User manual for CFturb0 software
This manual describes the usage of the software CFturbo and corresponds to the online help with regards to content.

© CFturbo GmbH, 2023

All rights reserved. No parts of this work may be reproduced in any form or by any means - graphic, electronic, or mechanical, including photocopying, recording, taping, or information storage and retrieval systems - without the written permission of the publisher.

Products that are referred to in this document may be either trademarks and/or registered trademarks of the respective owners. The publisher and the author make no claim to these trademarks.

While every precaution has been taken in the preparation of this document, the publisher and the author assume no responsibility for errors or omissions, or for damages resulting from the use of information contained in this document or from the use of programs and source code that may accompany it. In no event shall the publisher and the author be liable for any loss of profit or any other commercial damage caused or alleged to have been caused directly or indirectly by this document.
# Table of Contents

## Part I CFTurbo

## Part II What's new in CFTurbo

## Part III General

1. Installation
2. Licensing
3. Batch mode
4. Project structure
5. Windows Explorer integration
6. Logging
7. Troubleshooting

## Part IV Getting started
Part V Menu

1 File ................................................................................................................................... 97

Create new design ................................................................. 97
Stage designer ........................................................................ 99
Open/ Save design .................................................................. 100

2 PROJECT .............................................................................. 101

General .................................................................................. 101
Project information .............................................................. 101
Global setup ........................................................................... 103
Performance prediction ......................................................... 109
Euler based ........................................................................... 113
Casey/Robinson ..................................................................... 118
Undo ....................................................................................... 119

Additional ............................................................................... 120

Export .................................................................................... 120
Remarks ................................................................................ 126
Basic ...................................................................................... 127
STL ......................................................................................... 130
Tetrahedral volume mesh ....................................................... 131

CAD, CAM ................................................................................ 132
AutoCAD (Autodesk) ............................................................ 135
Ansys BladeGen (Ansys) ......................................................... 141
CATIA (Dassault Systèmes) .................................................... 143
Creo Parametric (PTC) .......................................................... 144
Inventor (Autodesk) ............................................................... 160
Ansys SpaceClaim (Ansys) ....................................................... 163
Known issues in Spaceclaim .................................................. 166

CFD ........................................................................................ 169
Ansys Meshting (Ansys) ......................................................... 173
OMNIS/AutoGrid (Cadence) .................................................... 174
Ansys ICEM-CFD (Ansys) ....................................................... 177
Simerics (Simerics) ............................................................... 178
TCFD (CFD Support) ............................................................. 188
Ansys TurboGrid (Ansys) ....................................................... 189

Batch mode/ Optimization ....................................................... 191
Reference components ......................................................... 194
Model finishing ...................................................................... 198

3 SETTINGS ......................................................................... 199

Licensing ................................................................................ 199
Preferences .......................................................................... 199
General .................................................................................. 200
Units ....................................................................................... 204
Basic units ............................................................................. 205
Specific speed ........................................................................ 207
Other units ............................................................................. 209
Impeller/ Stator ..................................................................... 210
Progression diagrams ........................................................... 211
Initial default settings ............................................................ 212
Warning level ........................................................................ 213
# Part VI Views

1 Meridian ......................................................................................................................... 240
2 3D Model .......................................................................................................................... 243
   Model display (top) ........................................................................................................... 245
   Model tree (left) ................................................................................................................ 252
   Problems when generating the 3D model .......................................................................... 256
3 Report ............................................................................................................................... 258

# Part VII Impeller

1 Main dimensions ............................................................................................................... 262
   Centrifugal, Mixed-flow Pump / Fan ................................................................................ 264
   Setup ................................................................................................................................. 265
   Parameters ....................................................................................................................... 267
   Dimensions ...................................................................................................................... 275
   Axial Pump / Fan ............................................................................................................... 282
   Setup ................................................................................................................................. 284
   Parameters Pump ............................................................................................................ 286
   Inducer ............................................................................................................................... 291
   Parameters Fan ............................................................................................................... 293
   Dimensions ...................................................................................................................... 299
   Centrifugal Compressor ................................................................................................. 307
   Setup ................................................................................................................................. 309
   Parameters ....................................................................................................................... 311
   Dimensions ...................................................................................................................... 318
   Radial-inflow Gas Turbine ............................................................................................... 326
   Setup ................................................................................................................................. 327
   Parameters ....................................................................................................................... 329
   Dimensions ...................................................................................................................... 335
   Axial Gas Turbine / Compressor ..................................................................................... 343
   Setup ................................................................................................................................. 345
   Parameters Gas Turbine ................................................................................................... 347
   Parameters Compressor ................................................................................................... 354
   Dimensions ...................................................................................................................... 358
   Francis Turbine ............................................................................................................... 365
   Setup ................................................................................................................................. 366
   Parameters ....................................................................................................................... 368
   Dimensions ...................................................................................................................... 372
   Kaplan Turbine ............................................................................................................... 377
   Setup ................................................................................................................................. 379
   Parameters ....................................................................................................................... 381
2 Meridional contour ................................................................. 398

3 Mean line design ................................................................. 429

Blade angles ........................................................................ 451

Outlet triangle ...................................................................... 455

Blades mean lines ................................................................ 467

Design mode ......................................................................... 474

Additional views ................................................................... 483

Informational values ............................................................... 486

Blade setup ........................................................................... 434

Blade angles ........................................................................ 451

Inlet triangle ......................................................................... 455

Spans .................................................................................. 450

Blade setup ........................................................................... 434

Radial element blade ............................................................. 449

Compound blade shapes ......................................................... 450

Blade setup ........................................................................... 434

Bezier .................................................................................. 403

Circular Arc + Straight line .................................................. 409

Contour ................................................................................ 412

Leading, Trailing edge .......................................................... 413

Meridional flow calculation .................................................. 415

Hub/Shroud materials ............................................................ 419

Bezier curve .......................................................................... 423

Line segment curve ............................................................... 425

Secondary flow path .............................................................. 425

Additional views ................................................................... 427

Material density .................................................................... 390

Multi stage ........................................................................... 391

Shaft/Hub ............................................................................. 393

Cordier ................................................................................ 394

Balje .................................................................................... 395

Dimensions .......................................................................... 385

© CFturbo GmbH
Part IX  Volute

1 Setup + Inlet ................................................................. 581
   Setup .............................................................................. 582
   Inlet details ................................................................. 587

2 Cross Section ............................................................. 588
  Bezier cross section ..................................................... 593
   Line Segments cross section ........................................ 594
   Radius based cross section .......................................... 597
   Internal cross sections ................................................ 598

3 Spiral development areas .............................................. 599
   Design rule .................................................................. 602
   Cut-water compensation ............................................. 604
   Circular arc approximation ......................................... 605
Double Volute ................................................................. 606
Additional views ........................................................... 610
4 Diffuser ........................................................................ 612
Additional views ........................................................... 619
5 Cut-water ...................................................................... 620
  Simple .......................................................................... 624
  Fillet ........................................................................... 627
  Sharp .......................................................................... 630
  Additional views ........................................................... 631
6 CFD setup .................................................................... 633
7 Model settings ............................................................. 634

Part X Appendix ............................................................ 636
1 References .................................................................... 637
2 Symbols ........................................................................ 641
3 Contact addresses ........................................................ 643
4 License agreement ........................................................ 643

Index ............................................................................. 652
Part I
CFturbo is made to interactively design centrifugal, mixed-flow and axial turbomachinery: pumps, fans, compressors, turbines. The software is easy to use and does enable quick generation and variation of impeller, stator and volute geometries. Several models can be displayed, compared and modified simultaneously.

It contains numerous approximation functions that may be customized by the user in order to implement user specific knowledge into the CFturbo-based design process. In spite of the creation of semiautomatic proposals, fundamental experiences in turbomachinery design are helpful but not necessary. An experienced turbomachinery design engineer should be able to design new high-quality impellers and volutes more easily and quickly.

CFturbo runs under the 64 bit Windows versions currently supported by Microsoft. Currently these are Windows 8.1, Windows 10 and Windows 11. Hardware requirement is a modern office PC.

The software user interface is in English language, whereas the manual is available in German and English.

Integration of geometry data into the CAE environment is easily possible by direct interfaces to various CAD- and CFD-systems.
Please read the License agreement before using the program.

Information about activating license you can read in chapter Licensing.

Contact persons you can find under Contact addresses, actual information on the CFturbo website.

Copyright © CFturbo GmbH
2 What's new in CFturbo

Information about news in the current and previous versions is available on the CFturbo website:

https://cfturbo.com/software/release-notes
3 General

This chapter contains some general program information about

- Installation
- Licensing
- Batch mode
- Project structure
- Handling
- Windows Explorer integration
- Troubleshooting

3.1 Installation

PLEASE NOTE:

- Administrator rights are required for the installation.
- CFturbo runs on Windows 8.1, 10 and 11.
- Microsoft .Net Framework 4.0 or higher is required for running CFturbo. More information can be found on http://www.microsoft.com/net.

(1) Download

https://cfturbo.com/software/download
Download is possible for authorized users only.

Registered users can change their password.

Registration is required to get download access.

Available downloads after login:

- CFturbo\_setup.exe
  CFturbo client setup (64 Bit)

- CFturboLicenseServer\_setup.exe
  CFturbo License server setup (for customers only)

(2) Installation

Start the installation program CFturbo\_setup.exe.

Uninstall existing version optionally.
Accept the license agreement.

Accept or select the destination directory.

If the directory already exists, the existing installation can be overwritten.
CFturbo Ansys Workbench Extension can be installed optionally.

The extension is installed into all existing Ansys installations from version 19.0. Of course, the extension can be installed later manually.

More information about the Cfturbo Ansys Workbench Extension is available here: https://cfturbo.com/software/interfaces-workflows/extension-for-ansys-workbench

Accept or modify the start menu name.

The installation takes about 20 seconds normally.
(3) Licensing

After successful installation of the software CFturbo the licensing must be performed.

3.1.1 Silent Mode

In some cases, you may need to install CFturbo in silent mode, so the installation is performed automatically without any user interaction.

For example, you install CFturbo in silent mode if you need to automate the installation procedure or deploy the application on multiple computers on your network.

You can install CFturbo in silent mode via specific command-line arguments.
Accept the terms of use

By installing CFturbo in silent mode, you confirm that you consent to all license terms and conditions.

Unattended installation

On the target computer, run the installation using the following command-line arguments:

<path_to_installation_package>\<CFturbo_installer_executable> /SILENT /VERYSILENT /SUPPRESSMSGBOXES

The setup will install CFturbo without user interaction.

Note: If a previously installed version of CFturbo is detected, silent setup will fail asking for uninstalling that version! In this case, the uninstallation of the previous version should be done prior to installing the new version.

Note: Administrator permissions are required to run the installation!

Unattended uninstallation

On the target computer, run the uninstall process using the following command-line arguments:

<path_to_installed_application>\uninstall.exe /SILENT /VERYSILENT /SUPPRESSMSGBOXES

The setup will uninstall CFturbo without user interaction.

As the uninstallation doesn't prompt for the removal of the settings, an additional switch

<path_to_installed_application>\uninstall.exe /SILENT /VERYSILENT /SUPPRESSMSGBOXES /REMOVESettings

can be added to remove the application settings during the uninstall process.

Note: Administrator permissions are required to run the uninstallation!

3.2 Licensing

? SETTINGS | General | Licensing
CFturbo can be used without a valid license in viewer mode. This mode allows to open project files independent of the included components for reading access. No changes can be done in viewer mode.

For modifying projects with CFturbo a valid license is necessary. Does a project include multiple components, only that ones can be modified, a valid license is present for.

A special feature of the CFturbo license model are stators. With every license for volute or impeller it is possible to create and modify vaneless stators.

In principle, 3 alternative types of licensing are available:

- **Node-locked license**
- **Floating license**
- **Cloud license**

Menu item **Licensing** enables license handling.

**License expiration**

If the license of a software module has expired, it can be reactivated by replacing the license with a new one.

A hint with remaining days appears on startup screen 20 days before expiration of the license. The number of days for this hint can be specified in **SETTINGS | Preferences | General**.

**Steps for licensing**
At the first start of CFturbo there is no running license available. For using the viewer mode, no further steps are necessary.

If projects are to be modified, one of the following actions must be performed:

a) A node-locked license has to be requested and installed

b) CFturbo has to be configured for using a floating license in place.

c) A cloud license has to be installed

In general all licensing steps can be performed using remote desktop connection (RDP). But keep in mind that finally a Node-locked license can be used directly on this computer only and not via a RDP session. For this purpose, a Floating license is required.

### 3.2.1 Node-locked license setup

For using CFturbo with a node-locked license 2 steps have to be performed:

1) Requesting a license using the CFturbo license dialog

2) Storing the received license file in the CFturbo installation directory

Note: If CFturbo is configured for using a floating license modules get checked out from that license first if available.

**Requesting a node-locked license**

If not either a node-locked license file is present or a floating license is configured, CFturbo will start the licensing dialog (SETTINGS | General | Licensing).
Here you can select **REQUEST new node-locked license by E-mail**.
Under **Modules** the CFturbo modules must get selected for which a license should be requested. Fill the **Company** field with the requesting company’s name.

The **Start date** of the requested license can be selected for e.g. sync a short time-period license to a project’s start date.

The so-called **Machine ID** and the **Checksum** are calculated automatically and ensure the singular usage of provided license information as well as to link the license to the local computer.

After input of all necessary information you can

- use the **Create E-Mail** button to prepare a message with the computer’s default mail client (the mail will NOT be sent automatically!)

  OR

- use the **Copy to Clipboard** button if you want to create the mail manually and paste the information (send the mail to **sales@cfturbo.com**).

**Install license file**
The license file you receive must be stored in the CFturbo installation directory (C:\Program Files\CFturbo *) you have chosen during the setup. It already has .lic as file extension, this extension must be preserved!

There should be only one license file (*.lic) present in this directory.

Afterwards you can restart CFturbo and check the license information.

### 3.2.2 Floating license setup

**Selecting the license server machine**

Network (floating) licensing requires a CFturbo license server software running on a server machine. The license server controls access of the clients to the CFturbo licenses.

The server machine should have the following properties:

- The operating system of the server machine has to be Microsoft Windows®. It's highly recommended to use a server system (Windows Server 20xx).
- The server machine must be accessible from the CFturbo clients. Normally, server machine and CFturbo clients have to be located in the same local area network (LAN). Usage of the floating licenses across locations (WAN) is allowed with the corresponding license agreement only.
- The server machine should be highly available, have high-speed Ethernet connection and a moderate level of network traffic.
- All license related files must be located on a local computer disk of the server machine.
- The server machine must have a static IP address.
- Make sure that the time and date of the server machine is correct. Do not manipulate these settings manually.

**License server on Virtual Machines**

The CFturbo license server software can be installed and used on a Virtual Machine (e.g. VMware). However, the license handling is not tested and certified on all Virtual Machine environments. Problems related to the use of virtual servers cannot be resolved by the CFturbo support and should be reported to the Virtual Machine supplier.

Note, that using Virtual Machines to duplicate the available CFturbo licenses is explicitly prohibited.

**Steps for network licensing**
For using CFturbo with a floating license the following steps have to be performed:

1. Setting up the CFturbo license server
2. Requesting a license using the Request Generator
3. Storing the received license file in the CFturbo license server installation directory
4. Configuring the clients for accessing the floating license

3.2.2.1 License server setup

Installing the license server

The CFturbo license server is installed by a setup separate from the CFturbo program. It includes the following components:

- Server files
- Windows Service "Reprise LM for CFturbo"
- Request Generator
- This manual

The license server will be installed as a Windows Service which is automatically started on system boot.

After starting the setup, selecting installation directory and start menu settings, the server parameters can be configured:
Allow autodiscovery of server enable the clients to find the license server automatically within the same network broadcast subnet.

The RepriseLM server has a built in web server. When Start web server is selected, the installed Windows service will also run a web server on the port configured here.

Note, that the setup is not checking for port conflicts, the port must be available. It can be changed e.g. by uninstalling and installing the server again.
The last wizard page offers to **Create a license request**. This option will start the Request Generator (see below).

The Windows Start menu contains the following items after installation:

- **CFturbo Website**
  - open CFturbo website in a webbrowser
- **Create License Request**
  - start the Request Generator, see below
- **Display Server Logfile**
  - display the server log file
- **Restart License Server**
  - restart the CFturbo license server
- **RLM End User Manual**
  - open Reprise License Manager (RLM) License Administration Manual in a webbrowser
- **Run License Server Diagnostics**
  - display detailed license server diagnostic information; required for support requests
- **Start License Server**
  - start the CFturbo license server
- **Stop License Server**
  - stop the CFturbo license server
- **Uninstall CFturbo License Server**
  - uninstall the CFturbo license server
Please note: License server handling (Start, Stop, Restart) requires administrator rights. "Run as administrator" in the context menu of the corresponding Start menu item has to be used.

**Requesting a floating license**

The Request Generator collects all information needed for the license request.

Under **Modules** the CFturbo modules must get selected for which a license should be requested. Fill the **Company** field with the requesting company’s name.

The **Start date** of the requested license can be selected for e.g. sync a short time-period license to a project's start date.

The so-called **Machine ID** and the **Checksum** are calculated automatically and ensure the singular usage of provided license information as well as to link the license to the network server.

The **Concurrent users** setting enables you to change to number of users you request the license for.

After input of all necessary information you can

- use the **Create E-Mail** button to prepare a message with the computer’s default mail client (the mail will NOT be sent automatically!)
OR

- use the **Copy to Clipboard** button if you want to create the mail manually and paste the information (send the mail to [sales@cfturbo.com](mailto:sales@cfturbo.com)).

**Install license file**

The license file you receive must be stored in the license server installation directory (C:\Program Files\CFTurbo \LicenseServer) you have chosen during the setup. It already has .lic as file extension, this **extension must be preserved!**

There should be only one license file (*.lic) present in this directory.

After placing the file in the folder, restart the Windows service ("Reprise LM for CFTurbo") by operating system control. Alternatively you can use "Restart License Server" under CFTurbo in the Windows Start menu.

**Please note:** Restarting the license server requires administrator rights. "Run as administrator" in the context menu of the corresponding Start menu item has to be used.

Now the logfile and the web server page can be checked for the licenses to be running.

**Firewall configuration**

If the server is protected by the Windows Firewall, exceptions for the license server application are created during the setup. Depending on your server configuration and network infrastructure, additional configuration steps may be necessary.

**Port configuration**

If you want to serve licenses across a firewall, at least two port numbers have to be allowed your firewall to pass requests on these ports:

- The RLM server itself defaults to port **5053**.

  This port must be modified when running different software on the same Reprise license server. The port number can be specified at the end of the HOST line of the license file, see section "HOST line" on page 21 of the "RLM License Administration" manual.

  e.g. ```
  HOST licserver CFIDNET2=81f344a7a69115ce4cfd7c46efd1f 5555
  ```

  Please note that the CFTurbo client is not able to automatically detect the license server when the port number is different to the default 5053. In this case the Windows environment variable on the client has to be specified explicitly, including the modified port number (see Client setup).

  e.g. ```
  CFTURBO_LICENSE = 5555@licserver
  ```
The ISV server starts with a dynamic port number which is not known before start-up time.

