>
<td>1</td>
<td>1</td>
<td>0</td>
<td>?</td>
<td></td>
</tr>
<tr>
<td>CCTurbocharger_Co2_PeriodicSS</td>
<td>rotationAMI</td>
<td>static</td>
<td>1</td>
<td>1</td>
<td>0</td>
<td>?</td>
<td></td>
</tr>
<tr>
<td>CCTurbocharger_Co2_Shroud</td>
<td>shroud</td>
<td>static</td>
<td>1</td>
<td>1</td>
<td>3</td>
<td>?</td>
<td></td>
</tr>
</tbody>
</table>

413 420 cells
TCFD® - Components - Advanced options

- Advanced meshing parameters - meshing small gaps - single blade STL
  - Previous approach cannot be used (no bladeCap, …. STL provided)
    - Default parameters are too rough to capture the clearance

<table>
<thead>
<tr>
<th>name</th>
<th>type</th>
<th>frame</th>
<th>grid</th>
<th>min ref</th>
<th>max ref</th>
<th>layers</th>
<th>model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ckturbinecharger_Co2_Blade</td>
<td>blade</td>
<td>rotating</td>
<td>@</td>
<td>2</td>
<td>2</td>
<td>3</td>
<td>?</td>
</tr>
<tr>
<td>Ckturbinecharger_Co2_Hub</td>
<td>hub</td>
<td>rotating</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>3</td>
<td>?</td>
</tr>
<tr>
<td>Ckturbinecharger_Co2_Inflow</td>
<td>inlet</td>
<td>static</td>
<td>@</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>?</td>
</tr>
<tr>
<td>Ckturbinecharger_Co2_Outflow</td>
<td>outlet</td>
<td>static</td>
<td>@</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>?</td>
</tr>
<tr>
<td>Ckturbinecharger_Co2_Periodic_Ps</td>
<td>static</td>
<td>rotation</td>
<td>AMI</td>
<td>1</td>
<td>1</td>
<td>0</td>
<td>?</td>
</tr>
<tr>
<td>Ckturbinecharger_Co2_Periodic_CS</td>
<td>static</td>
<td>rotation</td>
<td>AMI</td>
<td>1</td>
<td>1</td>
<td>0</td>
<td>?</td>
</tr>
<tr>
<td>Ckturbinecharger_Co2_Shroud</td>
<td>shroud</td>
<td>static</td>
<td>@</td>
<td>1</td>
<td>1</td>
<td>3</td>
<td>?</td>
</tr>
</tbody>
</table>

- Shroud refinement:
  - 1 850 359 cells

- Blade refinement:
  - 3 546 802 cells

301 028 cells
Advanced meshing parameters - meshing small gaps - single blade STL

- Use gap refinement feature

- Use gap refinement **locally refines** the mesh up to the preset level in gaps between surfaces for which the **gap ref** level is defined
**TCFD® - Components - Advanced options**

- **Advanced meshing parameters**
  
- **Level:**
  - Level of refinement to be applied

- **User refinement regions:**
  - Enables defining additional refinement regions

- **Type:**
  
- **Mode:**
  - Where to apply the mesh refinement
**Components**: Number of layers

- Different number of layers can be assigned to each boundary

**Meshing options**: BL parameters

- Boundary layer parameters are common for the whole component
**Meshing options:** BL parameters

- **These values are** relative/absolute
- **Sizes relative to the cell size next to the patch for which adding layers is defined**
- **Sizes in absolute values**

- **Minimum boundary thickness** below which BL is not added (default is OK)
- **Meshing options**: BL parameters

- **first layer** - thickness of the first layer in the boundary layer (the layer on the wall)

- **final layer** - is the thickness of the last layer in the boundary layer

- **whole boundary layer** - is the overall thickness of the whole boundary layer

- **Expansion ratio** - expansion ratio of neighbouring layers
TCFD® - Mesh parameters - Boundary Layer

- **Meshing options**: BL parameters

- **Feature angle** - above $\alpha$, the boundary layers are collapsed

- The remaining parameters are set to default (best-practice) values.
Meshing options:

- Background mesh size: 2 (mm)