It is possible to have RLM assign a fixed port number to the ISV server. In order to do this, you need to specify the port number for the ISV server on the ISV line of the license file. The port number is the fourth parameter in the ISV line:

```
ISV <isvname> <isv-binary-pathname> port=<port-number>
```

E.g. ISV cfturbo cfturbolm.exe port=5054

Except the web server port, all ports have to be reachable.

For details about the license file settings see RLM Support for License Administrators and Users and RLM License Administrator and User FAQs.

**Additional configuration options**

For additional configuration options check RLM Support for License Administrators and Users.

### 3.2.2.2 Client setup

Every client computer that should run CFturbo has to be configured for using the floating license.

**Auto configuration**

CFturbo is able to automatically detect running license servers in the network. No further configuration is needed on client side, if the detection succeeds. If the client is not able to find the license server, it has to be configured using the environment variable.

The detection relies on the client being in the same network broadcast subnet like the license server and a default configuration of the license server. For further details see RLM Support for License Administrators and Users.

**Manual configuration by environment variable**

The Windows environment variable CFTURBO_LICENSE is used to identify the location of the license server.

It is set to `<port>@<host>`

- `<port>`: port of the license server for connection between client and server
- `<host>`: host name of the license server machine (name or IP address)

The default port - if not configured in the server license file (on the SERVER or HOST line) - is 5053.
Example:

```
CFTURBO_LICENSE=5053@rlmhost
```

Multiple license servers are separated by semicolon:

```
CFTURBO_LICENSE=5053@rlmhost;5053@rlmhost2
```

For details about how to set environment variables, please consult your IT department or the Windows documentation.

### 3.2.3 Cloud license setup

The behavior is very similar to the "Floating license", with the difference that the license server is located in the cloud. Therefore, there is no need to run your own license server. A stable internet connection is required to use cloud licensing.

For using CFturbo with a cloud license 2 steps have to be performed:

1) Requesting a cloud license from CFturbo sales team
2) Storing the received license file in the CFturbo installation directory

**Note:** If CFturbo is configured for using a floating license modules get checked out from that license first if available.

### Install license file

The license file you receive must be stored in the CFturbo installation directory (C:\Program Files\CFturbo *) you have chosen during the setup. It already has .lic as file extension, this extension must be preserved.

There should be only one license file (*.lic) present in this directory.

Afterwards you can restart CFturbo and check the license information.

### 3.2.4 Show license information

Current license information are displayed here.

The **company** name is for information only.

**Path** is the license file location and the content of the environment variables used for defining floating license servers.
If available the last Error message of license checking is displayed.

### 3.2.5 Troubleshooting

#### Error messages

<table>
<thead>
<tr>
<th>Problem</th>
<th>Message</th>
<th>Reasons</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Diagnostic configuration

CFturbo and its license server are enabled to output diagnostic information about licensing. Start menu entries are created to run a script collecting useful information for the support:

- "Run CFturbo Diagnostics" on the client
- "Run License Server Diagnostics" on the server

Alternatively the client license dialog and the server request generator have a Diagnostics button in the bottom line to start the diagnostic.

The resulting text file will give among others the following information:

- time the program was run
- working directory
- relevant environment variables
- the license files in use, in the order RLM will use them (can be re-ordered from your normal list if RLM_PATH_RANDOMIZE is set)
- a list of all licenses which can be checked out

License server problems

If problems occur setting up or running the license server, the following can be checked:

- Service "Reprise LM for CFturbo" present and running (Windows® services)
- Server logfile (installation directory of license server, server.log and cfturbo.dlog)
- Server diagnostics (License server web interface -> Diagnostics)
3.3 Batch mode

CFturbo can be executed in **batch mode** to modify designs without any screen display and user interaction. This is essential for using CFturbo with optimization software.

**Syntax**

```
cfturbo.exe -batch <batch file> [-verbose] [-export <interface name>] [-log <log file>]
```

**Example**

<table>
<thead>
<tr>
<th>CFturbo is installed in:</th>
<th>c:\Program Files\CFturbo 20xx.x\</th>
</tr>
</thead>
<tbody>
<tr>
<td>Batch file is:</td>
<td>c:\tmp\Example.cft-batch</td>
</tr>
</tbody>
</table>

```
"c:\Program Files\CFturbo 20xx.x\cfturbo.exe" -batch c:\tmp\Example.cft-batch
```

**Options**

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>-batch &lt;batch file&gt;</td>
<td>Enables CFturbo batch mode. &lt;batch file&gt; can either be a CFturbo batch file (<em>.cft-batch) or a CFturbo project file (</em>.cft).</td>
</tr>
<tr>
<td>-verbose</td>
<td>Display log output on the command line.</td>
</tr>
<tr>
<td>-export &lt;interface name&gt;</td>
<td>If CFturbo is started with a CFturbo project file in batch mode, an export interface can be selected like in the batch file.</td>
</tr>
<tr>
<td>-log &lt;log file&gt;</td>
<td>Use specified log file for output</td>
</tr>
</tbody>
</table>

All other batch commands have to be defined in a "Batch file".

**Batch file**

The **batch mode** of CFturbo is controlled by an XML file *.cft-batch. For a specific CFturbo project this file can be created via [PROJECT | Batch mode/ Optimization](https://example.com).
Due to a close relation between the CFturbo file format (*.cft) and the batch mode format (*.cft-batch), only batch mode files created with the same version as your CFturbo file should be used. After an update of CFturbo a new batch mode file can be created and the needed adjustments can be done.

The resulting batch mode file contains all selected parameters of the CFturbo project as XML nodes supplemented by a short description and optional range definitions.

File structure:

```xml
<?xml version="1.0" standalone="yes"?>
<CFturboFile Version="20xx.x">
    <CFturboBatchProject InputFile="<InputFileName>">
        <Updates>
            [...]  
        </Updates>
        <BatchAction ...>
            [...]  
        </BatchAction>
        <BatchAction ...>
            [...]  
        </BatchAction>
    </CFturboBatchProject>
</CFturboFile>
```

A batch mode file can contain multiple elements of the CFturboBatchProject-type, each of which is handling a specific CFturbo project. This allows the combination of multiple batch mode files into one batch mode file.

All XML-subelements are optional and can occur multiple times except for the Updates-block which must occur once per CFturboBatchProject-element.

The InputFile-attribute of the CFturboBatchProject-element specifies the path of the CFturbo project file.

**Updates**
The **Updates** block contains all selected parameters that should be modified during batch run. These can be simple scalar values, points or arrays. Each parameter has its own description for more easy navigation in the file, e.g. for optimization setup.

**Some general remarks to the parameter update:**

- Parameters are available for batch mode only if they can be modified in interactive design mode.
  
  Examples:
  - **Impeller main dimensions**, **blade angles** are available only if automatic calculation is disabled.
  - Values for splitter blades are available when splitters are not geometrically linked to main blades.

- Parameters can be modified within the same constraints that exist in interactive design mode. Modifications that violate the constraints will be corrected automatically.

- All parameters of the **Updates** block with their new values **after** the batch run are saved in a resulting parameter file `<name of batch file>.cft-res`. The file structure is identical to the batch file `*.cft-batch` and allows a comparison between the desired and the realized parameter value which takes all restrictions into account.

**Batch actions**

Two different actions are available for further processing of the CFturbo projects loaded in batch mode. The **BatchAction**-element can occur multiple times, e.g. for exporting multiple parts of the geometry in different modelstates or saving an updated geometry.

- **Export action**

  The Export-action is used to export the project data utilizing the export interfaces CFturbo supports. Export is performed according to the export settings saved in the CFturbo file. The **ExportComponents** subelement can be specified for export interfaces supporting components selection. For details about the supported selection options for the specific interface see **Project | Export**.

  ```xml
  <BatchAction Type="Object" Name="Export">
    <WorkingDir>c:\Examples\Myexports\</WorkingDir>
    <BaseFileName>Pump1</BaseFileName>
    <ExportInterface Type="Enum">General</ExportInterface>
    <ExportComponents Count="3" Type="Array1" Desc="Components to be exported">
      <Value Type="Integer" Caption="Nozzle" Index="0">2</Value>
    </ExportComponents>
  </BatchAction>
  ```
<Value Type="Integer" Caption="Impeller" Index="1">3</Value>

<Value Type="Integer" Caption="Stator" Index="2">4</Value>

</ExportComponents>

</BatchAction>

<table>
<thead>
<tr>
<th>Attribute / Node</th>
<th>Value</th>
<th>optional</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>Export</td>
<td>no</td>
<td>Action name</td>
</tr>
<tr>
<td>WorkingDir</td>
<td>&lt;existing path&gt;</td>
<td>yes</td>
<td>Folder for exported files</td>
</tr>
<tr>
<td>BaseFileName</td>
<td>&lt;filename&gt;</td>
<td>yes</td>
<td>File name without extension</td>
</tr>
<tr>
<td>ExportInterface</td>
<td>e.g. &quot;General&quot;</td>
<td>no</td>
<td>Export interface to use. The following values are valid:</td>
</tr>
<tr>
<td>ExportComponents</td>
<td>-</td>
<td>yes</td>
<td>Components to be exported. Take into account that only components supported by the export interface will be exported</td>
</tr>
</tbody>
</table>

• Save action
Is used for saving the CFturbo project after applying batch updates. It can also be used for the automatic conversion of CFturbo files created with older program versions.

```xml
<BatchAction Type="Object" Name="Save" Desc="CFT file name of modified project">
  <OutputFile>C:\Testing\Examples\Impeller\Pump1_new.cft</OutputFile>
</BatchAction>
```

<table>
<thead>
<tr>
<th>Attribute / Node</th>
<th>Value</th>
<th>optional</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>Save</td>
<td>no</td>
<td>Action name</td>
</tr>
<tr>
<td>Desc</td>
<td>&lt;string&gt;</td>
<td>yes</td>
<td>Description of the modified file name</td>
</tr>
<tr>
<td>OutputFile</td>
<td>&lt;existing path&gt;</td>
<td>no</td>
<td>Specifies the path of the file save destination</td>
</tr>
</tbody>
</table>

### 3.3.1 Example

The example of a CFturbo batch file `pump.cft-batch` below, changes the number of impeller blades of the `pump.cft` example project.

Subsequently the modified impeller (component #2) gets exported for TurboGrid and the project is saved into the CFturbo project file `pump_modified.cft`. 
During runtime a log-file `pump.log` is created in the directory of `pump.cft-batch`:
3.3.2 Exit Codes

CFturbo provides the following exit codes, which report the result of the batch run:

<table>
<thead>
<tr>
<th>Exit Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>No errors or warnings occurred during batch run.</td>
</tr>
<tr>
<td>1</td>
<td>Last batch run was completed with warnings but no errors.</td>
</tr>
<tr>
<td>2</td>
<td>Last batch run was completed with errors.</td>
</tr>
</tbody>
</table>

3.4 Project structure

A CFturbo project describes a complete single-stage or multi-stage machine.

Project types

The following project/machine types are available:

- Pump
- Fan
- Compressor
- Gas Turbine
- Hydro Turbine

Project structure

A project consists of the global parts

- Project information
- Global setup
- Performance prediction
- Exp
- Batch mode
- Reference components

and the single component parts of the assembly. The following components are available:
- Up to 5 Impellers on any position (exception: only 1 runner in Francis turbine project)
- 1 Volute as first/last component
- any number of Stators (vaned or vaneless)

Components can be added directly in the components view.

**Coupling between components**

The following coupling types are available:

<table>
<thead>
<tr>
<th>Coupling Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>⬆️ ⬇️ Coupling in flow direction (Default)</td>
<td>Inlet cross section of a component is defined by the outlet cross section of previous component.</td>
</tr>
<tr>
<td>⬆️ ⬇️ Coupling reverse flow direction</td>
<td></td>
</tr>
</tbody>
</table>
Outlet cross section of a component is defined by the inlet cross section of next component.

**Uncoupled**

Both sides (inlet and outlet of the respective component) are independent. Gaps between neighboring components are possible. By default a newly added stator is uncoupled at its outlet. That makes it easy to add a stator between two adjacent but detached components.

The component coupling can be adjusted in the component view directly at inlet or outlet the neighboring components.

The impeller as the core component of a machine has primary sides both at inlet and outlet.

### 3.4.1 Add component

Components can be added to the current project by alternative possibilities:

<table>
<thead>
<tr>
<th>For new projects without components</th>
<th>For projects with existing components</th>
</tr>
</thead>
<tbody>
<tr>
<td>The menu is displayed automatically for empty projects</td>
<td>Use + button of the currently selected component</td>
</tr>
<tr>
<td><img src="image" alt="Menu screenshot" /></td>
<td>in the Meridian view</td>
</tr>
<tr>
<td></td>
<td>in the component list on left side</td>
</tr>
</tbody>
</table>

In the menu the component type can be selected. Depending on the specific insert position some of the types can be disabled:

- up to 5 impellers in a project
- a single volute only
• impeller can be added only if the flow direction on the selected position is suitable to the impeller geometry.

Please note:
• If you add a component on the first position of the project (in flow direction) then the inlet conditions defined in the Global setup are applied for this new component.
• After importing existing components, initially the component is disabled in order to preserve the original geometry. After activation the component will be updated and could be modified.

Import CFturbo component

CFlutter components can be extracted from existing CFturbo projects (*.cft) and inserted in the current project. The available components of the imported project are listed with their name and type and can be selected for import.

Import external geometry

New components can be created using external geometry description. Reverse engineering process can be started from here.

Details can be found under Import external geometry.
3.4.1.1 Import external geometry

This feature allows direct import of external geometry description.

Currently, 3 formats are supported:

- **CFturbo Exchange**: `.cft-geo`
- **Ansys BladeGen**: `.rtzt`
- **Reverse Engineering**: STEP, IGES, Parasolid, BREP
This format (*.cft-geo) can be used to import rotational symmetric components only. It is a XML format including the following information:

**General information** *(mandatory)*

- **Length unit type**
  - LengthMm for millimeters
  - LengthM for meters
  - LengthIn for inches

- **Unshrouded flag**
  - Only required for vaned designs. 1 = unshrouded, 0 = shrouded

- **xTipInlet**
  - Only required for vaned and unshrouded designs. Tip length at inlet.

- **xTipOutlet**
  - Only required for vaned and unshrouded designs. Tip length at outlet.

**Meridional contour information** *(mandatory)*

- **Hub contour**
  - Array of curves. At least one curve is required. Each curve contains an array of 2D-points (r, z coordinates).
  - Stretches on rotation axis can be specified as part of the hub contour. It is required, for example, for designs without hub (stators with pipe form).

- **Shroud contour**
  - Array of curves. At least one curve is required. Each curve contains an array of 2D-points (r, z coordinates).

**Blades information** *(only required for vaned designs)*

- **Mean line data**
  - Array of at least 2 curves. Each curve contains an array of 3D-points (r, T, z coordinates).

- **Thickness data**
  - Array of 2 curves. The first curve defines thickness data on hub, the second one on shroud. Each curve contains an array of points defined by two coordinates:
    - x = relative point position on mean line
    - y = blade thickness at this position
Thickness data are required at least for both relative positions, 0 (leading edge) and 1 (trailing edge). The thickness distribution along the mean line is interpolated using all values.

An example file can be easily generated by exporting any CFturbo component using the "CFturbo Exchange" export interface.

3.4.1.1.2 RTZT format

This format can be used if the following geometrical data are available on span sections: radius, axial coordinate, circumferential blade angle and blade thickness. Values must be separated using spaces.

The file must include the following information:

<table>
<thead>
<tr>
<th><strong>Possible values</strong></th>
<th><strong>Description</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of blades</td>
<td>Positive Integer</td>
</tr>
<tr>
<td>Splitters flag</td>
<td>0 or 1</td>
</tr>
<tr>
<td>Pitch fraction</td>
<td>Positive float number lower than 1</td>
</tr>
<tr>
<td>Number of layers</td>
<td>Integer greater than 1</td>
</tr>
<tr>
<td>Thickness flag</td>
<td>N or T</td>
</tr>
<tr>
<td>Span fraction</td>
<td>Float number between 0 and 1</td>
</tr>
<tr>
<td>Number of points</td>
<td>Integer greater than 1</td>
</tr>
<tr>
<td>r</td>
<td>Nonnegative float number</td>
</tr>
<tr>
<td>theta</td>
<td>Float number</td>
</tr>
<tr>
<td>z</td>
<td>Float number</td>
</tr>
<tr>
<td>thickness</td>
<td>Nonnegative float number</td>
</tr>
</tbody>
</table>

Number of main blades

0 if no splitter blades

Position of splitter blades (ignored for main blade)

Number of layers

N = Normal thickness values

T = Tangential thickness values

Relative position of a span section

Number of points describing geometrical data for a span section

Radial coordinate value for a point

Circumferential blade angle value for a point in radians

Axial coordinate value for a point

Blade thickness value for a point.
Parts of meridional contour without blades are represented using a zero thickness value

Information must be written using the following structure:

**Global information**
Number of blades
Splitters flag

For each blade (main and splitter if exists)
Pitch fraction
Number of layers Thickness flag

For each layer
Span fraction Number of points

For each point
r theta z thickness

An example file can be easily generated by exporting any example using the BladeGen export interface.

3.4.1.3 Reverse Engineering

Reverse Engineering (RE) is the process transforming 3D geometry from neutral file formats (STEP, IGES, Parasolid, BREP) into a parametric CFturbo model.

The process is semi-automatic and consists of the following basic stages:
1. [Creating/opening a CFturbo-project](#) considering the desired machine type
2. Specifying the design point ([Global setup](#))
3. Defining 3D geometry to be redesigned
4. Specifying parameters for automatic computation of the CFturbo model
5. Manual adjusting the CFturbo model to improve alignment with the initial 3D geometry
The stages 1. and 2. serve the purpose of specifying the turbomachinery’s operating point as usual. The stages 3. and 4. are performed inside a separate dialog enabling the user to create a CFturbo model quickly and efficiently. In stage 5. the user can modify the redesigned CFturbo-model as needed within the ordinary design process (impellers, stators).

**RE-dialog**

The RE-dialog is divided into the following parts:

1. **Steps** (top)
   - selecting the current active RE-step (loading 3D geometry, meridional redesign, blade redesign)
   - contains information related to the current step

2. **Imported geometry** (left)
   - loading external 3D geometry
   - displaying the imported 3D geometry
   - aligning geometry to 3D coordinate system of CFturbo model
   - specifying CFturbo semantics for related geometry-elements (e.g. hub, shroud, blade)

3. **Redesigned geometry** (middle)
   - presenting computation results of the CFturbo model as 3D geometry and meridional view
   - specifying parameters for computation of the CFturbo model via 2D contour’s context menu inside meridional view

4. **Settings** (right)
   - specifying parameters for computation of the CFturbo model

5. **Message tree** (bottom right)
   - contains information, warnings and errors occurred during the RE-process
Detailed description of the import process

1. Preparing the CFturbo-project
   - Creating/opening a CFturbo-project considering the desired machine type
   - Within Global setup: Specifying the design point suitable for the model to be redesigned

2. Load external 3D geometry
   - Preconditions:
     - RE-step Import must be selected at the top
     - Input geometry must be represented by parametric surface data (e.g., B-Spline surface, Cylindrical surface, etc.)
     - Input geometry can be defined within a single or multiple files representing neutral file formats STEP, IGES, Parasolid or BREP
   - Load geometry by clicking Load CAD data button or via drag & drop from a file explorer into the 3D-view.
   - Loaded geometry is displayed in the 3D view and contained geometry elements are listed in the model tree left.
3. 3D alignment of loaded 3D geometry

- In order to obtain an accurate CFturbo model the loaded geometry has to be aligned sufficiently to the 3D coordinate system of the CFturbo-model.

- This essentially includes aligning rotation/symmetry axis of loaded geometry with the z-axis of the global coordinate system.

- Achieving a valid alignment can directly be done inside the **Imported geometry** section using the **Transformation** panel.

4. Defining CFturbo-semantics on geometry elements

- In order to compute a CFturbo model from external 3D geometry a classification of imported 3D faces representing a specific CFturbo model part (e.g., hub, shroud, blade) must be applied (mapping).

- Mappings are applied context sensitive related to the currently selected RE-step (**Meridian**: Hub, Shroud/Tip, **Blade**: Main blade, Splitter blade) at the top.

- This can be achieved by selecting a single or multiple 3D face(s) in the 3D view or model tree and using one of the following options:
  - opening the context menu by clicking right mouse button and selecting an appropriate CFturbo model part (see first image below)
  - clicking buttons representing CFturbo model parts next to the model tree (see first image below)
- Mappings of 3D face(s) to CFturbo-model parts are represented by an information in the model tree of the **Imported geometry** section and by explicit nodes in the model tree of the **Redesigned geometry** section.

- Mappings can be deleted using one of the following options:
  - opening the context menu of the related tree node in the **Redesigned geometry** section and choosing the option **Remove mapping**
  - clicking button **Remove mapping** next to the model tree
5. Computation of CFturbo model

- Updating redesigned model:
  - within RE-step Meridian: computation of the CFturbo model is performed automatically related to user inputs
  - within RE-step Blade: computation of the CFturbo model is started manually by pressing Update design in the Settings section or is performed automatically related to user inputs in case of activated Automatic design update
  - Automatic computations are triggered by adding/removing assignments of 3D-faces to CFturbo model parts and by changing parameters in the Settings section and in the context menu of 2D meridional contours

- Parameters:

<table>
<thead>
<tr>
<th>RE-step</th>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Import</td>
<td>Rotation direction [Impeller only]</td>
<td>Defines impeller's rotation direction</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Meridian</td>
<td>Unshrouded</td>
<td>Closed (shrouded) or open (unshrouded) geometry at blade tip side</td>
</tr>
<tr>
<td><strong>Tip clearance</strong></td>
<td><strong>Gap between blade tip and casing</strong></td>
<td></td>
</tr>
<tr>
<td>-------------------</td>
<td>-----------------------------------</td>
<td></td>
</tr>
<tr>
<td>Define inlet/outlet extension [2D contour's context menu]</td>
<td>Hub/Shroud can be extended to user-defined z,r coordinates at the inlet/outlet. This can be applied in case the user-defined 3D-geometry of Hub/Shroud result in too short contours in order to obtain a valid Meridian.</td>
<td></td>
</tr>
</tbody>
</table>

| **Remove inlet/outlet extension [2D contour's context menu]** | Removes user-defined Hub/Shroud extension at the inlet/outlet. |
| **With blades [Stator only]** | **Definition of a vaned or vaneless stator** |

<table>
<thead>
<tr>
<th><strong>Blade</strong></th>
<th><strong>Number of blades</strong></th>
<th>Defines the number of blades of the CFturbo model.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td><strong>Splitter blades</strong></td>
<td>Defines whether geometry has splitter blades.</td>
</tr>
<tr>
<td></td>
<td><strong>Number of spans</strong></td>
<td>Defines number of intersections of the user-defined blade geometry used for blade profile computation.</td>
</tr>
</tbody>
</table>
Offset Hub/Shroud/Tip

Blade intersections are evenly distributed between Hub and Shroud. The area of intersections can be limited by offsets on Hub- and on Shroud-/Tip-side in order to avoid inaccurate blade profiles due to blade root fillets.

Suggestion: Remove blade root fillets from initial CAD-data, if possible.

Blade edge shape (Leading/Trailing edge)

Defines the shape of the Leading/Trailing edge of the user-defined blade geometry (blade profile computation depends on shape of blade’s Leading/Trailing edge).

---

Results of computations are displayed inside the Redesigned geometry section:
- 2D contours in z,r coordinate system inside 2D meridional view:
  - Hub / Shroud / Tip contours
  - Blade’sLeading / Trailing edge
  - Spans (light grey: contours for blade intersection; black: valid blade intersection; orange: invalid blade intersection)
- 3D design model in the 3D-view (incl. model tree)
6. Finalization of import process

- After computation of a valid CFturbo model the Reverse engineering dialog can be closed by pressing OK button (irreversible)
- Now the CFturbo project contains the new component which has to be activated finally

Example

In this section the RE-process from initial input geometry to a CFturbo design is documented for a mixed-flow pump exemplarily. Any redesigned CFturbo model can further be modified as needed inside the ordinary design process.

1. Preparing the CFturbo-project

- Creating a new CFturbo-project

- Specifying the design point within Global setup
• Adding a new component and specifying the import-format
2. Loading and preparing external 3D geometry

- load external 3D geometry from neutral 3D file formats
- align loaded geometry with the coordinate system of the CFturbo model
  - collinear alignment of rotation axis with the z-axis
  - fluid flow direction in positive z-axis direction (for turbines: opposite direction!)
- define desired rotation direction of the loaded geometry
3. Meridional redesign

- **Selection of Step** Meridian

- **Map** 3D-faces to Hub / Shroud

- It’s not necessary to select all 3D-faces representing the complete hub/shroud-surface. Instead a single face covering the surface from inlet to outlet is sufficient.

4. Blade redesign
• Selection of Step **Blade**

• **Map** 3D-faces of a single blade to **Blade**

• select the **Blade edge shape** of the loaded blade geometry (Round, Angular)

• choose sufficient **Blade intersection** parameters
  o default **Number of spans** is sufficient in general
  o offsets can be increased in case of large blade root fillets (optional)

• press **Update design** in order to compute the redesigned geometry

5. Finalization

• close RE-dialog by pressing **OK** after a valid design model is computed
  (initially opened/created project now contains the redesigned CFturbo-model)

• **activate** the redesigned component (in order to modify the design)

• save the created design by saving the project
Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hub / Shroud: Rotation axis of selected faces not collinear with global z-axis.</td>
<td>3D-alignment of specified Hub/ Shroud faces is inaccurate. Symmetry axis is not collinear with global z-axis.</td>
</tr>
<tr>
<td></td>
<td>Update 3D-transformation of imported geometry by aligning symmetry axis to global z-axis.</td>
</tr>
<tr>
<td>Calculation of following blade profile(s) failed: [span number(s)].</td>
<td>Cutting selected blade faces by span surfaces results in open blade profile(s). Redesigned blade shape may differ significantly from given blade faces.</td>
</tr>
<tr>
<td></td>
<td>Ensure there are no gaps between specified blade faces in the range of span surfaces.</td>
</tr>
<tr>
<td></td>
<td>Improve redesigned blade shape manually in the ordinary design process after finishing the RE-process.</td>
</tr>
<tr>
<td>M-distribution of following meanline(s) is not strictly increasing: [span number(s)].</td>
<td>Unsupported meanline shape (no strictly increasing m-distribution in m,t-coordinate)</td>
</tr>
<tr>
<td></td>
<td>Improve redesigned blade shape manually in the ordinary design process after finishing the RE-process.</td>
</tr>
<tr>
<td>Problem</td>
<td>Possible solutions</td>
</tr>
<tr>
<td>------------------------------------------------------------------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>system). Redesigned blade shape may differ significantly from given blade faces.</td>
<td></td>
</tr>
<tr>
<td><strong>No valid Hub / Shroud / Blade tip contour could be computed.</strong></td>
<td>A 2D meridional contour couldn't be computed from given 3D-faces.</td>
</tr>
<tr>
<td></td>
<td>Ensure all specified faces represent the Hub and Shroud/ Blade tip of the model.</td>
</tr>
<tr>
<td></td>
<td>Ensure correct 3D-alignment of specified faces with the global coordinate system of the CFturbo model.</td>
</tr>
<tr>
<td><strong>Shroud / Blade tip has self-intersections.</strong></td>
<td>2D meridional contour intersects itself.</td>
</tr>
<tr>
<td></td>
<td>Reduce tip clearance.</td>
</tr>
<tr>
<td></td>
<td>Apply or adjust extensions of hub/shroud contours.</td>
</tr>
<tr>
<td><strong>Zero length meridional contour(s).</strong></td>
<td>2D meridional contour of Inlet / Outlet has zero length.</td>
</tr>
<tr>
<td></td>
<td>Ensure specified faces for Hub and for Shroud/ Blade tip are not connected to each other.</td>
</tr>
<tr>
<td><strong>Meridional contours intersect each other.</strong></td>
<td>2D meridional contours of Hub and Shroud intersect each other.</td>
</tr>
<tr>
<td></td>
<td>Ensure all specified faces represent the Hub and Shroud/ Blade tip of the model.</td>
</tr>
<tr>
<td></td>
<td>Apply or adjust extensions of hub/shroud contours.</td>
</tr>
<tr>
<td><strong>Computing spans for blade surface intersection failed.</strong></td>
<td>Span surfaces used for blade surface intersection could not be computed from given hub and shroud/tip contours.</td>
</tr>
<tr>
<td></td>
<td>Ensure no undulations are present in hub and in shroud/tip contours due to insufficient input geometry or insufficient 3D-alignment.</td>
</tr>
<tr>
<td><strong>Calculation of blade profiles failed.</strong></td>
<td></td>
</tr>
</tbody>
</table>
### Problem

A redesigned blade could not be computed due to too less successfully computed blade profiles (at least two valid blade profiles must be available).

### Possible solutions

- Ensure there are no gaps between specified blade faces in the range of span surfaces.
- Ensure all specified faces represent the Main/Splitter blade of the model.
- Ensure blade does not exceed meridional boundaries.

**Unconnected blade faces selected.**

There are blade faces specified not being connected to others.

Ensure specified blade faces are connected to each other.

### 3.4.2 Geometric coupling

The sketch illustrates the general layout of the geometric coupling of two neighboring components in case they are coupled:

**Primary / Secondary**
In a group of two neighboring components that are coupled, one is primary always, the other one is secondary. The secondary side is aligned to the primary side. Only the secondary side can define an offset to the neighboring component.

If the geometry of the primary component is changing, the component at the secondary side is adjusted automatically. If a component is deactivated (see Active/ Rename/ Delete), no adjustment will be effected - therefore an overlapping of neighboring components is possible, which is illustrated by a warning (see Components). The same applies if neighboring components are not coupled.

**Coupling definition**

The coupling definition at volute inlet as well as at stator inlet and outlet is made in a uniform manner.

<table>
<thead>
<tr>
<th>Extent</th>
<th>Inlet</th>
<th>Outlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>Neighboring component outlet</td>
<td>Coupling</td>
<td>In flow direction (Fixed by Upstream Outlet)</td>
</tr>
<tr>
<td>Hub</td>
<td>$z = 40,\text{mm}$</td>
<td>$r = 137.3,\text{mm}$</td>
</tr>
<tr>
<td>Shroud</td>
<td>$z = 25.4,\text{mm}$</td>
<td>$r = 137.3,\text{mm}$</td>
</tr>
</tbody>
</table>

**Coupling**
- Information about coupling direction
- Neighboring component's inlet/ outlet
- Inlet/ outlet position at hub and shroud side
- Coordinate transfer from neighboring component (if that is primary)

**Inlet/ outlet**
- Geometry definition optionally by
  - Points on Hub & Shroud
  - Point on Center line, width and angle
- Alternatively absolute coordinates or an Offset can be used, which are automatically converted into each other.

**Rotor-Stator-Interface**

Rotor-Stator-Interface (RSI) at impeller outlet can be defined in the CFD-Setup of the impeller, otherwise it's located directly on the impeller outlet.

**Flow direction (angle)**

Beside the geometrical information the flow direction is an important property. The flow direction at the component inlet is defined by the flow direction at the outlet of the upstream component.
Outlet flow direction of a component is determined by its blade or by constant swirl for vaneless components.

The first component of the project has no predecessor and gets the flow direction information from pre-swirl definition in the Global setup.

**Possible warnings**

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
</table>
| Flow inlet does not match previous outlet  
Flow outlet does not match next inlet | Check both sides (components) if the hub and shroud points are identical and no gap between the components exists.  
For stators, the "Inlet + Outlet" mode is more suitable to define exact geometry endpoints. For "Center line" specifications, one of the pre-defined angle definitions should be used if possible.  
Please note that for volutes the inlet is always axis-parallel. An upstream stator can only match the volute inlet if its outlet is also axis-parallel.  
For impellers, an existing gap of the flow region can alternatively be closed by  
- secondary flow path design in the meridional contour design step  
- extension (with option "connected") or RSI connection in the CFD setup |

**3.4.3 Automatic calculations**

Some component design steps contain automatic calculations.

Currently these are:

- **Impeller main dimensions**: calculation of empirical parameters and efficiencies, calculation of dimension values
- Impeller blade properties: calculation of blade angles, $\beta_{B1}$, $\beta_{B1}$ (meanline design mode) or Profile properties (airfoil/ hydrofoil design mode)
- **Volute development areas**: calculation of spiral contour using updated inlet flow properties
These automatic calculations can be activated or deactivated. Both approaches have their specific advantages and disadvantages:

- **Automatic calculation:**
  It's assured that the calculation results are up-to-date based on the latest input parameters. The formerly used values could be modified.

- **No automatic calculation:**
  It's assured that the exact original values are used, which were calculated or specified formerly, including optional manual adjustment. The values could be not suitable to any modifications of input parameters or modified geometry parts.

When opening older CFturbo projects containing automatic calculations the calculated values can deviate from the original values due to the re-calculation - therefore the geometry could be modified slightly compared to the original one. Generally it's recommended to **deactivate** all automatic calculations after the design process is finished and the CFturbo file is archived.

If a CFturbo project was created by an older version and contains automatic calculations the user will be asked for deactivating it when opening such a file. This should assure identical geometry over several CFturbo versions.
3.5 Windows Explorer integration

CFturbo offers a Windows Explorer integration to support the user handling CFturbo design files by offering information on the content without the need to open it.

There are 3 different ways in which information is displayed, described in detail below.

**Preview**

The preview pane of the Windows Explorer can be used to display a preview of the content of a file. It can be enabled on the view ribbon/menu of Windows Explorer or by pressing Alt + P and will show an excerpt of the report and an image of the meridian shape of the design.

Note: Previews are only available for CFturbo design files created or modified with CFturbo 10.2 and newer.

**Property sheet**

For CFturbo design files a property sheet is available in the file properties (right click on file) showing the project information.
Additional columns and Windows Search

In Windows Explorer additional columns are available for CFturbo design files:

- Title
- Author
- Comment
- File version
- Classification
- Project type
- Creation date
- Modification date
- Component count
Columns can be added to the details view by using the context menu of a column header (right click on header) and selecting "More..." from the menu. It's possible to select multiple columns at once. After confirming the selection, the new columns will show up in the Windows Explorer view.

The columns can be used for sorting.

The information from this columns is also indexed by the Windows Search, allowing you to find your CFturbo design files not only based on their file name but also based on these values.

The general Indexing Options of Windows can be set using Windows system settings. The folders containing your files should be included in the list of locations for them to get indexed. Usually it can take some time for the indexer to add new files. In the advanced Indexing Options you can also force a rebuild of the index.

For further details about the Windows Explorer and Windows Search see the Windows documentation.
3.6 Logging

CFturbo provides logging information when running.

File location

The resulting log file is saved in the following directory, depending on the run mode:

- **Interactive**: 
  C:\Users\<user name>\AppData\Local\Temp\CFturbo\<Version>\Log
  The log file is deleted at program shutdown if it does not contain any error messages.
  Otherwise, old files are deleted automatically if they are older than 7 days.

- **Batch mode**: 
  directory of batch file *.cft-batch

Message levels

The log file contains logging information in the following levels:

- Information
- Warning
- Error

File format

The log file is a simple text file. A single line in the log file has the following format:

<Date> <Time> [<Message level>] <Message category> <Message>

Example:

2020-05-27 17:01:07 [ERROR] ProjectManager - Requested project type not supported.

3.7 Troubleshooting

This chapter provides information on how problems can be handled:

- Error reporting
- Emergency recovery
- Known problems
3.7.1 Error reporting

CFturbo includes an error reporting function which helps you to send the relevant information to the support team.

As bug reports help us to find and solve problems, we **always recommend** to send the report and include as much information as you can provide to reproduce the error.

If an error occurred a window will appear that informs you about the error and provides 3 options:

- **Send bug report**
  
  Follow the *Send assistant* to add user and contact information as well as configuring the bug report. Finally, the report will be sent to our web server encrypted.

- **Show bug report**
  
  View collected information that will be included in the bug report.

- **Continue application (Default)**
  
  Continue working with CFTurbo without sending the bug report.

**Send assistant**

The Send assistant will guide you sending the bug report.

In the first step, you will be asked for your contact information so that the support team is able to contact you if additional information is needed or a solution for the problem is available.
The second step asks you for the details of the situation, the error occurred in. Please note that it is extremely helpful if the error can be reproduced.

Here you also can choose, if the currently loaded project should be attached to the bug report.

Finally you can choose if a screenshot should be attached. If Continue is clicked, the report will be sent encrypted to our web server.
If automatic sending fails, e.g. due to missing network connection, a mail with all details and attachments will open in your default mail client and you have to send it manually.

**Detail view**

The detail view shows you the information that is collected about the error and the current state of CFturbo. Also basic system information is included.
### 3.7.2 Emergency recovery

If CFturbo terminates abnormally the last project state is still available and can be restored at next program start.

In this case the following message is displayed and one can open this last project state optionally.
The last project state is the newest item of the Undo list of the previous project.

### 3.7.3 Known problems

The following table lists known problems together with their possible solutions:

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>When CFturbo is started, the following error message is displayed:</td>
<td>Update the graphics card driver.</td>
</tr>
<tr>
<td><img src="image1.png" alt="Error message" /> LoadLibrary failed on Windows 10</td>
<td></td>
</tr>
<tr>
<td>The anti-aliasing of the 3D model does not work.</td>
<td>Anti-aliasing (MSAA) settings of CFturbo should not be overridden by graphics card driver. Change your driver settings to use application settings.</td>
</tr>
</tbody>
</table>
Part IV
4 Getting started

This chapter describes how to start using CFturbo:

- Start
- Opened project
- Component design process
- Activate/ Rename/ Delete components
- Remove design steps

4.1 Start

After starting the program you see the following screen:

Create new project

Here you can create a new project by selecting the desired machine type:
These 4 buttons correspond to the menu item File/ New.