- BL parameters:
  - First layer thickness has a relative size 0.2 compared to the cell size on the blade.
  - Blade refinement is set to 3 (min ref=max ref), i.e., cell size at the wall is 0.25 (2/2^3)
  - Therefore, the first layer thickness with is 0.05 (0.25*0.2)

Equivalent parameters for absolute sizes:
TCFD® - Boundary Layer - Example - Axial fan

- **Meshing options**: BL parameters
  - **Equivalent results** for both setups

<table>
<thead>
<tr>
<th>name</th>
<th>type</th>
<th>frame</th>
<th>grp</th>
<th>min ref</th>
<th>max ref</th>
<th>layers</th>
<th>mxf</th>
</tr>
</thead>
<tbody>
<tr>
<td>impeller-blades</td>
<td>blade</td>
<td>rotating</td>
<td>3</td>
<td>3</td>
<td>5</td>
<td>?</td>
<td></td>
</tr>
</tbody>
</table>

![Meshing options and equivalent results images]
• **Meshing options**: Absolute sizes
  
  • **BL collapses** at the refined parts of the blade
TCFD® - Boundary Layer - Example - Axial fan

- **Meshing options**: Relative sizes
  - **BL well added** at the refined parts of the blade
- Interface matching is very important for overall accuracy
- Meshing parameters of the neighbouring interfaces should be similar
- Try to keep cell sizes at the both interfaces similar

### Radial pump

#### Interfaces meshing - Example

<table>
<thead>
<tr>
<th>Interface</th>
<th>Type</th>
<th>Boundary</th>
<th>Size</th>
<th>16</th>
<th>4</th>
</tr>
</thead>
<tbody>
<tr>
<td>inletPipe-inlet</td>
<td>inlet</td>
<td>static</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>inletPipe-outlet</td>
<td>outletInterface</td>
<td>static</td>
<td>2</td>
<td>2</td>
<td></td>
</tr>
<tr>
<td>inletPipe-wall</td>
<td>wall</td>
<td>static</td>
<td>3</td>
<td>3</td>
<td></td>
</tr>
<tr>
<td>Background mesh size</td>
<td></td>
<td></td>
<td>16</td>
<td>16</td>
<td>16</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Interface</th>
<th>Type</th>
<th>Boundary</th>
<th>Size</th>
<th>8</th>
<th>8</th>
<th>8</th>
</tr>
</thead>
<tbody>
<tr>
<td>volute-inlet</td>
<td>inletInterface</td>
<td>static</td>
<td>2</td>
<td>2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>volute-outlet</td>
<td>outlet</td>
<td>static</td>
<td>0</td>
<td>0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>volute-wall</td>
<td>wall</td>
<td>static</td>
<td>1</td>
<td>1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Background mesh size</td>
<td></td>
<td></td>
<td>8</td>
<td>8</td>
<td>8</td>
<td></td>
</tr>
</tbody>
</table>

- $16/2^2 = 4/2^0$
- $4/2^1 = 8/2^2$
● Example of a nice alignment:
- Example of a rougher alignment (still ok for simulation, a bit larger interpolation error)
- SnappyHexMesh parameters can be found in the advanced section
- This section is divided into several subsections
- Each subsection defines parameters per component
**Feature edges included angle**
- Generating feature edges for better snapping and local edge refinement
- 0 - no edges
- 180 - all edges
**Surface hook-up**

- Can patch non-watertight geometries
- Always check the surface after hook-up!

 STL manipulation

(STL manipulation parameters are used only for components that are meshed from STL files. They control the additional pre-processing of the STL files.)