After creating a new project the Global Setup dialog is starting automatically.

 Afterwards several components can be added to the project.

Open existing project

Here you can select existing projects:

- **Select CFturbo file...** Open any CFturbo project (*.cft) via file opening dialog (corresponds to the menu item “File/ Open”).

- **Recent projects** Here you can select one of the 10 recently used projects. The full filename is displayed as a hint if you move the mouse cursor over any item.

  - **Open directory...**
  - **Remove this item**
  - **Clear entire list**

- **Sample projects** Open one of the CFturbo sample projects from the installation directory.

Alternatively, a CFturbo project can be opened by drag & drop the corresponding *.cft file.

For Recent projects and Sample projects, a preview of the corresponding project is available for the item under the mouse cursor if there is enough space to the right of the lists.
4.2 Opened project

After creating a new design or opening an existing project the main window looks as shown below:
On top you can find the ribbon style menu providing access to all functionality. Some of the ribbon pages are context sensitive.

The CFturbo application window is divided into three main areas:

a) Component list on the left side

This ordered list contains an icon for each component of the project. The currently selected component is framed.

Clicking on the icon selects the component (alternatively you can click on component in the meridional view).

At inlet or outlet side of the selected component you can add additional components by the buttons.

After selecting a component, the ribbon changes to the specific tab for this component type. The context menu of the icons allows (de)activating, renaming and deleting the component.

The following component types are possible:

- Centrifugal or mixed-flow impeller
- Axial impeller
- Stator (vaned or unvaned)
- Volute

b) Three alternative views in the central part

see Views

c) Message panel on the right side

The message panel shows errors, warnings and information for all components of the project. The design step causing the message is also displayed by icon.

It depends on the opinion of the user to accept warnings or to modify the design by adequate actions to avoid them. Reasons for errors should be eliminated.
The overall number of errors, warnings and information is displayed above, where you can filter according to these categories.

If a help link is available providing additional information concerning the message, a question mark is shown next to the cursor. The help can then be opened by clicking on the message.

4.3 Component design process

The design process for CFturbo project components requires the completion of a specific sequence of obligatory design steps for each component type (see impeller, volute, stator).

After completing a components basic design process, optional design steps related to model finishing and CFD setup become available.

Each design step comes with its own dialog that can be accessed via the component specific menus or the components context menu in the meridian view.

If the design step is executed for the first time, CFturbo generates an initial design automatically. "(Default design)" is added to the dialog title bar in this case.

**Design step dialog controls**

Generally, dialogs in CFturbo provide the following standard controls:

- **OK** Closes the dialog and saves user changes into the project.
- **Cancel** Closes the dialog and discards all changes made.
- **Help** Opens the help topic related to the current design step.
Getting started

Undo (Ctrl Z)/ Redo (Ctrl Y) the last action, which is displayed as hint.

Additional options:

**Reset to default**
Reset to default settings (current modifications are lost)

**Restore from project**
Restore previous project state (current modifications are lost)

Fast Navigation and Automated component design

Dialogs that are part of the basic component design process provide navigation buttons:

This feature enables you to quickly and comfortably navigate to any other design steps.

Any existing design step can be selected as well as the next design step for new designs by

- **Left** mouse button: close current design step with `<OK>` (save user changes into the project)
- **Right** mouse button: close current design step with `<Cancel>` (ignore user changes)

Currently not existing design steps are disabled.

**Complete all design steps**

- For new design steps this button closes the dialog and saves user changes into the project. Finally, it completes all subsequent mandatory design steps of the selected component with default values.
- This option is only available if the selected component has a next design step that has never been completed.
You may use this option as soon as the main dimensions of a component are defined to get to a preliminary automatic design within seconds. You can change all design parameters according to your requirements later on.

The automatic design may fail or lead to unsatisfactory results if global project settings and/or previously completed design steps are unsound. In this case you will be informed about the issue via warnings in the message panel or a message box.

**Update Messages**

After any modification the current design step is checked for information/ warnings/ errors.

These messages are displayed close to the <OK> button of the design step dialog. Usually you can find more information about a message in the online help by clicking on its text.

If the design step dialog is closed by <OK> then this information as well as the messages of all other design steps is also displayed in the Messages area right in the main form.

See also [Opened project/ message panel](#). Usually you can find more information about a message in the online help by clicking on its text.

After any design step modification all dependent design steps are updated automatically.

**4.4 Component handling**

The general component handling can be executed in the following manner alternatively:

- Context menu of the corresponding component left in panel **Components**
- Context menu of the corresponding component right in the **Meridian** view
**Active**

An inactive component is read-only and also not going to be updated automatically. Inactive components are colored gray in all views.

**Rename**
Change the caption of a component. The caption is displayed left in the components list as a hint when moving the mouse cursor on the icon, in the Meridian view, the 3D view and the report.

**Delete**

The selected component is deleted. If the Meridian view is selected, the `<Del>` key on the keyboard can be used alternatively.

**Color**

A component color can be selected from 15 predefined colors. The color is used left in the components list and in the Meridian view. Furthermore it’s used for text color in the report, the 3D model tree and the export window.

4.5 **Remove design steps**

If you make any design modifications on the current component then all following design steps are adapted automatically (parametric model).

However, if you would like to start with an automatic generated CFturbo initial design, certain design steps can be removed manually. Then CFturbo continues with new initial design data. For that purpose you have to select the appropriate design step to be removed and then press the **OK**-button.

Of course, all following design steps after the selected one are removed too.
4.6 Handling

This chapter contains some general information about program handling:

- General handling
- Graphical dialogs
4.6.1 General handling

Value input

Value input in

- Edit controls
- String grids

is not applied automatically, but by

- Pressing <Enter> or
- Leaving the focused control by selecting another one by mouse-click or by pressing <Tab>

(behavior like in Microsoft Excel)

Miscellaneous

- Position and size of dialogs are saved to restore it in the same way when they are called again.

4.6.2 Graphical dialogs

Most component design step dialogs contain 2D graphical representation. The user interface is uniform concerning the following topics.

Diagram popup menu

All graphical representations are made in diagrams that are automatically scaled according to displayed objects. All diagrams have a popup menu (right click on empty diagram area) with basic functions. Alternatively you can use the buttons on the top side of the diagram:
Getting started

© CFturbo GmbH

- Zoom window by mouse
- Fit view
- Lens magnification at mouse cursor position
- Copy diagram to clipboard
- Save diagram as BMP, GIF, JPG, PNG or WMF
- Print diagram
- Paste points from clipboard into an extra polyline
- Add any polyline from file (x,y points) to compare different curves (alternatively by drag & drop)
- Remove all imported polylines
- Measure distance (use left-click to define start position and keep the button pressed until the mouse cursor reached the end position)
- Configure diagram

Context sensitive popup menus

If the mouse cursor is moved over a graphical object (e.g. polyline, Bezier point) then this is highlighted by color or by increased line width. Right mouse click is now related to this object and does open a special popup menu or a small dialog window for data input.

Bezier curves are used for geometrical contours by default. This continuous polylines are described by the position of a few Bezier points. Therefore a simple modification of the curve is possible but on the other hand the numerical representation of the curve is accurate.

For Bezier curves popup menus are available for special actions concerning the curve.

An alternate method to specifying Bezier points by the mouse, you may enter the accurate coordinates of Bezier points in a small dialog window that appears by clicking the right mouse button on the chosen Bezier point.

One or two coordinate values can be entered in dependence of geometrical boundary conditions. As a rule these values are normalized relative values describing the position of the point between...
extreme values left or bottom (0) and right or top (1). Normalized relative coordinates are giving the advantageous possibility of an automatic update of the entire design if a parameter is modified.

Display options

Some diagrams (both main and additional progression diagrams) have several display options to switch on/off some elements. These display options can be handled by a menu in the lower left corner of the diagram.

The state of each display option is saved internally and restored next time.

Miscellaneous

- Coordinates of mouse cursor are displayed in format x:y bottom left in the status bar.

Auto fit view (for docking diagrams)

For docking diagrams, automatic fit view can be switched on/off in the upper right corner when moving the mouse over the diagram.
4.6.3 Progression dialog

This dialog allows to set different progression types for a given variable.

![Progression dialog](image)

**Availability**

The Progression dialog can currently be used for the following variables:

- Cross section progression, in Meridional contour
- Angular positions, in Blade mean lines
- Spiral cross section progression, in Spiral development areas

Visibility of the single progression types depends on the specific context of use.

**Import Polyline**

If the option Polyline is selected, a text file containing a user defined progression can be imported.

Text file format:
# cross section distribution
# start/end tangential, 
# midsection linear
# (spline interpolation 9 points)
0.00 0.00000
0.04 0.01728
0.08 0.03830
0.12 0.06368
0.16 0.09404
0.20 0.13000
0.24 0.17164
0.28 0.21687
0.32 0.26314
0.36 0.31018
0.40 0.36000
0.44 0.41404
0.48 0.47102
0.52 0.52898
0.56 0.58596
0.60 0.64000
0.64 0.68982
0.68 0.73686
0.72 0.78313
0.76 0.82836
0.80 0.87000
0.84 0.90596
0.88 0.93632
0.92 0.96170
0.96 0.98272
1.00 1.00000

- All lines starting with a "#" symbol are comments. All other lines contain the numerical values.
- x and y coordinate values can be separated by "comma", "semicolon", "space" or "tabulator".
- "Dot" character is required to be used as decimal separator.
- Values are imported in the currently active units of the diagram axes.
- The file can have any or no filename extension.

A sample file can be generated by right clicking the progression curve and selecting “Save polyline”.

### 4.6.4 Polyline to Bezier

In several design steps, imported polylines can be converted into Bezier curves for further modifications.

The approximation window can be started via the context menu of the curve, typically.
First the desired polyline is imported via "Import polyline".

The imported curve is displayed in red color, the original curve in black.

By pressing the "Start!" button, the position of the Bezier points is calculated in order to approximate the imported polyline as exact as possible.

The Bezier points can be adapted for better matching the imported curve.

4.6.5 Edit fields with empirical functions

Some edit fields are connected with empirical functions. This becomes visible when activating the edit field by mouse click.

Default

Default appearance of edit field.
Mouse-over

Appearance if the mouse cursor is over the edit field. Min. and max. values are displayed if a recommended range exists.

Focused

Appearance, if the edit field is focused (mouse click into the field). If a recommended range exists, min. and max. values are displayed as well as a sliding bar below.

Default value

The default value can be selected by pressing the arrow button left. The numerical default value is displayed as hint.

Empirical function

The connected empirical function can be displayed by pressing the settings button on the right side. Furthermore the currently selected function is visible as hint of this button.
5 Menu

In CFturbo all menus of the main window are located in a ribbon with tabs. Every tab page contains groups with control elements.

Hints

The buttons have hints if they are not self-explanatory. The hint becomes visible when the mouse cursor is on the button.

Keyboard shortcuts

Key tips are displayed, when you press and release the ALT key.

In order to execute a command, you have to press the the ALT key and the shown key(s) one after another.

In the title bar the quick access toolbar is placed. It can be customized by using the context menu of any element in the ribbon.

The file menu left in the ribbon contains basic file handling operations.

The tab pages contain control elements grouped by functionality:

<table>
<thead>
<tr>
<th>page</th>
<th>visible</th>
</tr>
</thead>
<tbody>
<tr>
<td>PROJECT</td>
<td>always</td>
</tr>
<tr>
<td>SETTINGS</td>
<td>always</td>
</tr>
</tbody>
</table>
5.1 File

The file menu can be found on the left border of the ribbon and contains the basic file operations.

Right behind the menu buttons you can open one of the recently used files by selecting it from the list.

This list is also available in the main window directly after starting the program (see Start).

5.1.1 Create new design

? File | New

When creating a new project one of the following project types can be selected:

- Pump
- Fan
- Compressor
- Gas Turbine
- Hydro Turbine

Equivalent to using this menu, the buttons in the **Create new project** area can be used, see **Start**.

The **Global Setup** dialog will be started automatically right after creating a new project.

After finishing the Global Setup you will see an empty project where you can add components.
More information

- General information about adding new components: see Add component
- Specific for multi-stage machines: see Stage designer

5.1.1.1 Stage designer

Using the stage designer is an alternative method of creating multi-stage machines.

The number of stages can be specified and for each impeller the shape and the type can be selected. The distribution of the energy transmission defined in the Global Setup to the individual stages can be relative or absolute. As a result, the corresponding impellers are created with their Main dimensions and Meridional contour.

On the right side a meridional preview is available as well as a table with the most important thermodynamic values of each impeller.
The space between the designed impellers can be filled with vaned or vaneless stators afterwards.

Each impeller created by the stage designer is considered to have no inlet swirl. That is to say its property **consider upstream swirl** is false by default, see page Setup on Main dimensions. Normally there will be vaned stationary components located between adjacent impellers that will change the swirl in a certain way. Very often that change will result in a zero inflow swirl for the next impeller.

### 5.1.2 Open/ Save design

[File | Open/ Save/ Save as]

CFturbo projects are saved as *.CFT files (XML file format).

A list of recently used files is available by selecting the menu **File | Recent projects**. Alternatively, you can select the design directly from the list **Recent Projects** if no design is opened, see **Start**.

The user can modify the filename by the **Save as**- function in order to save modified designs under different file names.
5.2 PROJECT

A project can consist of several components (see Project structure and interfaces). All components can be designed separately, whereas they influence each other on the interfaces due to geometrical constraints and fluidic coupling.

The Project menu is split into general and additional functions.

5.2.1 General

PROJECT | General

The group General contains all those actions that are related to the whole project.

- Project information
- Global setup
- Performance prediction
- Undo

5.2.1.1 Project information

PROJECT | General | Project Information
For identification of the project can be specified:

- Project name
- Classification (e.g. version or sub name)
- User name
- Comments

This information is not mandatory and should support the identification of CFturbo projects & sessions.

The working directory, the creation date and the date of last modification are displayed too.
5.2.1.2 Global setup

? PROJECT | General | Global setup

Here the global project settings are defined valid for all components.

Depending on the project type different input parameters are required (see below). As examples you see the Global setup dialog for pumps on the left side, for compressors on the right side.

![Global setup dialog for pumps and compressors]

**Design point**

Here you have to enter the design point data:

1. **Flow rate:**
   - for pumps, fans, hydro turbines: volume flow $Q$ or mass flow $m$?
   - for compressors: mass flow $m$ or volume flow $Q$ (referring to total state on suction side)
   - for gas turbines: mass flow $m$?

2. **Energy transmission:**
   - for pumps: head $H$ or total pressure difference $\Delta p_1$
   - for fans: total pressure difference $\Delta p_1$
for compressors: total pressure ratio $\pi_t$ or total pressure difference $\Delta p_t$ or specific work $Y$

for gas turbines: total pressure ratio $\pi_{tt}$ or total-to-static pressure ratio $\pi_{ts}$

for hydro turbines: head $H$

(3) Number of revolutions $n$

Energy transmission and number of revolutions can be specified separately for each impeller in the project, see Main dimensions. The sum of energy transmissions of all impellers in multi-stage machines should be equal to the Global Setup value.

Fluid

Here the fluid has to be defined.

One has to select one of the predefined fluids. The list of existing fluids can be modified in the Fluid manager.

For compressors and turbines the gas model has to be specified additionally: Perfect, Redlich-Kwong, Aungier/Redlich-Kwong, Soave/Redlich-Kwong, Peng-Robinson or CoolProp.

Inlet conditions

Here the total state at the inlet - total pressure $p_t$ and total temperature $T_t$ - has to be defined. The latter applies only for compressors and turbines. For pumps and fans the inlet total pressure is not design relevant but will be used within the interfaces for the CFD export as well as for the calculation of informative values at the interfaces of any component.

Optional

Here some optional parameters can be defined. Their default values remain unchanged normally.

- Direction of impeller rotation, seen in negative axis direction.

- Additional efficiency, which contains all additional (non-typical) flow losses in impeller and casing parts of the machine. This efficiency value is used for impeller dimensioning as well as overall efficiency calculation in addition to the efficiency values specified in the impeller design.
Pre-Swirl [for pumps, fans, compressors only]

Here you may define the inflow swirl at hub and shroud. The following definitions are available:

<table>
<thead>
<tr>
<th></th>
<th>Flow angle</th>
<th>Swirl number</th>
<th>Swirl energy number</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$\alpha = \arctan(\frac{c_m}{c_u})$</td>
<td>$\delta_r = 1 - \frac{c_u}{u}$</td>
<td>$\delta_Y = \frac{uc_u}{Y}$</td>
</tr>
<tr>
<td>Positive swirl</td>
<td>$\alpha &lt; 90^\circ$</td>
<td>$\delta_r &lt; 1$</td>
<td>$\delta_Y &gt; 0$</td>
</tr>
<tr>
<td>Negative swirl</td>
<td>$\alpha &gt; 90^\circ$</td>
<td>$\delta_r &gt; 1$</td>
<td>$\delta_Y &lt; 0$</td>
</tr>
<tr>
<td>No swirl</td>
<td>$\alpha = 90^\circ$</td>
<td>$\delta_r = 1$</td>
<td>$\delta_Y = 0$</td>
</tr>
</tbody>
</table>

Negative swirl is increasing the head and may often have no good affect to the suction behavior. Inflow through a straight pipe usually leads to swirl-free flow.

The different parameters can be converted:

$$
\delta_r = 1 - \frac{c_m}{u \tan \alpha} = 1 - \frac{4Q}{\pi \left( \frac{d_S^2}{4} - \frac{d_H^2}{4} \right) \tan \alpha}
$$

$$
\delta_r = 1 - \frac{\delta_Y \cdot u^2}{u^2} = \frac{u^2 \left( 1 - \delta_r \right)}{Y}
$$

The conversion $\delta_r - \alpha$ is only valid for certain diameters $d_u$ and $d_s$.