<table>
<thead>
<tr>
<th>component1</th>
<th>component2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Feature edges included angle</td>
<td>95</td>
</tr>
</tbody>
</table>

**Surface hook-up**
**TCFD® - SnappyHexMesh - Advanced parameters**

- **Maximum allowed number of all cells during the meshing process** - for large meshes increase these values!!!
- **Cell refinement procedure is stopped if the number of cells to be refined is less then this value**
- **Parameter for re-balancing algorithm (no need to change)**
- **Cells between levels**: number of cells between different refinement levels

![Cells between levels diagram](image)
**Feature edges level:** refinement on edges

<table>
<thead>
<tr>
<th>name</th>
<th>type</th>
<th>frame grp</th>
<th>min ref</th>
<th>max ref</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ahmed-body-35</td>
<td>wall</td>
<td>3</td>
<td>3</td>
<td>3</td>
</tr>
</tbody>
</table>

Feature edges level: 0

Feature edges level: 5
**Resolve feature angle:** Feature angle refinement = to apply maximum level of refinement to cells that can see intersections whose angle exceeds this value.
• **Snap mesh** and **Mesh quality**
  ○ Advanced parameters set to our best-practice values
  ○ Any change can affect mesh quality

### Advanced Snappy Hex mesh options

<table>
<thead>
<tr>
<th>Component1</th>
<th>Component2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Snap mesh</td>
<td></td>
</tr>
<tr>
<td>√ Snap mesh</td>
<td></td>
</tr>
<tr>
<td>nSmoothPatch</td>
<td>3</td>
</tr>
<tr>
<td>Tolerance</td>
<td>2</td>
</tr>
<tr>
<td>Solve iter</td>
<td>30</td>
</tr>
<tr>
<td>Relax iter</td>
<td>5</td>
</tr>
<tr>
<td>Feature snap iter</td>
<td>10</td>
</tr>
<tr>
<td>√ Implicit feature snap</td>
<td></td>
</tr>
<tr>
<td>√ Explicit feature snap</td>
<td></td>
</tr>
<tr>
<td>√ Multi region feature snap</td>
<td></td>
</tr>
</tbody>
</table>

### Mesh quality

Mesh quality parameters are used only for components that are meshed from STL files. They restrict the cell shapes and control the balance between a faithful representation of the geometry and numerical properties of the mesh.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max non-ortho</td>
<td>65</td>
</tr>
<tr>
<td>Max boundary skewness</td>
<td>20</td>
</tr>
<tr>
<td>Max internal skewness</td>
<td>3</td>
</tr>
<tr>
<td>Max concave</td>
<td>80</td>
</tr>
<tr>
<td>Min vol</td>
<td>1e-16</td>
</tr>
<tr>
<td>Min tet quality</td>
<td>-1e+30</td>
</tr>
<tr>
<td>Min area</td>
<td>1e-13</td>
</tr>
<tr>
<td>Min twist</td>
<td>0.02</td>
</tr>
<tr>
<td>Min determinant</td>
<td>0.001</td>
</tr>
<tr>
<td>Min face weight</td>
<td>0.02</td>
</tr>
<tr>
<td>Min vol ratio</td>
<td>0.01</td>
</tr>
<tr>
<td>MinTriangleTwist</td>
<td>-1</td>
</tr>
<tr>
<td>Smooth scale</td>
<td>4</td>
</tr>
<tr>
<td>Error reduction</td>
<td>0.75</td>
</tr>
</tbody>
</table>
Mesh quality check
- TCFD® generates a report including mesh statistics:
  - **Aspect ratio** - high values appear in very fine boundary layers, slows down the convergence
  - **Skewness** - higher values may impair accuracy (always depends on application)
  - **Non-orthogonality** - parameter affecting both accuracy and stability:
    - $nO < 70$ - safe range
    - $70 < nO < 90$ - be careful, requires special treatment (nonOrthoCorrectors, numerical schemes)
    - $nO > 90$ - bad mesh, cannot be used for simulation
• **Have a perfect STLs**
  ○ High resolution, watertight, enough boundary splitting, avoid sharp edges and tiny gaps
• **Start from rough meshes**
  ○ Start from rougher mesh to see if your setup and mesh definition are OK
  ○ Refine the mesh gradually and step-by-step to follow your requirements
• **Add boundary layer (BL) at the end**
  ○ BL should be done for a nice mesh without BL, otherwise you waste your time
• **Always check the mesh visually**
  ○ After the meshing, visually check all the geometry details which may be problematic
• **Be patient! ;-)**
TCFD® Source - SIMULATION - Solver

- **Numerical order**
  - *first* - for the very first simulation and complicated flows
  - *second* - if the *first* goes well use *second*

- The simulation is considered converged when **efficiency** and **inlet/outlet flow rates** are changing less than **Convergence tolerance** over the last **Averaging window** parameter

\[
\max_i |\eta_i - \langle \eta \rangle| \leq \xi \langle \eta \rangle, \\
\max_i |\phi_{in,i} - \langle \phi_{in} \rangle| \leq \xi \langle \phi_{in} \rangle, \\
\max_i |\phi_{out,i} - \langle \phi_{out} \rangle| \leq \xi \langle \phi_{out} \rangle, \\
\max_i \frac{||\phi_{in,i}|-|\phi_{out,i}||}{||\phi_{in,i}|+|\phi_{out,i}||/2} \leq \xi.
\]
● Averaging window: 200

● Averaging window: 25
**TCFD® Source - SIMULATION - Solver - Time step**

- Transient solver is based on a robust PIMPLE (PISO-SIMPLE) algorithm allowing larger time stepping \((Co >> 1)\)

- **Time step: Adaptive**
  - Adaptive time stepping based on the Courant number
    \[ Co = \frac{\Delta t}{2V} \sum_{i \in \text{faces}} |\Phi_i| \]

- **Time step: Constant**
  - Constant time step for the whole simulation
  - Large time step -> lower time resolution, lower accuracy, faster simulation
  - Small time step -> higher time resolution, higher accuracy, slower simulation
Users can incorporate their own OF solvers into TCFD framework.

On Linux systems, one can set easily parallel simulation on several computational nodes.

User can set particular number of slots/cores to be used on given node for simulation.

Prevents migration of parallel processes between cores, possible resulting in some speedup.
### Under-relaxation factors
- lower values increases stability of computation
- lower values slows down speed of convergence
- always use default values if you are not an expert
- default values ensure robustness

### Field bounding limits
- increases stability - usually at an early stage of simulation
- solver cuts off all the value out of the range and set them to the limits
PIMPLE algorithm settings

<table>
<thead>
<tr>
<th>Setting</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inner correctors</td>
<td>1</td>
</tr>
<tr>
<td>Outer correctors</td>
<td>30</td>
</tr>
<tr>
<td>Pressure tolerance</td>
<td>0.001</td>
</tr>
<tr>
<td>Velocity tolerance</td>
<td>0.0001</td>
</tr>
</tbody>
</table>

- **Inner correctors**
  - Number of iterations for correcting pressure field without re-calculation of a momentum matrix (default: 1)

- **Outer correctors**
  - Number of iterations for re-calculating of the pressure-momentum coupling (~SIMPLE)
  - 1 ~ PISO (default: 30)

- **Pressure tolerance**
  - Convergence tolerance for pressure initial residuals during the outer loop (default: 1e-3)

- **Velocity tolerance**
  - Convergence tolerance for velocity components initial residuals during the outer loop (default: 1e-4)

- **Advanced mode**
- **Available for transient simulations**
TCFD® Source - Simulation Controls - PIMPLE

Single time step computation
(window size = number of outer correctors)

p tolerance

U tolerance
- **Absolute tolerances** for linear solvers during a SIMPLE iteration
- **Relative tolerances** of linear solvers during a SIMPLE iteration
- The linear solver is assumed solved if either relative or absolute tolerances are satisfied
- **Final absolute tolerances** for very the last iteration of PIMPLE (transient simulation)
● **Non-ortho correctors:**
  ○ Can be increased up to 3 if simulation shows large time step continuity error

● **Consistent solver**
  ○ Enables SIMPLEC (SIMPLE-Consistent) formulation
  ○ It does not need strong relaxation resulting in faster convergence and more robust solution
  ○ Highly experimental, cannot be used for general cases
  ○ Applicable for “nice” flows:
    ■ 1st order schemes and coarse meshes
    ■ Nice and smooth flows without heavy recirculations (e.g., BEP points)
It can significantly improve convergence speed and simulation time
● **Scripting** gives user a tool for adjusting TCFD simulations by a python script
● Any python tool can be used
● Includes TCFD-specific functions `SetEntry` and `WriteFile` for accessing OF-like dictionaries

```python
patchForDistanceRefinement = 'surface.stl'
distanceRefinementLevels = '((0.2 3) (0.5 2) (1.0 1))'
CoSnappyDict = 'meshFactory/component1/system/snappyHexMeshDict'
CoSnappySubDict = 'castellatedMeshControls/refinementRegions/' + patchForDistanceRefinement

SetEntry(CoSnappyDict, CoSnappySubDict + '/mode', 'distance')
SetEntry(CoSnappyDict, CoSnappySubDict + '/levels', distanceRefinementLevels)
WriteFile(CoSnappyDict)
```
TCFD® Source - Boundary Conditions
TCFD® Source - BC: Inlet/Outlet