Values

Except for radial-inflow turbines the general meridional shape of the machine, depending on the specific speed, is displayed in the right Values area:
Furthermore some calculated variables are displayed:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Specific speed</strong></td>
<td>Points to machine type and general shape of impeller (see Specific speed definitions)</td>
</tr>
<tr>
<td><strong>Specific energy Y</strong></td>
<td>Pumps, Fans, Hydro Turbines: $Y = gH = \Delta p/\rho$ Compressors, Gas Turbines (perfect gas model): $Y = \left(\frac{\kappa - 1}{\kappa} - 1\right) c_p T_t$</td>
</tr>
<tr>
<td><strong>Power output $P_Q$</strong></td>
<td>$P_Q = \dot{m} Y$ Pumps, Fans: $P_Q = \rho gHQ$</td>
</tr>
<tr>
<td><strong>Mass flow $m$</strong></td>
<td>Pumps, Fans, Hydro Turbines: $\dot{m} = \rho Q$ Compressors: $\dot{m} = Q_i \cdot \rho_t (\rho_t, T_t)$ (density according to gas model)</td>
</tr>
<tr>
<td><strong>Total pressure difference $\Delta p_t$</strong></td>
<td>Pumps, Fans: $\Delta p_t = \rho gH$</td>
</tr>
</tbody>
</table>

**Compressor:**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Volume flow (total)</strong></td>
<td>$Q_i = \frac{\dot{m}}{\rho_t (\rho_t, T_t)}$ (density according to gas model)</td>
</tr>
<tr>
<td><strong>Inlet speed of sound (total)</strong></td>
<td>(perfect gas model)</td>
</tr>
<tr>
<td><strong>Inlet density (total)</strong></td>
<td>(density according to gas model)</td>
</tr>
</tbody>
</table>
Outlet pressure (total) \[ p_{12} = p_1 (\pi_{11}, \Delta \pi, Y) \]

Outlet temperature (total, isentropic) \[ T_{12is} = T_1 (\pi_u^{\kappa-1}) \] (perfect gas model)

**Gas Turbine:**

Volume flow (total) \[ Q_i = \frac{\dot{m}}{\rho_i (\pi, T)} \] (density according to gas model)

Inlet speed of sound (total) \[ a_i = \sqrt{\kappa \cdot R_{gas} \cdot Z_i \cdot T_i} \] (perfect gas model)

Inlet density (total) \[ \rho_i = \rho_i (\pi_i, T_i) \] (density according to gas model)

Outlet pressure (total) or static \[ p_{12} = p_i (\pi_{11}, \pi_u) \]

Outflow pressure \[ p_2 = p(\pi_{11}, \pi_u) \]

Outlet temperature (total, isentropic) or static, isentropic \[ T_{12is} = T_1 (\pi_u^{\kappa-1}) \] (perfect gas model)

Hydro Turbine:

Net head \[ H_n = H \cdot \eta_h \]

**Cordier**

The **Cordier diagram** contains relationship between specific speed and specific diameter.

See **Cordier** [here]
General remarks

- In general for cost reasons single-stage and single-intake machines are preferred covering a range of about $10 < n_q < 400$.

- In exceptional cases it may become necessary to design an impeller for extremely low specific speed values ($n_q < 10$). These impellers are characterized by large impeller diameters and low impeller widths. The ratio of free flow cross section area to wetted surfaces becomes unfavorable and is causing high frictional losses. To prevent this one may increase either rotational speed $n$ or flow rate $Q$ if possible. An alternative solution could be the design of a multi-stage machine reducing the energy transmission of the single-stage.

- If especially high specific speed values ($n_q > 400$) do occur one can reduce rotational speed $n$ or flow rate $Q$ if feasible. Another option would be to operate several single-stage machines - having a lower $n_q$ - in parallel.

- Please note: CFturbo is preferably used between $10 < n_q < 400$ – centrifugal, mixed-flow and axial impellers.
# Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Energy transmission of all impellers is different to globally defined value.</strong></td>
<td>The sum of energy transmission defined for each impeller deviates from the globally defined value in Global setup. Check and adapt the energy transmission of the impellers (see <a href="#">Main dimensions</a>) to get altogether 100% of the initially defined value of the Global setup.</td>
</tr>
<tr>
<td><strong>Total inlet pressure too small.</strong></td>
<td>Hydro turbines only: Head together with the inlet total pressure yields a discharge total pressure smaller then 0 with: $p_{\text{t, out}} = p_{\text{t, in}} - \rho \cdot g \cdot H$. Increase total inlet pressure.</td>
</tr>
</tbody>
</table>

## 5.2.1.3 Performance prediction

The Performance prediction is an empirical based estimation of the performance map of the machine.

The performance prediction is not available for:
- incomplete impellers
- impellers in "Manual dimensioning" mode (see [Main dimensions/ Setup](#))
- axial gas turbines

Two approaches are used to estimate the characteristics:

- **Euler**: Losses are estimated and subtracted from the theoretical Euler-line.

- **Casey/Robinson**: Characteristics will be derived from similarity considerations. For details see [Casey/Robinson](#). This model is applicable for compressors only.

**Please note**: This is an estimation. The actual performance may differ from the prediction.
General

A performance curve of the current design is estimated on the basis of the Euler-Equation:

\[ H_{th} = \frac{1}{g} (u_2 \cdot c_{u2} - u_1 \cdot c_{u1}) \quad \text{and} \quad Y_{th} = \frac{\Delta p_{th}}{\rho} = u_2 \cdot c_{u2} - u_1 \cdot c_{u1} \]

respectively.

In these and all the following equations all variables are averaged values. E.g. the circumferential velocity \( u_2 \) is calculated with an average impeller diameter \( d_{M2} \) that is the impeller diameter \( d_2 \) for centrifugal impeller and the area averaged diameter for axial impeller respectively. The latter reads as:

\[ d_{M2} = \sqrt[2]{\frac{1}{2} (d_{s2}^2 + d_{H2}^2)} \]

Variables

All types of turbo machines have in common: The characteristics can be displayed in a diagram with dimensions as well as without dimensions.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Pump</th>
<th>Fan</th>
<th>Compressor</th>
<th>Gas Turbine</th>
<th>Hydro Turbine</th>
</tr>
</thead>
<tbody>
<tr>
<td>( H )</td>
<td>head</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>net head</td>
</tr>
<tr>
<td>( \Delta p )</td>
<td>pressure difference (total-total)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>( \Delta p_{ts} )</td>
<td>-</td>
<td>pressure difference (total-static)</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>&amp;  &amp;  &amp;  &amp;  &amp;</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>( \Delta p_{ts} = \Delta p - \frac{\rho}{2} c_2^2 )</td>
<td>&amp;  &amp;  &amp;  &amp;</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>( \psi )</td>
<td>work coefficient</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>&amp;  &amp;  &amp;  &amp;  &amp;</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>( \psi = \frac{g \cdot H}{u_2^2/2} )</td>
<td>( \psi = \frac{Y}{u_2^2/2} )</td>
<td>( \psi = \frac{Y}{u_2^2/2} )</td>
<td>( \psi = \frac{g \cdot H}{u_2^2/2} )</td>
<td>&amp;</td>
<td></td>
</tr>
<tr>
<td>( \frac{H}{H_{opt}} )</td>
<td>head ratio</td>
<td>-</td>
<td>-</td>
<td>head ratio</td>
<td></td>
</tr>
<tr>
<td>Symbol</td>
<td>Definition</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>-------------</td>
<td>-------------------------------------------------</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\Delta p/\Delta p_o$</td>
<td>total pressure difference ratio</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\pi_{tt}$</td>
<td>pressure ratio (total-total)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\pi_{ts}$</td>
<td>pressure ratio (total-static)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\eta$</td>
<td>efficiency</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\eta_r$</td>
<td>efficiency incl. motor</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\eta_v$</td>
<td>volumetric efficiency</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$P$</td>
<td>required driving power</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$Q$</td>
<td>volume flow</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\varphi_m$</td>
<td>meridional flow coefficient</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\varphi$</td>
<td>flow coefficient</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$Q/Q_{\text{opt}}$</td>
<td>flow ratio</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$Q_t$</td>
<td>volume flow total</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- $P = \frac{\rho \cdot g \cdot H_{\text{Deer}}}{\eta_{\text{mech}} \eta_{\text{mot}} \eta_{s_f}} \cdot (Q + Q_{\text{leak}})$
- $P = \frac{\rho \cdot Y_{\text{Deer}}}{\eta_{\text{mech}} \eta_{\text{mot}}}$
- $P = \frac{\rho \cdot g \cdot H_{\text{Deer}}}{\eta_{\text{mech}}}$

- $Q = \frac{\dot{m}}{\rho}$
- $Q = \frac{\dot{m}}{\rho_2}$
- $Q = \frac{\dot{m}}{\rho_1}$
- $Q = \frac{\dot{m}}{\rho}$

- $\varphi_m = \frac{c_{m2}}{u_2}$
- $\varphi_m = \frac{c_{m1}}{u_1}$

- $\varphi = \frac{Q}{\pi/4 \; d_2^2 u_2}$
- $\varphi = \frac{Q}{\pi/4 \; d_1^2 u_1}$
All combinations of flow and energy variables are possible.

It is common practice in the case of gas turbines - contrary to all other type of turbo machines - that the flow variable is given as a function of the energy variable. Beyond it characteristics of different rotational speeds will not be displayed over the whole theoretical pressure interval but only piecewise.

The choice of the variables is to be made in the tab “Variables”.

**Reference curves**

For comparison purposes with the present design saved designs can be loaded (soft button “configure”).

**Information**

On the right hand site in the panel information some design point information can be found. Beyond it also the mass flow (or equivalent) for the tangential (shockless) flow towards the leading edge of the impeller blades as well as its relative deviation from the design point is given. The value of the shockless flow is also represented as a vertical line in the performance diagram. The visibility of that line can be toggled by the display options (lower corner in the left).
The **Euler-based** performance prediction is available for all types of turbomachines.

### Kinds of losses

There are different kinds of losses that are considered in different curves:

<table>
<thead>
<tr>
<th>Kind</th>
<th>Description</th>
<th>Parameter</th>
</tr>
</thead>
</table>
| Decreased power | Based on the Euler-Equation and the decreased power that is calculated in the Blade properties \( \Delta H_{\text{Decr}} \). In the design point the decreased power line is shifted by a pressure head loss equivalent to the decreased power \( (H_{\text{Decr}} = H_{th} - \Delta H_{\text{Decr}}) \). The decreased power line can be parallel to the Euler-Line at \( \Delta p = 0 \). | \( c_l \):  
\( c_l = 1 \)...parallel position,  
\( c_l = 0 \)...intersection with Euler-Line at \( \Delta p = 0 \). |
Euler-Line as well as positioned that way, that it intersects the Euler-Line at $\Delta p = 0$.

<table>
<thead>
<tr>
<th>Hydraulic losses</th>
<th>Based on the Euler-Line including the decreased power minus the losses due to friction. Yields a downwards opened parabola, that touches the decreased power curve at $Q = 0$.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shock losses (by turbulence and separation)</td>
<td>Includes all the effects listed above plus turbulence and separation losses at the inlet and outlet. Yields a downwards opened parabola. It touches the curve, in which decreased power and hydraulic losses are considered, in the point of shockless flow $Q_{opt}$. Here the flow direction is tangential towards the leading edge.</td>
</tr>
</tbody>
</table>

$\zeta_h$: General approach: 
\[ \Delta H_{Hydr} = \zeta_h \cdot F \cdot Q^2 \]

$F$: Flow factor that considers the geometry of the component (inlet and outlet area)

$\zeta_s$: General approach: 
\[ \Delta H_{Shock} = \zeta_s \cdot F \cdot (Q - ...) \]

The display of resulting performance curves can be toggled by the check box "All performance curves" (display options lower corner in the left). In case the curves are to be hidden only the actual performance curve (red color) considering all losses will be visible.

A loss coefficient, that describes the hydraulic losses, can be calculated by pressing "Calculate $\zeta$" in a way, that as a result the actual performance curve (red) of the flow efficiency will go through the best point. For this calculation the ratio between the loss coefficients is important. This ratio $\zeta_h/\zeta_s$ can be set in the panel Parameter, see table below, second column.

### Settings

Energy and flow rate variables plus flow rate limits (for turbines additionally a max pressure ratio)  

Coefficients influencing the decreased power (c\textsubscript{l}) and the hydraulic as well as shock losses ($\zeta_h$, $\zeta_s$)  

Additional curves with different speeds and diameter plus system characteristic

<table>
<thead>
<tr>
<th>Energy and flow rate variables plus flow rate limits (for turbines additionally a max pressure ratio)</th>
<th>Coefficients influencing the decreased power (c\textsubscript{l}) and the hydraulic as well as shock losses ($\zeta_h$, $\zeta_s$)</th>
<th>Additional curves with different speeds and diameter plus system characteristic</th>
</tr>
</thead>
</table>

© CFturbo GmbH
The two quadratic approaches towards the description of the hydraulic as well as shock losses (i.e. turbulent and separation losses) tend to generate characteristics that have their efficiency maximum at flow values smaller than the design flow. To overcome or mitigate this certain parameters can be adjusted.

The general approach for the hydraulic losses is extended by an extra offset that is caused by a blind flow $Q_{\text{Blind}}$ due to recirculation at a flow of $Q = 0$. This blind flow $Q_{\text{Blind}}$ is determined with:

$$Q_{\text{Blind}} = \frac{Q_{\text{Design}}}{2 \cdot \eta_{\text{vol}}}.$$

Herewith the hydraulic losses become:

$$\Delta H_{\text{Hydr}} = \zeta' h \cdot F \cdot \left( Q^2 + \text{weight} \cdot Q_{\text{Blind}}^2 \right),$$

where weight can be influenced by the weight factor $\zeta_w$ in the panel Parameter, see table above, second column.

To influence the determination of shock losses at $Q < Q_{\text{opt}}$ a second weight factor $c_w$ is available. With the help of this parameter the shock losses become:
Surge [ for fans, compressors only ]

The prediction of surge line is based on the following model: The pressure difference between outlet and inlet yields a back flow within the compressor. Amongst pressure difference and back flow a correlation exits, that can be found in the table "Kinds of losses", column "Hydraulic losses". Within the applied model the compressor is thought as a parallel connection between a flow source and a hydraulic resistance. Then, surge will occur when the back flow in the hydraulic resistance becomes as big as the flow in the flow source.

The surge line can be controlled by the loss coefficient "Surge loss coefficient". Of course it is impossible to consider non-steady effects that are characteristic for the onset of the surge with this model. The surge line can be displayed only in case dimensional variables has been chosen and the checkbox "Surge line" has been set (display options lower corner in the left).

With centrifugal fans surge may only happen if the pressure difference is big enough (~0.3 bar).

Choke [ for compressors only ]

Choked flow will happen if the flow reaches sonic speed somewhere in a duct. As the rothalpy is constant at any point in the flow channel the temperature (critical temperature within the narrowest cross section) at a flow at sonic speed can be calculated by:

$$T_c = \frac{c_p T_{01} + \frac{u_c^2}{2}}{c_p + \frac{Z \cdot \kappa \cdot R}{2}}$$

and critical sonic speed becomes:

$$a_c = \sqrt{Z \cdot \kappa \cdot R \cdot T_c}$$

With an approximation of the critical density and the influence of the boundary layer blockage the choked mass flow is:

$$m_{ch} = A \cdot a_c \cdot \rho_c \cdot (1 - B)$$

The blockage of the boundary layer is expressed by the factor B that is 0.02 by default. This theoretical choke line can be displayed when the checkbox "Consider choke" has been set (display options lower corner in the left).

Characteristics with different rotational speeds
With the current set of parameters performance curves with different rotational speeds can be calculated and displayed. This procedure is feasible only if the rotational speeds are not too far from the design point. If they are, similarity relations are not valid any longer.

Running a turbomachines with a speed different from the design point the resulting efficiency will be smaller as the design point efficiency. To take this into account losses are scaled with the help of a Speed/diameter correction factor $n_D$, see table Settings, last column. The resulting losses will be:

$$\text{Loss}(n) = \text{Loss}(n_{Design}) \left[ 1 - nD \cdot \left( 1 - \frac{n}{n_{Design}} \right)^2 \right].$$

**Characteristics with different diameters** [for pumps, compressors only]

Performance curves for impellers with decreased diameter can be calculated and displayed too. The decrease of the impellers means that the geometric similarity is not given anymore. Therefore performance curves are calculated by the following empirical correlations: $H' = H \left( \frac{d'}{d} \right)^{nH}$ and $Q' = Q \left( \frac{d'}{d} \right)^{nQ}$. The exponent $mH$ should be within 2..3, $mQ$ should be 1 or slightly bigger.

Similar to the correction of characteristics with different speeds those with different diameters will be corrected with:

$$\text{Loss}(D) = \text{Loss}(D_{Design}) \left[ 1 - nD \cdot \left( 1 - \frac{D}{D_{Design}} \right)^2 \right].$$

**System characteristic - pumps, fans and compressors only**

An operating point, in which a turbo machine could possibly run, can be determined by a fictive system characteristic. The display of a system characteristic can be controlled by the checkbox "System Characteristic". The system characteristic consists of a static and a dynamic part. The static part is dependent on the parameter "Geodetic Head" (pumps only) and "Static part" respectively, whereas the dynamic part is dependent on the parameter "System hydraulic resistance". The system characteristic can only be displayed if head or total pressure difference have been chosen as variable.
The Performance prediction by **Casey/Robinson** can be chosen by clicking at the appropriate button. It is available for centrifugal compressors only.

**Settings**

- Energy and flow rate variables plus flow rate limits (reset default flow rate with ![ ]
- Diverse coefficients influencing Euler work (disk friction coefficient), location of surge and choke as well as efficiency
- Additional curves with different speeds and diameter plus system characteristic

**WARNING:**
The curves are based on simple empirical estimations. Significant deviation from experimental performance data or CFD results cannot be avoided without calibration.
5.2.1.4 Undo

? PROJECT | General | Undo

The design history can be opened by clicking the undo-button. It contains all modifications from opening of the project or session in chronological order.

By selecting a list entry, this design step and all following ones are removed. Prior to that you can save the current design optionally.
5.2.2 Additional

? PROJECT | Additional

- Export
- Batch mode/ Optimization
- Reference components
- Model finishing

5.2.2.1 Export

? PROJECT | Export
The Export offers the designed geometry to be exported in standard file formats or for several CAE applications.

For geometry export you have to:

1. Select interface in panel Interfaces
2. Select component(s)
3. Set export settings
4. Press Export data button

**Export interfaces**

Available interfaces are grouped into three blocks: Basic, CAD/CAM and CFD.

Generally, there are three export types available:
• 3D model export
  all parts of the 3D model according to a pre-defined model state

• Predefined 3D model export
  flow domain or material domain as solid faces

• Point based export
  pre-defined set of points/ splines (independent of the 3D model)

For 3D model exports, the formats IGES, STEP, STL, Parasolid and BREP can be selected. The point density can be configured in the Model settings/ 3D model of each component (Impeller, Stator, Volute). The export unit is mm always.

For point based exports, the point density and export unit can be configured in the Model settings/ Point export of each component (Impeller, Stator, Volute). If the blade shape is ruled surface then points of mean lines as well as profiles (pressure and suction side) are not affected by the model settings for the point based export.

Please note: The results of surface-based operations, e.g. fillets, cannot be exported to point-based formats.

Components

The list contains all components of the project. If the interface supports multi-component export then you can select multiple components, otherwise only a single one. For 3D model exports, no component can be selected because the geometry to be exported is defined by its visibility in the 3D model.

Some of the interfaces support special component types only, e.g impellers. Therefore some of the components could be deactivated.

Settings

Additional parameters depending on the selected export interface can be specified on the right side of the window.

Export

Above the log area the export destination and the base name of exported files can be specified.

By pressing the Export data button the export procedure is started. Some logging information are displayed in the log area.