- **Inlet BC:**
  - Mass flow rate
  - Directed mass flow rate
  - Volumetric flow rate
  - Directed volumetric flow rate
  - Total pressure
  - Fixed velocity
  - Velocity profile
  - Fan pressure
  - Opening

- **Outlet BC:**
  - **Velocity like BC:**
    - Mass flow rate
    - Directed mass flow rate
    - Volumetric flow rate
    - Directed volumetric flow rate
    - Fixed velocity
    - Velocity profile
  - **Pressure like BC:**
    - Total pressure
    - Fixed pressure
    - Fixed mean pressure
    - Fan pressure (for **stator** only)

- **Special BC:**
  - Opening (flow direction is unknown)
## TCFD® Source - BC: Inlet/Outlet

- Typical combinations of BC inlet/outlet:

<table>
<thead>
<tr>
<th>INLET</th>
<th>OUTLET</th>
<th>NOTE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total pressure</td>
<td>Fixed / mean <em>(static)</em> pressure</td>
<td></td>
</tr>
<tr>
<td>Flow rate / Velocity</td>
<td>Fixed / mean <em>(static)</em> pressure</td>
<td>Most robust</td>
</tr>
<tr>
<td>Total pressure</td>
<td>Flow rate / Velocity</td>
<td>Allowed for frozen rotor (MXP = 0) only</td>
</tr>
<tr>
<td>Total pressure</td>
<td>Outlet vent</td>
<td>Designed for compressors, blowers, turbo fans, ...</td>
</tr>
</tbody>
</table>
There are two variants of static pressure BC at the outlet:

**Fixed pressure**
- Sets uniform value of static pressure
- Constant values within the whole simulation (for all iterations or time steps)
- More stable, suitable for “long” outlets, where a flow is uniform enough

**Fixed mean pressure**
- Sets the average value, i.e., the particular distribution depends on the flow at the boundary
- Only the mean value is constant, values can vary locally
- Less stable; applicable for “short” outlets, where the static pressure distribution cannot be considered as uniform
Velocity profile at the inlet can be defined.

- Direction along which a profile will be set.
- First column in csv file has a meaning of distance from the point [0,0,0] along the direction, i.e., in this case it is basically the z-coordinate.
- 2\textsuperscript{nd}, 3\textsuperscript{rd} and 4\textsuperscript{th} column stores x, y and z-component of the velocity vector at the inlet.
TCFD® Source - BC: Inlet/Outlet - Velocity profile
TCFD® Source - BC: Inlet/Outlet - Fan pressure

- Fan pressure BC can mimic a real fan characteristic
- The input has a list format
  - 1st line defines number of items (lines) in the list
  - Each item is a couple holding:
    - \((\text{vol}_\text{flow}[\text{m}^3/\text{s}] \ \delta_p\text{Tot}[\text{Pa or Pa/rho}])\)
- This BC sets the total pressure to \(p_0 + \delta_p\text{Tot}[\text{Pa or Pa/rho}]\) based on the actual volumetric flow rate
TCFD® Source - BC: Inlet/Outlet - Opening

- Opening BC can be used for an open domains where the direction of the flow is unknown at the inlet/outlet boundaries

- Total pressure value for inlet flow regions

- Turbulent quantities values for inlet flow regions
BC: Inlet/Outlet - Opening
TCFD® Source - BC: Inlet/Outlet

- BC setup example: incompressible case (pump)

- Reference values (real conditions at inlet/outlet) have to be correctly defined in *Physics* section.

- For incompressible cases zero pressure at the inlet/outlet is a standard
TCFD® Source - BC: Inlet/Outlet

- BC setup example - Compressible simulation (turbine)

- For compressible simulation, temperature at the inlet has to be set
- No reference values in physics are needed
TCFD® Source - BC: Inlet/Outlet

- BC setup example - Compressible simulation (compressor)

- Outlet vent BC for compressor/fan simulations

\[ p_{BC} = p_0 + \frac{1}{2} \rho R |U|^2 \]
TCFD® Source - BC: Inlet - Turb. quantities

- Inlet values of turbulent model specific variables can be computed from selected quantities:
  - Turbulent intensity and hydraulic diameter
  - Turbulent intensity and length scale
  - Turbulent intensity and viscosity ratio
  - Turbulent viscosity ratio (for SA)
  - Model quantities

\[ k = \frac{3}{2} (U_{avg} I)^2 \]
\[ \epsilon = C_\mu \frac{k^{\frac{3}{2}}}{l} \]
\[ \omega = \frac{k^{\frac{1}{2}}}{C_\mu^\frac{3}{4} l} \]
\[ \epsilon = C_\mu \frac{\rho k^2}{\mu} \left( \frac{\mu_t}{\mu} \right)^{-1} \]
\[ \omega = \frac{\rho k}{\mu} \left( \frac{\mu_t}{\mu} \right)^{-1} \]
\[ \tilde{v} = \mu \left( \frac{\mu_t}{\mu} \right) \]
**TCFD® Source - BC: Wall**

- Compressible and Heat Transfer simulation offers following Wall BCs:

  - **Adiabatic wall**
  - **Fixed temperature**
  - **Fixed power**
  - **Fixed heat flux**
  - **Fixed heat transfer coeff**

### Adiabatic Wall

- **Assign to patches**: 1:Chair, 1:Couch, 1:Table, 1:Top-Room
- **Temperature BC Type**: Adiabatic wall

### Fixed Temperature

- **Temperature BC Type**: Fixed temperature
- **Assign to patches**: 1:BeerMug

### Fixed Heat Flux

- **Temperature BC Type**: Fixed heat flux
- **Heat flux**: -50 Wm²

### Fixed Heat Transfer Coeff

- **Temperature BC Type**: Fixed heat transfer coeff
- **Heat transfer coefficient**: 60 Wm²K⁻¹
- **Wall layers**
  - **Thickness [mm]**: 1.00, 1.00
  - **Thermal conduct**: 0.62, 0.032
**TCFD® Source - BC: Interface Conditions**

- **Interface conditions** *(in advanced mode)* offers user to set the **Pressure jump** at any couple of interfaces.

### BOUNDARY CONDITIONS

- **Inlet**
- **Outlet**
  - **Interface conditions**
    - Number of interface conditions: 1
    - Interfaces: 1:pipe_pipe_interface+2:pipe_pipe_interface
    - Interface condition type: Pressure jump
    - Pressure jump:
      - speedline 1
      - point 1: 2000 Pa
      - point 2: 5000 Pa
TCFD® Source - Initial Condition & Sim. Controls

- Initial conditions = reasonable values
- Mapped = initialization by previous simulation
- Turbulent quantities can be initialized by the inlet values
TCFD® Source - Post Processing
## TCFD® Source - Post-processing - Report

<table>
<thead>
<tr>
<th>Section</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Sections</strong></td>
<td>(advanced menu) List of sections to appear in the report</td>
</tr>
<tr>
<td><strong>Report units</strong></td>
<td>main units for final html report</td>
</tr>
<tr>
<td><strong>Stream path</strong></td>
<td>path given by components labels for plotting graphs at components interfaces</td>
</tr>
<tr>
<td><strong>Additional data files</strong></td>
<td>external csv files for comparisons with measurements, external data, previous simulations, ...</td>
</tr>
<tr>
<td><strong>Efficiency probes</strong></td>
<td>which parts will be used for efficiency evaluation, each efficiency probe generates its own html report</td>
</tr>
<tr>
<td><strong>Forces</strong></td>
<td>evaluation of forces, force coefficients and torques for given patches</td>
</tr>
</tbody>
</table>