For some CAD and CFD applications the exported geometry can be opened in the target application automatically. The product version has to be selected from a list or the installation directory can be defined manually.
### Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>CFD setup</strong></td>
<td></td>
</tr>
<tr>
<td>Blade tip projection to casing recommended (see CFD setup).</td>
<td></td>
</tr>
<tr>
<td>Blade tip projection not accomplished.</td>
<td>Check “Blade projection” in CFD setup</td>
</tr>
<tr>
<td><strong>Gap between leading/trailing edge and inlet/outlet required.</strong></td>
<td></td>
</tr>
<tr>
<td>Select stator on inlet/outlet side.</td>
<td></td>
</tr>
<tr>
<td>Alternatively, the CFD extension can be activated (see CFD setup).</td>
<td></td>
</tr>
<tr>
<td>Some space around blade edges is required for meshing. This can be generated by creating a CFD extension or by selecting a neighbouring stator component. Note for TurboGrid: a vaneless stator has to be selected, which has to be considered as part of the rotating domain in TurboGrid.</td>
<td>Try to increase the distance between leading/ trailing edge and meridional inlet/ outlet by a) moving leading/ trailing edge in meridional contour if edge is not fixed on inlet/ outlet or b) selecting a neighbouring stator if possible. or c) activating CFD-Extension in CFD setup/ Extension</td>
</tr>
<tr>
<td><strong>Gap between leading/trailing edge and inlet/outlet recommended.</strong></td>
<td></td>
</tr>
<tr>
<td>CFD extension can be activated (see CFD setup).</td>
<td></td>
</tr>
<tr>
<td>Some space around blade edges is recommended.</td>
<td>Try to increase the distance between leading/ trailing edge and meridional inlet/ outlet by a) moving leading/ trailing edge in meridional contour if edge is not fixed on inlet/ outlet or b) activating CFD-Extension in CFD setup/ Extension</td>
</tr>
<tr>
<td><strong>Small gap between blade/leading edge and inlet/outlet may cause problems.</strong></td>
<td></td>
</tr>
<tr>
<td>Problem</td>
<td>Possible solutions</td>
</tr>
<tr>
<td>------------------------------------------------------------------------</td>
<td>-------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Increase it in case of import problems.</td>
<td>See message. Try to increase the distance between leading/ trailing edge and meridional inlet/outlet by a) moving leading/ trailing edge in meridional contour if edge is not fixed on inlet/outlet or b) activating CFD-Extension in CFD setup/ Extension (only for impellers).</td>
</tr>
<tr>
<td>Finishing</td>
<td><strong>Beware: &quot;Solid&quot; and &quot;Solid faces&quot; are handled differently in various target systems.</strong></td>
</tr>
<tr>
<td></td>
<td>To be taken into account if a mixed selection of solids and solid faces was selected in the 3D model tree.</td>
</tr>
<tr>
<td>Blades</td>
<td><strong>Blades with thickness definition &quot;perpendicular to mean surface&quot; are not supported.</strong></td>
</tr>
<tr>
<td></td>
<td>See message. Choose another blade thickness definition method (see thickness definition in Blade Profiles).</td>
</tr>
<tr>
<td>Volute</td>
<td><strong>&quot;Flow domain&quot; export may not work.</strong></td>
</tr>
<tr>
<td></td>
<td>The STEP export of &quot;Flow domain.Solid&quot; or &quot;Flow domain.Solid faces.Spiral&quot; might be defective if the spiral face spans a wrap angle of 360°. This occurs for internal volutes. Select &quot;Spiral.Surface&quot; instead in the 3D model tree.</td>
</tr>
<tr>
<td>Model settings</td>
<td><strong>Model geometry does not fit a cube of size (-500, -500, -500) to (500, 500, 500). Choose another export unit to scale geometry appropriately.</strong></td>
</tr>
<tr>
<td></td>
<td>A geometry can be correctly represented only if it is fully included in a cube between the points (-500,-500,-500) and (500,500,500) due to a Parasolid™ library limitation. Change length unit in export parameters dialog for selected export interface.</td>
</tr>
<tr>
<td>Problem</td>
<td>Possible solutions</td>
</tr>
<tr>
<td>------------------------------------------------------------------------</td>
<td>--------------------------------------------------------</td>
</tr>
<tr>
<td>Current point export settings may cause problems in Autodesk Inventor</td>
<td>Change number of points in <strong>Model settings/Point export</strong>.</td>
</tr>
<tr>
<td>due to high number of points.</td>
<td></td>
</tr>
<tr>
<td>See message.</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>General</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Completion of all design steps required.</td>
<td></td>
</tr>
<tr>
<td>Only for CFD-Applications. One or more design steps were not finished.</td>
<td>Complete all design steps.</td>
</tr>
<tr>
<td>Special license for this interface required.</td>
<td></td>
</tr>
<tr>
<td>License for this interface not found.</td>
<td>Check the license information in <strong>SETTNGS/ Licensing</strong>.</td>
</tr>
<tr>
<td>Export not possible due to license restrictions.</td>
<td></td>
</tr>
<tr>
<td>The corresponding module is not licensed or CFturbo is running with a</td>
<td>Only designs corresponding with licensed modules or unmodified default examples using a trial license can be exported.</td>
</tr>
<tr>
<td>trial license.</td>
<td></td>
</tr>
<tr>
<td>Material domain is not available</td>
<td></td>
</tr>
<tr>
<td>Components without material domain are not supported by this interface.</td>
<td>The material domain is created with</td>
</tr>
<tr>
<td></td>
<td>• <strong>Hub/Shroud materials</strong> designed in the Meridian design step</td>
</tr>
<tr>
<td></td>
<td>• Blade solids</td>
</tr>
<tr>
<td></td>
<td>For unvaned stators, the material domain cannot be created because hub/shroud solids are not connected by the blade solid.</td>
</tr>
<tr>
<td>Model state contains no geometry for export.</td>
<td></td>
</tr>
<tr>
<td>See message.</td>
<td>Select via &quot;Set parameters&quot; a proper model state containing desired parts to be exported.</td>
</tr>
<tr>
<td></td>
<td>Imports can only be exported via the context menu of the 3D model tree.</td>
</tr>
<tr>
<td>Meshing process ran out of memory.</td>
<td></td>
</tr>
<tr>
<td>Problem</td>
<td>Possible solutions</td>
</tr>
<tr>
<td>------------------------------------------------------------------------</td>
<td>--------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>A too fine-grained mesh was produced. This resulted in exceeding virtual memory.</td>
<td>Adjust the triangulation parameters, especially minimum and maximum element length. Or if you are using the 32 bit version of CFturbo change to 64 bit.</td>
</tr>
<tr>
<td><strong>Invalid viscosity value.</strong></td>
<td>Set a valid viscosity value in fluid manager.</td>
</tr>
<tr>
<td>See message.</td>
<td></td>
</tr>
<tr>
<td><strong>Some objects didn’t get meshed properly: [Face names]</strong></td>
<td></td>
</tr>
<tr>
<td>The exported mesh is not watertight due to some invalid areas inside the mesh. These areas are equivalent to the specified faces in the warning message.</td>
<td>Check the specified faces in the 3D-model due to invalid geometry. Vary the meshing parameters.</td>
</tr>
<tr>
<td>The exported mesh has overlapping triangles. In this case, the number of overlapping triangles is specified in the warning message. Usually, there are just a few triangles overlapping.</td>
<td>Vary the meshing parameters (finer as well as coarser parameters might solve the overlapping). If it can't be resolved through varying meshing parameters, then attention should be paid on the generation of the volumetric mesh in CFD-applications. Mesh irregularities might be handled during the pre-processing, too.</td>
</tr>
</tbody>
</table>

**5.2.2.1.1 Remarks**

**Virtual and real geometry**

Some export interfaces allow the export of real and virtual geometry (if available). Depending on the selected option, some messages will be displayed for the respective components in the export's component tree.
Export limitations

- Rental or Permanent license
  When using CFturbo with a normal license (rental or permanent) the export is **not restricted** in any way.

- Demo/ Trial license
  Export functionality can **be restricted** when using CFturbo with a Demo/ Trial license. Data export can then be disabled for all individually designed components. To demonstrate the performance of the export interfaces, the data export is enabled for CFturbo default examples only. These default examples can be found in the CFturbo installation directory, sub-directory **Examples**.

5.2.2.1.2 Basic

Under **Basic** the basic export interfaces are grouped which are available independently of the component type.
Export interfaces

- **Basic**
  - General geometry
  - CFturbo exchange
  - Design report
  - **DXF**
  - Tetrahedral volume mesh
  - **Performance data**

> **CAD, CAM**

> **CFD**

Export preconditions

The Basic export interfaces are available always.

[I = Impeller]  [S = Stator]  [V = Volute]  [MC = Multi-Component export supported]

<table>
<thead>
<tr>
<th>Menu item</th>
<th>Description</th>
<th>Component type</th>
</tr>
</thead>
<tbody>
<tr>
<td>General geometry</td>
<td>*.geo-txt</td>
<td>plain text file or XML file</td>
</tr>
<tr>
<td></td>
<td>*.geo-xml</td>
<td></td>
</tr>
</tbody>
</table>

File contains detailed geometry data of the design for any further processing.

**Impeller/stator:**
- Meridional section:
  - z, r of hub, shroud, leading edge
- Blade mean lines, Blade profiles:
  - x, y, z: cartesian coordinates
  - r: radius
  - t: angle
  - T: tangential length
  - m: meridional radius based length
  - m/m_{TE}: meridional radius based length (0..1)
  - M: meridional absolute length
  - M/M_{TE}: meridional absolute length (0..1)
  - β: blade angle
  - s: blade thickness
  - L: 3D length
  - la: lean angle

**Volute:**
<table>
<thead>
<tr>
<th>File Type</th>
<th>Description</th>
<th>Formats</th>
<th>File Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFturbo exchange</td>
<td>* .cft-geo XML file</td>
<td>I S V MC</td>
<td>File contains geometry data that can be re-imported in CFturbo (component data exchange format) see Add component</td>
</tr>
<tr>
<td>Design report</td>
<td>* .html, * .rtf, * .csv, * .txt design report</td>
<td>I S V MC</td>
<td>Design information as text file; Summary of most important design parameters see Report</td>
</tr>
<tr>
<td>DXF</td>
<td>* .dxf neutral format (Drawing Interchange File Format)</td>
<td>I S V MC</td>
<td>File contains designed geometry of the selected component as 3D polylines.</td>
</tr>
<tr>
<td>Tetrahedral volume mesh</td>
<td>* .msh, * .vol, polyMesh available file formats: Fluent, Netgen, OpenFOAM, Abaqus</td>
<td>I S V MC</td>
<td>File or folder contains designed geometry as tetrahedral volume mesh for simulation.</td>
</tr>
<tr>
<td>Performance data</td>
<td>* .cft-pp XML file</td>
<td>I S V MC</td>
<td>File contains results of Performance prediction. Two curves are exported: Δp versus m and η_{Sl} versus ?. Units are those in accordance to preferences.</td>
</tr>
</tbody>
</table>

The project is always exported in its entirety.
Some parameters are available in the **Settings** area to influence the quality / resolution of the STL geometry.

**STL write mode**: Format (Binary / ASCII) for writing STL files.

**Minimum element length**: Minimum mesh element length.

**Maximum element length**: Maximum mesh element length.

**Granularity**: Policy of mesh element construction. 5 levels from very coarse to very fine are available.

### Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solution</th>
</tr>
</thead>
<tbody>
<tr>
<td>Model state contains no surfaces and no solids.</td>
<td>See message.</td>
</tr>
<tr>
<td>STL files describe triangulated surfaces. Points and curves cannot be</td>
<td>STL files describe triangulated surfaces. Points and curves cannot be represented. Select via &quot;Set parameters&quot; a proper model state containing surfaces or solids to be exported.</td>
</tr>
<tr>
<td>represented. Select via &quot;Set parameters&quot; a proper model state containing surfaces or solids to be exported.</td>
<td></td>
</tr>
</tbody>
</table>
5.2.2.1.2.2  Tetrahedral volume mesh

In addition to the settings for triangulation, the following settings are available:

<table>
<thead>
<tr>
<th>Setting</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Format</td>
<td>OpenFOAM</td>
</tr>
<tr>
<td>Domain type</td>
<td>Flow domain (virtual)</td>
</tr>
<tr>
<td>Periodicity</td>
<td>Full 360°</td>
</tr>
<tr>
<td>Min. element length</td>
<td>0.2 mm</td>
</tr>
<tr>
<td>Max. element length</td>
<td>4 mm</td>
</tr>
<tr>
<td>Granularity</td>
<td>Moderate</td>
</tr>
</tbody>
</table>

**Flow vs. Material domain**

The user can select between:

- Flow domain: for CFD purposes, flow domain watertight surfaces are exported
- Material domain: for FEM purposes, material domain watertight surfaces are exported

**Export Format**

Four export formats can be selected:

**Fluent**: *.msh file is exported

**OpenFOAM**: necessary *.gz files and directory structure are exported

**Netgen**: *.vol file is exported
Abaqus: *.inp file is exported

5.2.2.1.3 CAD, CAM

? PROJECT | Export | CAD, CAM

The CAD, CAM group contains the supported CAD, CAM product interfaces.

Export interfaces

- Basic
- CAD, CAM
  - STEP, IGES, STL, Parasolid, ...
  - Ansys BladeGen
  - Ansys DesignModeler
  - Ansys SpaceClaim
  - AutoCAD
  - CATIA
  - Creo Parametric
  - hyperMILL
  - inventor
  - NX
  - SOLIDWORKS
  - ZW3D
- CFD

Export preconditions

The export availability of CAD, CAM interfaces depends on component type and design progress.

<table>
<thead>
<tr>
<th>Component type</th>
<th>Export available from design step</th>
</tr>
</thead>
<tbody>
<tr>
<td>Impeller, stator with blades</td>
<td>&quot;Mean lines&quot;</td>
</tr>
<tr>
<td>Stator without blades</td>
<td>&quot;Meridional contour&quot;</td>
</tr>
</tbody>
</table>
### Menu entry | Description | Component type
---|---|---
STEP, IGES, STL, Parasolid... | *.stp; *.igs; *.stl; *.x_t; *.x_b; *.brep | I S V MC
Ansys DesignModeler | *.stp; *.x_t; *.x_b; | I S V MC
ZW3D | *.stp; *.x_t; *.x_b | I S V MC
AutoCAD | *.txt | I S V MC
Ansys BladeGen | *.rtzt | I S V MC
CATIA | *.catvbs | I S V MC
Creo Parametric | *.ibl, *.pts | I S V MC
hyperMILL | *.stp | I S V MC

[ I = Impeller  S = Stator  V = Volute  MC = Multi-Component export supported ]
<table>
<thead>
<tr>
<th>Software</th>
<th>File Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inventor</td>
<td>*.bas</td>
<td>The macro generates a surface model + generating splines. &lt;br&gt;• Tools</td>
</tr>
<tr>
<td>NX</td>
<td>*.dat</td>
<td>Some files per component are created. &lt;br&gt;• New</td>
</tr>
<tr>
<td>SOLIDWORKS</td>
<td>*.swb</td>
<td>The macro generates a surface model + generating splines. &lt;br&gt;• Tools</td>
</tr>
<tr>
<td>Ansys SpaceClaim</td>
<td>*.stp; *.x_t; *.x_b</td>
<td>File contains designed geometry as volume model. &lt;br&gt;• File</td>
</tr>
</tbody>
</table>
5.2.2.1.3.1  AutoCAD (Autodesk)

The data import from CFturbo is realized by a LISP-script.

**Loading the LISP-Application and Import of the Geometry**

- Tools | Load Application (command: _appload)
- Select file "xyz2spline.lsp" from CFturbo-installation directory, load and close dialog
- Execute loaded LISP-application by command *xyz2spline*
- Select and open *.txt file exported from CFturbo

Attention: If "; Error: Bad argument type: FILE nil" occurs as error message it can be bypassed by typing the filename in the open-file-dialog manually instead of selecting the file by mouse click.
Selection of xyz2spline.lsp file

Construction of Impeller

Creating the blades

• Use the command _loft to create surfaces from curves
Sample-view after data import
Creating rotational surfaces (Hub, Shroud)

- Command `_revolve`
- Select hub and shroud curves
  - *Specify axis start point or define axis by [Object/X/Y/Z] <Object>:* 0,0,0
  - *Specify axis endpoint:* 0,0,1
  - Specify angle of revolution or [STart angle/Reverse/EXpression] <360>: 360
Hub and Shroud surfaces

Construction of Volute
Creating the open part of volute geometry

1. Command _loft

2. Select profile-curves to loft (part by part, starting with the open one)

3. Enter an option [Guides/Path/Cross-sections only] <Cross-sections only>: cross-sections only
4. Repeat steps 1 to 4 for remaining parts of the volute

5.2.2.1.3.2 Ansys BladeGen (Ansys)

Geometry can be exported in two alternative formats:

- RTZT (Camber line data file)
- BGI (Batch input file)

Export format can be selected under Settings:
If the exported RTZT-File is expected to be imported in Ansys BladeGen, enabling cut-off of rounded edges is recommended, since blade edges shape must be specified using Blade Property Dialog within Ansys BladeGen. Therefore, the shape of the blade edges designed in CFturbo and the ones created in Ansys BladeGen could differ substantially depending on these settings. Besides that, importing rounded blades could fail in Ansys BladeGen.

If the exported RTZT-File is expected to be re-imported in CFturbo, disabling cut-off of rounded edges is recommended, since whole thickness distribution including rounded edges can be imported in CFturbo matching the blade shape exactly.

**Ansys BladeGen import**

In Ansys BladeGen, by choosing Ellipse type and setting a ratio value for leading or trailing edges, Ansys BladeGen will always apply the elliptic shape to the blade camberline/ thickness definition. This will override any thickness distribution imported for the region of the ellipse.
### Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solution</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Blades with asymmetric thickness distribution not supported.</strong></td>
<td></td>
</tr>
<tr>
<td>Blades with asymmetric thickness distribution will be imported in Ansys BladeGen in such a way that the thickness distribution is symmetric with respect to the camberline.</td>
<td></td>
</tr>
<tr>
<td><strong>Edge shape should be specified in Ansys BladeGen.</strong></td>
<td></td>
</tr>
<tr>
<td>Ansys BladeGen settings will override any blade edges thickness distribution imported from CFturbo (more information above).</td>
<td></td>
</tr>
</tbody>
</table>

5.2.2.1.3.3 CATIA (Dassault Systèmes)

The data-import is realized by a macro that is created for each geometry individually by CFturbo. The macro is loaded and executed in Inventor.

Open the macro dialog

- **Tools | Makro | Makros or <Alt> + <F8>**
- Select an existing macro library
or

- Create a new macro library: <Makrobibliotheken…>, add directory which contains the macro files created in CFturbo (<Vorhandene Bibliothek hinzufügen…>)

5.2.2.1.3.4 Creo Parametric (PTC)

The following files are exported by CFturbo for impellers:

- *-hub.ibl, *-shroud.ibl: points of hub and shroud
- *-profile.ibl: points for blade profiles
- *.ibl: all points for hub, shroud and blades

The following files are exported by CFturbo for volutes:

- *-contour-line.ibl: spiral contour points
- *-section-closed.ibl: points for all spiral, cut-water and closed diffuser sections
- *-section-open.ibl: points for all open diffuser sections

**Import of curves**

1. Home | New | Part
2. Model | Get Data | Import. Select *.pts or *.ibl file
3. In "File" dialog, select desired import options

4. Confirm to finish import process
All curves can be imported in this way
Creating revolution surfaces

1. Model | Datum | Axis: create axis of revolution selecting the two proper datum planes. (Note: use Ctrl for multi-selection)
2. Model | Datum | Sketch: create a new sketch

- Select the plane containing the curve to be revolved. Reference and orientation items are set automatically after selection.
- Sketch | Sketching | Project: do a projection of the curve selecting the curve. Select option "Single" and click on "Close"
Finalize sketching task by clicking on "OK"
3. Select the curve and click on Model | Shapes | Revolve
4. Click on field "Axis" under tab "Placements" and select the revolution axis. Surface of revolution will be generated.
5. Finalize revolve task by clicking on "OK"
Creating lofted surfaces

Lofted surfaces are created from blade profiles and spiral section curves.

1. Model | Surface | Boundary Blend
2. Select desired curves (use Ctrl for multi-selection)
3. Finalize Boundary Blend task by clicking on "OK"
5.2.2.1.3.5 Inventor (Autodesk)

The data-import is realized by a macro that is created for each geometry individually by CFturbo. The macro is loaded and executed in Inventor.

To execute a macro it has to be imported into an existing VBA-project.

- Tools | VBA Editor
- Open file-open-dialog by File | Import File... and select *.bas macro-file, possibly a new project has to be created File | New Project

- Execute imported macro: Run / Run Macro (F5) close dialog by Run
• The time for executing depends on the complexity of the geometry.
**Troubleshooting**

- Selecting the maximal number of points for one or all components in Model settings/Point export could cause too large exported files and "Out of memory" error message while importing in Inventor:

  To avoid this problem, reduce the selected number of points.

5.2.2.1.3.6 Ansys SpaceClaim (Ansys)

There are 2 alternative methods how to transfer the geometry from CFturbo to Ansys SpaceClaim:

1. **Using the "CAD, CAM/ Ansys SpaceClaim" export interface**

When using the Ansys SpaceClaim interface, the following files are exported:

- Separate STEP files *.stp for each selected component

The STEP files contain solid bodies for the CFturbo flow or material domain, which can be selected under "Parameters" in the CFturbo export dialog. STEP files can be added to the Ansys SpaceClaim project by "Assembly/ File" and will be displayed in the "Structure" panel:

- One Python script *.py
This script file is used to extract the names of the single solid parts ("Named Selections") from the STEP files because these names are not detected automatically while opening in Ansys SpaceClaim.

To run the script:
1) Open the "Script Editor" by "File/ New/ Script"
2) Open and run script:

Hint: It's more easy to select the script file together with the STEP files (see above)

The detected names are visible in Ansys SpaceClaim on the panel "Groups". Ansys SpaceClaim and Ansys have a full associativity. Therefore, the created groups will be available as named selections in Ansys Meshing and Ansys CFX.

Click on a specific group to display the objects that are held in this group.

The described procedure is carried out automatically when the option "Open exported files" is selected in the CFturbo export dialog.
2. Using the "Basic/STEP" export interface

When using the generic STEP export interface all parts currently displayed in the CFturbo 3D view are exported.

Names are visible in Ansys SpaceClaim only if solid faces are selected in the CFturbo 3D view. The disadvantage in this case is that the Ansys SpaceClaim model contains only surfaces and no solid bodies.

Following settings should be checked under Ansys SpaceClaim Options in order to allow the recognition of names for each imported surface:
When importing the geometry files and the Python script together, there is a chance that SpaceClaim will automatically activate the “record” function in the script editor. It is strongly advised to check and deactivate this feature, to ensure the proper function of the pre-built Python
Depending on the complexity of the geometry, SpaceClaim may try to simplify the geometry imports. Any simplification will prevent the Python script from functioning properly. It is therefore advised to deactivate the “Clean and simplify geometry” option in the SpaceClaim Options under the “General” entry in the “Import options” section.
• Under certain circumstances SpaceClaim is not able to import a geometry component properly. This will result in missing faces and the geometry component in being unclosed. This is a known problem in SpaceClaim, Defect Number: 174865. As a Result, the affected body will appear transparent.

When running the Python script, it will detect the problematic body, mark it red and ask for a repair. If the repair suggestion is confirmed, the script tries to create a closed volume body by rebuilding the missing faces. Since the original geometry information are partially lost, a creation of named selections for the affected body will not be possible. After the repair attempt the body will be colored green while repaired faces will be marked red.
Under certain circumstances some parts of the geometry are missing when forwarding the geometry from SpaceClaim to Ansys meshing. The critical area is mainly the splitter edge of double volutes. In SpaceClaim the geometry looks fine. This is a known problem in SpaceClaim since version 19.2, Defect Number: 130684.

As workaround, you can create a new Windows environment variable with the name ANS_READER_HEAL and set the value to TRUE.

5.2.2.1.4 CFD

The CFD group contains the supported CFD product interfaces.
Export preconditions

The export availability of CFD interfaces depends on component type and design progress.

<table>
<thead>
<tr>
<th>Component type</th>
<th>Export available from design step</th>
</tr>
</thead>
<tbody>
<tr>
<td>Impeller, stator with blades</td>
<td>&quot;Blade edges&quot;</td>
</tr>
<tr>
<td>Stator without blades</td>
<td>&quot;Meridional contour&quot;</td>
</tr>
<tr>
<td>Volute</td>
<td>&quot;Diffuser geometry&quot;</td>
</tr>
</tbody>
</table>

The interfaces ANSA, Ansys ICEM-CFD, Ansys Meshing, Ansys TurboGrid, OMNIS/AutoGrid, OpenFOAM, Pointwise, Simerics, Simcenter Star-CCM+ and TCFD support multi-component export.

\[ I = \text{Impeller} \quad S = \text{Stator} \quad V = \text{Volute} \quad MC = \text{Multi-Component export supported} \]

<table>
<thead>
<tr>
<th>Menu entry</th>
<th>Description</th>
<th>Component type</th>
</tr>
</thead>
<tbody>
<tr>
<td>ANSA</td>
<td>*.igs</td>
<td>I S V MC</td>
</tr>
<tr>
<td></td>
<td>• File</td>
<td>Open</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Select *.igs file</td>
</tr>
<tr>
<td>Ansys Meshing</td>
<td>*.stp; *.x.t; *.x_b</td>
<td>I S V MC</td>
</tr>
<tr>
<td></td>
<td>Ansys Workbench:</td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Add &quot;Mesh&quot; container to &quot;Project Schematic&quot;</td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Right click on &quot;Geometry&quot; inside the &quot;Mesh&quot; container:</td>
<td></td>
</tr>
<tr>
<td></td>
<td>select &quot;Import geometry&quot;</td>
<td></td>
</tr>
<tr>
<td>OMNIS/AutoGrid</td>
<td>*.geomTurbo</td>
<td>I S V MC</td>
</tr>
<tr>
<td></td>
<td>• File</td>
<td>New Project</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• &quot;Initialize a New Project from a geomTurbo File&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Select *.geomTurbo file</td>
</tr>
<tr>
<td>Omnis/Hexpress</td>
<td>*.stp</td>
<td>I S V MC</td>
</tr>
<tr>
<td></td>
<td>File contains designed geometry as volume.</td>
<td></td>
</tr>
<tr>
<td>Ansys ICEM-CFD</td>
<td>*.tinXML, *.stp</td>
<td>I S V MC</td>
</tr>
<tr>
<td></td>
<td>A STEP file with named geometries is created. The names</td>
<td></td>
</tr>
<tr>
<td></td>
<td>are visible in ICEM-CFD if the file is imported via Workbench Reader.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Parameters are saved in a separate XML file.</td>
<td></td>
</tr>
<tr>
<td>IGG</td>
<td>*.dat</td>
<td>I S V MC</td>
</tr>
</tbody>
</table>
### Multiple data files are generated: section.dat, diffusor.dat, curves.dat
- **File | Import | IGG Data**
- **Select *.dat file**
- **Repeat steps for remaining files**

### Pointwise
- ***.step**
- **File | Import | Database**
- **Select *.step file**

### Simerics
- ***.spro, *.stl**
  - The *.spro file contains all project information. The *.stl files contain the geometry in STL format as triangulated surfaces.
  - In SimericsMP/ SimericsMP+: Select *.spro file under File | Open

### Simcenter STAR-CCM+
- ***.stp**
  - **File | Import | Import Surface Mesh...**
  - **Select *.stp file**

### Ansys TurboGrid
- ***.tse or *.inf, *.curve**
  - 2 alternative formats are available:
    a) Session file *.tse (not available within Ansys Workbench)
      - 4 files are created: a session file <filename>.tse and 3 geometry files <filename>_hub.curve, <filename>_shroud.curve, <filename>_profile.curve.
      - Load the session file <filename>.tse under Session | Play Session.
    b) Initialization file *.inf
      - 4 files are created: a initialization file <filename>.inf and 3 geometry files <filename>_hub.curve, <filename>_shroud.curve, <filename>_profile.curve.
      - Load the initialization file <filename>.inf under File | Load TurboGrid Init File.
  - Alternatively you can open the curve files (<filename>_hub.curve, <filename>_shroud.curve, <filename>_profile.curve) manually under File | Load Profile Points. Beside the selection of the *.curve files one has to specify the number of blades, define the z axis as rotational axis, select the cartesian coordinate system and the length unit specified in CFturbo for export

### TCFD
- ***.tcfd, *.stl**
The *.tcfd file contains all the CFD project information. The *.stl files contain the geometry in STL format as triangulated 3D surfaces. The .tcfd file is read and the simulation is performed automatically.

In TCFD run command:
$ CFDProcessor -setup fan.tcfd -allrun &

More info at CFD support website

Ansys Vista TF
- *.fil, *.con, *.geo, *.aer, *.cor

5 files are created:
- default file <filename>.fil
- control data file <filename>.con
- geometry data file <filename>.geo
- aerodynamic data file <filename>.aer
- correlation data file <filename>.cor

Run compiled executable version of the Vista TF code. Exported files need to be in the same folder than the executable file.

GridPro
- *.stp; *.stl

File contains designed geometry as volume model.

Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flow inlet does not match previous outlet.</td>
<td>If possible, activate the RSI connection in CFD setup of the impeller.</td>
</tr>
</tbody>
</table>

- Impeller - Impeller
- Impeller - Stator
- Impeller - Volute
- Stator - Impeller
- Stator - Stator
- Stator - Volute

Inlet geometry was defined in such a way that it does not match the inlet of the previous component. See geometric coupling for more details.
Problem | Possible solutions
--- | ---
Flow outlet does not match next inlet. | Outlet geometry was defined in such a way that it does not match the outlet of the next component. See geometric coupling for more details.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Stator - Stator</td>
<td></td>
</tr>
<tr>
<td>Stator - Volute</td>
<td></td>
</tr>
</tbody>
</table>

5.2.2.1.4.1 Ansys Meshing (Ansys)

Ansys Meshing can be used within Ansys Workbench only. Using the CFturbo Workbench extension is recommended to create smooth workflows very comfortable.

For manual use of Ansys Meshing the following prerequisite needs to be fulfilled in order to use pre-defined names for the geometry parts:

- For Ansys Workbench versions older than 2022R1, the Parasolid file format should be selected in CFturbo instead of STEP, because Ansys Meshing is not able to detect the names from the STEP file in those versions.

The export format can be selected in the Settings area.
In Ansys Workbench, the usage of **Named Selections** must be enabled manually:
- menu **Tools/ Options**
- select **Geometry Import** bottom left
- scroll down to **Basic Options** on the right
- Activate **Named Selections** option
- set **Filtering Prefixes** to **NS**

5.2.2.1.4.2 OMNIS/AutoGrid (Cadence)

The geometry data for impeller is exported by CFturbo to „geomTurbo“-files which can be loaded by AutoGrid.

- Start IGG
- Change to AutoGrid5-mode: **Modules / AutoGrid5**
• Open a new project: File | New Project

• Close dialog by Initialize a New Project from a geomTurbo File
• If the model have more than one vaned component, add so many rows as additional vaned components

• Select *.geomTurbo-file
For unshrouded impellers the tip clearance has to be applied in AutoGrid manually.

5.2.2.1.4.3 Ansys ICEM-CFD (Ansys)

This interface supports the script solution CFturbo2ICEM, a script for automatic geometry generation and meshing of CFturbo components. Therefore, it should be used only in combination with CFturbo2ICEM. Detailed information can be found on the CFturbo website.

Two files are exported: a *.tinXML file containing all meshing parameters specified in CFturbo and a *.stp file containing the designed geometry with specific naming conventions.
For more information about using CFturbo2ICEM please see the available documentation.

5.2.2.1.4.4 Simerics (Simerics)

**Geometry settings**
In addition to the STL settings, the user can select which files should be exported.

**Export all files:** Configuration file (*.spro) and STL files are exported.

**Export configuration file only:** STL files are not exported. This option can be useful for saving export time if the user wants to generate a new configuration file with different settings (e.g. mesh parameters, rotational speed, boundary condition values, fluid data etc.). In this case, all geometrical export requirements (like solid trimming) are disabled in the CFturbo Export window.

**Export STL files only:** The configuration file is not exported. This option is useful, e.g. if STL files for some (but not all) components have to be exported again due to an unsatisfactory triangulation. In this case, the original configuration file, which refers to all components, should not be overwritten.

**Mesh settings**
Mesh settings are specified globally for all selected CFturbo components to be exported.

The following global mesh parameters are available:

- Min. cell size
- Max. cell size
- Cell size on surfaces

If the user wants to set specific parameters for predefined regions, the option “Use local cell size on surfaces” must be activated.

If not, the global value “Cell size on surfaces” is used for all regions. Specific mesh parameters can be set for the following regions:

<table>
<thead>
<tr>
<th>Rotational symmetric components</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hub</td>
<td>-</td>
</tr>
<tr>
<td>Shroud</td>
<td>-</td>
</tr>
<tr>
<td>Blade sides</td>
<td>Blades suction and pressure sides</td>
</tr>
<tr>
<td>Blades LE / TE / Tip</td>
<td>Blades edges leading, trailing and tip</td>
</tr>
<tr>
<td>Secondary flow path hub</td>
<td>Hub material domain surfaces</td>
</tr>
<tr>
<td>Secondary flow path shroud</td>
<td>Shroud material domain surfaces</td>
</tr>
<tr>
<td>Inlet</td>
<td>-</td>
</tr>
</tbody>
</table>
Outlet

For RSI-Connection surfaces, the “Hub” parameters are used for meshing:

Secondary flow path mesh parameters are useful if a fine meshing is necessary in gap zones between solid bodies and casing.
Enabling the option “Refine casing”, local cell sizes set for hub and shroud are also used for the casing surfaces.
For RSI-Connection surfaces, the “Spiral” parameters are used for meshing:
In case the volute extension is exported, “Diffuser” mesh parameters are used for the extension walls.
“Inlet” and “Outlet” mesh parameters are used for the extension inlet and outlet:

### Refinement zones

In some cases mesh refinement zones are more effective than applying local mesh sizes. This might be true e.g. for secondary flow path simulations where very small gaps have to be meshed. The images below show a typical case:
Mesh resulting from setting local mesh parameters on hub and shroud as well as casing surfaces.

Mesh resulting from using refinement zones in isolated regions.
The use of refinement zones allows for high quality refinement in regions with small gaps, avoiding high numbers of cells. Refinement zones can be applied within the whole geometry and can be set by defining the refinement cell size and cylindrical refinement area:

**Solver settings**

The following parameters are available:
- **Solver** selection (SimericsMP or SimericsMP+)
- Selection of **simulation type**: Steady or transient
- **Number of iterations**
- **Result saving frequency** (only for transient simulations)

For transient simulations, two special parameters are available. These parameters differ from the Simerics original ones, but are more comfortable for turbomachinery:

- **Number of revolutions**: number of impeller rotations to be simulated. The original Simerics parameter “Simulation Time (Duration)” is calculated using this new parameter and the rotational speed of the impeller. Default value is 3 revolutions.
- **Rotation angle per step**: number of degrees the mesh is rotated by per step. The original Simerics parameter “Number of time steps” is calculated using this new parameter and the number of revolutions to be simulated. Default value is 3 degrees.

- A global value for **converge criterion** can be set. This value is used for all active modules. In case the default Simerics value (0.001 for steady simulations and 0.1 for transient simulations) is used, values are not written in the SPRO-file allowing Simerics to change the values automatically if the user switches interactively in the Simerics-GUI between steady and transient simulation.
- **Numeric scheme** and **Relaxation** values
- **Cavitation** (activated by default for pumps). Not available for compressible flow (compressors, gas turbines, fans)
- **Reversible pump turbine**: enabling this option, adapted boundary conditions are used to simulate the pump as a turbine.
- **Turbomachinery expressions** are useful to check the simulation convergence. IMPORTANT: Plotting of user defined expressions is not supported for Simerics versions lower than 4.2.8 and can cause problems in these versions.

**Not supported characters**

The character "_" is not allowed for component names for the time being.
In addition to the Triangulation settings, the following parameters are available:

**Files parameters**

 Allows to define which files should be exported.

**Export all files:** Configuration file (*.tcfd) and STL files are exported.
Export configuration file only: STL files are not exported. This option can be useful for saving export time if the user wants to generate a new configuration file with different settings (rotational speed, boundary condition values, fluid data etc.). In this case, all geometrical export requirements (like solid trimming) are disabled in the CFturbo Export window.

Export STL files only: The configuration file is not exported. This option is useful, e.g. if STL files for some (but not all) components have to be exported again due to an unsatisfactory triangulation. In this case, the original configuration file, which refers to all components, should not be overwritten.

Full 360° or periodic segment

Allows to define whether the full geometry or only a periodic segment should be exported for selected rotational symmetric components.

Not supported characters

Following characters are not permitted: 'space' < > [ ] { } ( ) \ " : ? ! * & ^ % $ # @ ¦ °

5.2.2.1.4.6 Ansys TurboGrid (Ansys)

Troubleshooting

- Surfaces can be described in Ansys TurboGrid by two different options: "Ruled" (linear) or "B-Spline".
  More than 4 sections could result in an oscillating surface if the curves are not located exactly on the surface.

To avoid the problem you should select the Surface Type 'Ruled' under 'Blade Set' in the Ansys TurboGrid object tree.
• For open impellers and stators, a small region between leading/trailing edge and meridional inlet/outlet could result in the following error message while importing in Ansys TurboGrid:
  "Error extending the shroud tip line. Try reducing the “Tip expansion factor” value."

Two options are available to increase this region:

  a) moving the leading/trailing edge in meridional contour. The edge has not to be fixed on inlet/outlet. This option incurs a geometrical modification
  b) activating a CFD extension at inlet (for radial or mixed flow turbine impellers) or outlet (for the rest of impellers) in CFD setup/Extension. This option does not incur a geometrical modification of the component but of the neighboring one if exists.