### Efficiency probes

<table>
<thead>
<tr>
<th>Probe Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>inlet patches</td>
<td>torques on patches</td>
</tr>
<tr>
<td>1:impeller-inflow</td>
<td>torques on inlet patches</td>
</tr>
<tr>
<td>1:impeller-bladeps</td>
<td>torques on blade patches</td>
</tr>
<tr>
<td>1:impeller</td>
<td>torques on impeller</td>
</tr>
</tbody>
</table>

### Forces

<table>
<thead>
<tr>
<th>Force Component</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>patches</td>
<td>Forces</td>
</tr>
<tr>
<td>liftX</td>
<td>Forces</td>
</tr>
<tr>
<td>liftY</td>
<td>Forces</td>
</tr>
<tr>
<td>liftZ</td>
<td>Forces</td>
</tr>
<tr>
<td>dragX</td>
<td>Forces</td>
</tr>
<tr>
<td>dragY</td>
<td>Forces</td>
</tr>
<tr>
<td>dragZ</td>
<td>Forces</td>
</tr>
</tbody>
</table>
**TCFD® Source - Post-processing - Samples**

- **Surface samples**: TCFD® can export pressure and/or temperature field for any geometry part.

- Exported values are stored in raw format:
  - `postProcessing/surfaceSample-0/1000/p_surfacesSet.raw`
  - `postProcessing/surfaceSample-0/1000/T_surfacesSet.raw`
**Probes:** TCFD® can export any field at any point location

- Exported values are stored in raw format:

  `postProcessing/probe-0/0/p`
  `postProcessing/probe-1/0/URel`
• **Blade to blade views** - automatically rendered blade-to-blade views
• **Group by point** - B2B views ordering
• **Meridional averages** - automatically rendered meridional averages
TCFD® can write mesh and results into CGNS file format

- **Snapshot interval and fields** - time interval for saving intermediate results (**fields**) - for transient simulation only
New feature offers the export of forces at all boundary patches

It mainly serves as a support for external FEA and FSI tools

- It automatically exports the values of forces on every face for every patch
- It saves the values in CSV format, each line includes data for individual face:
  - Coordinate of the face center
  - Force (components) on the face
  - Area of the face

/TCFDCase/postProcessing/surfaceQuantities/1000/rotor_rotor-blade_wall/surfaceQuantities.csv
User can include post-processing features for live monitoring:

- Efficiency probes
- Forces
- Probes

### Efficiency probes

<table>
<thead>
<tr>
<th>Efficiency probes</th>
<th>torque patches</th>
<th>monitor</th>
</tr>
</thead>
<tbody>
<tr>
<td>inlet patches</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1:rotor-inlet</td>
<td>1:rotor-inlet-hub,1:rotor-blade,1:rotor-hub</td>
<td></td>
</tr>
<tr>
<td>2:stator-outlet</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Fields</th>
<th>X coor</th>
<th>Y coor</th>
<th>Z coor</th>
<th>monitor</th>
</tr>
</thead>
<tbody>
<tr>
<td>p,U</td>
<td>0.025</td>
<td>0.105</td>
<td>-0.015</td>
<td></td>
</tr>
<tr>
<td>p,U,k,pTot</td>
<td>0.035</td>
<td>0.115</td>
<td>0.015</td>
<td>✓</td>
</tr>
</tbody>
</table>
TCFD® Manager - Simulation
TCFD® GUI - TCFD Manager

- Gives user full control over the workflow
- It manages:
  - Case writing
  - Mesh generation
  - Simulation run
  - Live monitoring
  - Report generation
  - Results visualization

Clicking on **Settings** item enables **TCFD Manager** button.
TCFD® GUI - TCFD Manager

1. Meshing
2. Calculation
3. Report generation

- Runs all the steps:
- Writes TCFD case
- Writes and delete old results
- Manual results visualization
TCFD® GUI - TCFD Manager - Advanced mode

- Advanced mode allows run-time adjustment of:
  - Time step
  - Relaxation parameters
  - Non-ortho correctors
  - PIMPLE parameters

### Manual Run: Run-time tuning

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>DeltaT [s]</td>
<td>0.0174533</td>
</tr>
<tr>
<td>Pressure relaxation</td>
<td>0.4</td>
</tr>
<tr>
<td>Velocity relaxation</td>
<td>0.7</td>
</tr>
<tr>
<td>Turb relaxation</td>
<td>0.3</td>
</tr>
<tr>
<td>Non-ortho correctors</td>
<td>0</td>
</tr>
<tr>
<td>PIMPLE inner correctors</td>
<td>2</td>
</tr>
<tr>
<td>PIMPLE outer correctors</td>
<td>1</td>
</tr>
<tr>
<td>PIMPLE pressure tolerance</td>
<td>0.001</td>
</tr>
<tr>
<td>PIMPLE velocity tolerance</td>
<td>0.0001</td>
</tr>
</tbody>
</table>

- Suitable for parameter tuning
- Not convenient for final simulation
TCFD® GUI - TCFD Manager - Live Monitoring

- Residuals plot
- Quantities plots

- Any quantity from the list can be selected
- Any number of plots can be created
TCFD® Results - HTML Report

- Can be generated any time during a simulation - **Light Report**
- At the end of a simulation - **Full Report**
- Contains relevant quantities for each **Machine type**
  - Simulation statistics
  - Efficiency (formulas in PDF manual)
  - Pressure difference
  - etc.
TCFD® results - CSV files

- All the evaluated quantities are tabulated in CSV files
  - .../Directory name/postProcessing/efficiency-X/0
  - `efficiency.csv` - values for all solver iterations
  - `efficiency-final.csv` - final values (when solver is finished)

- There are two columns for each quantity:
  - Instant values for given time (first column)
  - Averaged values over the last *Averaging window*

- Useful for advanced analysis -> direct import into excel sheets
  - Quantities have always fixed position, e.g., columns 2,3 always hold efficiency values

<table>
<thead>
<tr>
<th>time,</th>
<th>efficiency[-],</th>
<th>average[-],</th>
<th>massFlowIn[kg/s],</th>
<th>average[kg/s],</th>
<th>massFlowOut[kg/s],</th>
<th>average[kg/s],</th>
</tr>
</thead>
<tbody>
<tr>
<td>500,</td>
<td>0.6362959,</td>
<td>0.6367227,</td>
<td>0.216,</td>
<td>0.216,</td>
<td>0.2160035,</td>
<td>0.2160035,</td>
</tr>
<tr>
<td>1000,</td>
<td>0.6604231,</td>
<td>0.6606596,</td>
<td>0.2399999,</td>
<td>0.2399999,</td>
<td>0.2400033,</td>
<td>0.2400033,</td>
</tr>
<tr>
<td>1500,</td>
<td>0.6751947,</td>
<td>0.6754181,</td>
<td>0.2880004,</td>
<td>0.2880004,</td>
<td>0.2880051,</td>
<td>0.2880051,</td>
</tr>
<tr>
<td>2000,</td>
<td>0.6560014,</td>
<td>0.6561028,</td>
<td>0.3120001,</td>
<td>0.3120001,</td>
<td>0.3120058,</td>
<td>0.3120058,</td>
</tr>
<tr>
<td>2500,</td>
<td>0.6061445,</td>
<td>0.6062409,</td>
<td>0.3359999,</td>
<td>0.3359999,</td>
<td>0.3360065,</td>
<td>0.3360065,</td>
</tr>
</tbody>
</table>
TCFD® Results Visualization

- Loading results:

In TCFD Manager

New GUI window, by opening `case (case.foam)` file in your favourite file explorer
TCFD® - Batch Mode

- TCFD® includes program **CFDProcessor**, which is responsible for all the process behind the GUI
- **CFDProcessor** can be run from a command line
  - The executable is placed in: C:\TCFD\19.04\bin
  - Can be added to “Path” in Environmental Variables
- **Usage:**
  - Run command line (cmd)
  - Type C:\TCFD\19.04\bin\CFDProcessor for help message
- **Typical run:**
  - CFDProcessor -setup setup.tcfd -dir myDir -allrun
Thank You for Your Attention!