5.2.2.2 Batch mode/ Optimization

This feature can be used to prepare batch mode runs of CFturbo for systematic parameter variations (DoE) or configure optimization jobs, see Batch mode.

Load, Save

The working directory is displayed on top of the dialog.

Complete configurations as batch mode files in XML format (*.cft-batch) can be loaded by Load ... and saved by Save or Save as... .

The input CFturbo file is the currently opened project always, whose directory is specified in relative notation by default. A copy of the current CFturbo project is stored in the destination directory of the batch mode file (*.cft-batch).

If CFturbo is running inside Ansys Workbench, the batch mode configuration is saved automatically.

More information about CFturbo inside Ansys workbench is available on the CFturbo website:
https://cfturbo.com/software/interfaces-workflows/extension-for-ansys-workbench

Parameters

All available parameters of the project are displayed in a tree structure according to the components and design steps.

File name of modified project [not available inside Ansys Workbench]
If parameters of the CFturbo project will be modified manually or by an optimization software during the batch run, the **File name of modified project** (*.cft) can be specified at the top of the page. Saving this file can be blocked by defining an empty file name.

Each parameter can be activated or deactivated individually.

For activated parameters the range can be specified optionally either by

- a small dialog (click on the button on right side of the range cell)
or

- using the following syntax:

<table>
<thead>
<tr>
<th>Resolution</th>
<th>Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>Continuous</td>
<td>&lt;Min. value&gt;:&lt;Max. value&gt;</td>
</tr>
<tr>
<td>Discrete</td>
<td>&lt;Min. value&gt;:&lt;Max. value&gt;:&lt;Count&gt; or &lt;Value1&gt;,&lt;Value2&gt;,&lt;Value3&gt;...</td>
</tr>
</tbody>
</table>

If a discrete range definition was specified a list of the discrete values is displayed as hint when moving the mouse over the cell.

**Export actions**

On this page the export actions for the batch run can be specified. Multiple export actions can be defined by selecting the export formats and the corresponding components on the right side (very similar to the Export dialog) and pressing the Add button.
Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solution</th>
</tr>
</thead>
<tbody>
<tr>
<td>Export requirements are not fulfilled.</td>
<td></td>
</tr>
<tr>
<td>Some of selected components no longer fulfill the export requirements.</td>
<td>Remove or replace the affected export actions.</td>
</tr>
<tr>
<td>Component selection already used.</td>
<td></td>
</tr>
<tr>
<td>Applicable only in <strong>Ansys Workbench</strong> mode. At least one selected component was already used for the same export interface in previous export actions. The critical export item is highlighted in red.</td>
<td>Remove affected export actions.</td>
</tr>
</tbody>
</table>

5.2.2.3 Reference components

This functionality can be used for simultaneous display of various designs to compare each other and for purposeful modification.
Using the **Add**-button any reference project (*.CFT-file) can be added. All components of the reference project are grouped under the selected file name.

Each component has its own color and line width (panel **Options**). Multiple components can be selected using <Shift> and <Ctrl> keys. Clicking on the group header area selects all components of the corresponding project, <Ctrl> <A> selects all components.

With the **Remove**-button the selected reference project with all its components can be deleted from the list. However single reference components may be deactivated by the check box at the beginning of the line.

**Display in dialogs**

Reference geometries are displayed in the dialogs with selected color and line width. Numerical values appear as small hints on input fields when mouse is moved over it.
Down right in the design step dialog windows you could completely switch off the display of reference geometries and start the configuration dialog.

Please note: If you add reference designs in a design step dialog the imported geometry could be invisible initially if it's far away from the currently designed geometry. There is no automatic scaling of the diagram.
Display in 3D-model

Reference geometry is displayed as 3D model additionally.

All reference geometries are arranged in the model tree in the region “Reference designs”, whereas the single parts can be configured like the normal geometry.
5.2.2.4 Model finishing

Model finishing is started globally for all components of the project. Each vaned component has its own model finishing configuration, see Model finishing for impeller.

This operation could be time-consuming because the model finishing of a single component takes 10-50 seconds.
5.3  SETTINGS

This menu is used for specifying some general program settings:

- Licensing
- Preferences
- Approximation functions
- Fluids
- Profiles

5.3.1  Licensing

?  SETTINGS | General | Licensing

See General/ Licensing

5.3.2  Preferences

?  SETTINGS | Preferences

This menu item is used for global program options. Alternatively it can be accessed by the File menu.

The dialog contains several topics on the left side and the corresponding settings on the right side.
There are 3 main groups of topics available on the left side:

- **General**
- **Units**
- **Impeller/ Stator**

5.3.2.1 General

Topic **GENERAL** is used for global program options.
Language of online help

In this dialog the language of online help can be set. The default is English.

Warning before license expiration
Furthermore you can specify the number of days for license expiration warning at startup. Default value is 20 days.

The warning message looks as follows:

Initial view after loading file

Select which view should be displayed after file loading. Choosing the 3D Model will increase the time needed for loading, because the model gets updated first.

Reset "Additional Views" configuration

Deletes the configuration of "Additional Views" of all dialogs. The configuration contains the visibility as well as width and height of the visible elements.

Ask for deactivating automatic calculations when loading older file

If a CFturbo project was created by an older version and contains automatic calculations the user will be asked for deactivating it when opening such a file. This should assure identical geometry over several CFturbo versions. See Automatic calculations.
3D model mouse handling

Here you can assign functions (Rotate, Zoom, Move) to the mouse buttons (Left, Middle, Right) for handling the 3D model.

Action when double-clicking component

The default action for double-clicking on a component in the component list can be set. This enables the user to quickly switch to the menu needed.
Check for available updates

Optionally, you can check for available updates at program startup. 3 alternative intervals are available: at each start, weekly, monthly.

An update check can be started directly using the button “Check now...” (see Check for Updates). The date of last update check is displayed for information.

5.3.2.2 Units

? SETTINGS | Preferences | Units

Unit settings can be used for selecting the display units in CFturbo.

It’s divided in 3 parts:

→ Basic units: general unit selection

→ Specific speed: selecting a suitable specific speed definition


5.3.2.2.1 Basic units

Here the physical units used in the dialogs can be set.

Following grouped units are available:

<table>
<thead>
<tr>
<th>Category</th>
<th>Unit</th>
<th>Symbol</th>
<th>available</th>
</tr>
</thead>
<tbody>
<tr>
<td>Geometry</td>
<td>Length</td>
<td>L</td>
<td>mm, in, m</td>
</tr>
<tr>
<td></td>
<td>Area</td>
<td>A</td>
<td>mm², m², in²</td>
</tr>
<tr>
<td></td>
<td>Volume</td>
<td>V</td>
<td>mm³, m³, in³</td>
</tr>
<tr>
<td></td>
<td>Angle</td>
<td>δ</td>
<td>°, - (radian)</td>
</tr>
<tr>
<td>Energy</td>
<td>Specific energy, Enthalpy</td>
<td>Y, h</td>
<td>m²/s², J/kg, kJ/kg, Nm/kg, Ws/kg, ft²/s², BTU/lb</td>
</tr>
<tr>
<td>------------------------------</td>
<td>---------------------------</td>
<td>-------------</td>
<td>-----------------------------------------------</td>
</tr>
<tr>
<td>Head</td>
<td>H</td>
<td>m, ft</td>
<td></td>
</tr>
<tr>
<td>Pressure</td>
<td>p</td>
<td>MPa, PSI, bar, mbar, Pa, kPa, mm H20, mm Hg, in H20, ft H20</td>
<td></td>
</tr>
<tr>
<td>Stress</td>
<td>τ</td>
<td>MPa, Pa, PSI</td>
<td></td>
</tr>
<tr>
<td>Temperature</td>
<td>T</td>
<td>°C, K, °F, °R</td>
<td></td>
</tr>
<tr>
<td>Power</td>
<td>P</td>
<td>W, kW, hp</td>
<td></td>
</tr>
<tr>
<td>Torque</td>
<td>T</td>
<td>Nm, lbft, lbin</td>
<td></td>
</tr>
<tr>
<td>Force</td>
<td>F</td>
<td>N, kN, lbf</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Flow</th>
<th>Volume flow</th>
<th>Q</th>
<th>m³/h, m³/min, m³/s, l/min, l/s, ft³/min, ft³/s, gpm, gps</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass flow</td>
<td>m</td>
<td>kg/s, lb/s</td>
<td></td>
</tr>
<tr>
<td>Velocity</td>
<td>v</td>
<td>m/s, ft/s</td>
<td></td>
</tr>
<tr>
<td>Swirl</td>
<td>v·r</td>
<td>m²/s, ft²/s, in²/s</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Fluid</th>
<th>Density</th>
<th>ρ</th>
<th>kg/m³, lb/ft³</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dynamic viscosity</td>
<td>μ</td>
<td>Pa·s, cP</td>
<td></td>
</tr>
<tr>
<td>Kinematic viscosity</td>
<td>ν</td>
<td>m²/s, ft²/s</td>
<td></td>
</tr>
<tr>
<td>Heat capacity</td>
<td>cp</td>
<td>J/(kg·K), BTU/(lb·°F)</td>
<td></td>
</tr>
<tr>
<td>Molar weight</td>
<td>M</td>
<td>kg/mol</td>
<td></td>
</tr>
<tr>
<td>Thermal conductivity</td>
<td>λ</td>
<td>W/(m·K)</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Other</th>
<th>Ratio</th>
<th>x/y</th>
<th>%, - (absolute)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Revolutions</td>
<td>n</td>
<td>/min, /s</td>
<td></td>
</tr>
</tbody>
</table>
5.3.2.2.2 Specific speed

Here the specific speed definition can be selected. This definition is mainly used for the Approximation functions.

The definitions mainly differ in the units used for rotational speed, flow rate and energy transmission.

Following definitions are available:
- General specific speed $n_q^*$ (dimensionless)
  $$n_q^* = n \frac{Q^{1/2}}{Y^{3/4}}$$

- Type number $\omega_s$ (dimensionless)
  $$\omega_s = n_s = 2\pi n \frac{Q^{1/2}}{Y^{3/4}}$$

- Speed coefficient $\sigma$
  $$\sigma = \frac{\varphi^{1/2}}{\psi^{3/4}} = 2.11 \cdot n \frac{Q^{1/2}}{Y^{3/4}}$$

- European definition $n_q^E$
  $$n_q^E = n_{[\text{min}^{-1}]} \frac{Q[m^3/s]^{1/2}}{H[m]^{3/4}}$$

- US definition $N_s$
  $$N_s = n_{[\text{rpm}]} \frac{Q[gpm]^{1/2}}{H[ft]^{3/4}}$$

- Asian definition $n_q^A$
  $$n_q^A = n_{[\text{min}^{-1}]} \frac{Q[m^3/min]^{1/2}}{H[m]^{3/4}}$$

Furthermore it’s possible to select an alternatively specific speed definition using the separate units for Revolutions, Flow rate and Head.

On the bottom side some information for the currently selected specific speed definition is displayed. The **Factor on dimensionless value** is the factor used to convert the General specific speed $n_q^*$ to the currently selected definition. Furthermore the **Typical range** of the specific speed definition for centrifugal, mixed-flow and axial machines is displayed in the table.
Here some additional unit settings can be selected.

**Blade/flow angle $\alpha$, $\beta$**

Angles measured against circumferential direction

(internal angles of the velocity triangle)

allowed range: $0^\circ \ldots 180^\circ$
Suction specific speed

There are 3 alternative possibilities to define the suction specific speed for pumps:

- **SI definition (dimensionless)** $n_{ss}^*$
  \[ n_{ss}^* = n \frac{Q^{1/2}}{\left(\gamma \cdot NPSH\right)^{3/4}} \]

- **European definition** $n_{ss}$
  \[ n_{ss} = n_{\text{min}} \left( \frac{Q}{NPSH[n]} \right)^{2/3} \]

- **US definition** $N_{ss}$
  \[ N_{ss} = n_{\text{rpm}} \left( \frac{Q}{NPSH[ft]} \right)^{2/4} \]

### 5.3.2.3 Impeller/ Stator

**? SETTINGS | Preferences | Impeller/ Stator**

These settings define properties related to stator and impeller design.

It's divided in 3 parts:

- **Progression diagrams** x-axis definition of the progression diagrams in design step windows
- **Initial default settings** used by default when creating a new impeller
- **Warning level** define recommended range and warning level for some important values
5.3.2.3.1 Progression diagrams

At **Progression diagrams** one can specify, which parameter should be used for the x-axis of the progression diagrams in the Meridional contour and Blading windows (Meanline, Profiles, Edges, 3D cross section).

- abs. meridional length \( M \)
- rel. meridional length \( M/M_{\text{Max}} \)
- abs. radius based meridional length \( m \)
- rel. radius based meridional length \( m/m_{\text{Max}} \)
- abs. radius \( r \)
- rel. radius \( r/r_{\text{Max}} \)
- abs. axial length \( z \)
- rel. axial \( \Delta z/\Delta z_{\text{Max}} \)
- abs. chord length \( l \)
- rel. chord length \( l/l_{\text{Max}} \)
Some constellations may yield undefined x-values due to reference (e.g. $r_{\text{Max}}$, $\Delta z_{\text{Max}}$) values. Those constellations will be marked in the diagrams. One should use another option in such a case.

5.3.2.3.2 Initial default settings

At Initial default settings one can select which settings should be used by default when creating a new impeller. Individual settings can be specified for each machine type (Pump, Fan, Compressor, Turbine), separately for each impeller subtype.

Of course these settings can be modified manually in the design step dialogs if required.
5.3.2.3.3 Warning level

At **Warning level** one can specify recommended ranges and warning levels for some important impeller properties.

**Blade angle β**

**Blade angle at leading edge βB1, trailing edge βB2**

The usual and the recommended range can be specified. The displayed error levels cannot be customized.

Usually, the blade angle values at the **outer blade tip/shroud span** of the blade are checked.

An exception is the outlet side of centrifugal impellers: due to the nearly constant radius the values at all spans are checked.

**Max. βB difference hub-to-shroud**

This max. allowed ΔβB along leading or trailing edge is used to avoid highly twisted blades.
Max. camber angle $\phi$

This max. allowed value for $\phi = \Delta \beta = |\beta_{B2} - \beta_{B1}|$ is used to avoid too high flow deflection on a single span.

The values are used to

- colorize the display of resulting blade angle $\beta_{B2}$ in the main dimensions window of centrifugal impellers
- create warning messages in the Blade properties window/ design step
- colorize the blade angle values $\beta_{B1}$ and $\beta_{B2}$ in the
  - Blade properties window
  - Blade mean line window
  - Blade profile (Airfoil) window

Max. blade blockage

Blade thickness $s$ is blocking a part of the flow passage $u = \pi d / \text{number of blades}$. The blockage factor is calculated as $F = s / u$.

The max. allowed blade blockage factor can be defined.

Blade overlap factor (for centrifugal impellers only)

The overlap factor is defined as $F = \text{wrap angle } \Delta \phi / \text{pitch angle } t$ ($t = 360^\circ / \text{number of blades}$). Min. and max. limits can be specified to avoid too low overlapping (poor flow guidance) and too high overlapping (high flow blockage).

Stagger angle $\gamma$ (for airfoil/ hydrofoil design only)

Min. and max. limits can be specified to avoid unreasonable blade twist.

Slip angle $\delta$

Warning and error level can be specified to avoid high deviation between blade and flow direction at trailing edge.
5.3.3 Approximation functions

CFturbo uses many approximation functions. These functions are based on published measurement data that facilitate the forecast of optimal or accessible values.

In this dialog the approximation functions are displayed graphically and can be customized. If an open project is available then only the project relevant functions are displayed, otherwise all functions are available.

Currently about 150 functions are available for the following individual component types and sub-types:

<table>
<thead>
<tr>
<th>Axial impeller</th>
<th>Centrifugal impeller</th>
<th>Casing</th>
</tr>
</thead>
<tbody>
<tr>
<td>• Axial Compressor Impeller</td>
<td></td>
<td></td>
</tr>
<tr>
<td>o Standard</td>
<td></td>
<td></td>
</tr>
<tr>
<td>• Centrifugal Compressor Impeller</td>
<td></td>
<td></td>
</tr>
<tr>
<td>o Standard</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>• Axial Fan Impeller</th>
</tr>
</thead>
<tbody>
<tr>
<td>o Standard</td>
</tr>
<tr>
<td>o Automotive Cooling</td>
</tr>
<tr>
<td>• Centrifugal Fan Impeller</td>
</tr>
<tr>
<td>o Standard</td>
</tr>
<tr>
<td>o Squirrel cage</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>• Axial Gas Turbine Rotor</th>
</tr>
</thead>
<tbody>
<tr>
<td>o Standard</td>
</tr>
<tr>
<td>o Rocket Engine</td>
</tr>
<tr>
<td>• Radial Gas Turbine Rotor</td>
</tr>
<tr>
<td>o Standard</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>• Axial Pump Impeller</th>
</tr>
</thead>
<tbody>
<tr>
<td>o Standard</td>
</tr>
<tr>
<td>o Inducer</td>
</tr>
<tr>
<td>• Centrifugal Pump Impeller</td>
</tr>
<tr>
<td>o Standard</td>
</tr>
<tr>
<td>o Wastewater</td>
</tr>
<tr>
<td>o Barske (low nq)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>• Francis Turbine Runner</th>
</tr>
</thead>
<tbody>
<tr>
<td>o Standard</td>
</tr>
</tbody>
</table>
Each function has a hard coded default function. For each of these functions custom point wise defined curves can be added alternatively. These custom defined curves are saved in the file `Functions.cftfu` that contains the custom defined functions only. The default functions are not saved in any external file and cannot be deleted. The default functions can only be deactivated by defining any custom function that is saved in the Functions file.

On the top left at **File location**, the name of the file is shown that contains all user-defined functions. In general this file is called `Functions.cftfu`, and is located in the installation directory of CFturbo. Modifications to functions are saved automatically if you leave the dialog window by pressing the **OK**-button. In case the user has no write permissions one could choose a different directory to save the file. Changing filename and directory is possible by using the **Save as**-function. By clicking the **Open**-button a previously saved functions file can be opened.

The link to the functions file is part of each major/minor installation (CFturbo x.y). All updates by bug-fix releases (CFturbo x.y.z) do not modify the link to the existing function file.

The function file will not be overwritten by any update. By default the functions file is located in the CFturbo installation directory. When you define any user-defined functions it’s recommended to save the functions file not in the CFturbo installation directory but anywhere in the company network for two reasons:

- all users can use the same database for their design
- there is no risk of losing data by uninstall older versions of CFturbo
All available functions are listed in a tree structure in the panel **Functions** left from diagram, sorted by machine type.

The user must first select the variable under the corresponding machine type. CFturbo’s internal function is displayed in the diagram in blue color. You can add any user defined function for each variable. Selected function is displayed in the diagram in addition to CFturbo's internal function. Function with active check box is used by CFturbo for calculations. If no function has active checkbox or no additional function is defined at all, then the CFturbo internal function is used.

With these buttons below the tree you can add, delete or rename functions. Alternatively you can use the context menu by right click on any function.

The following hierarchy exist in the tree:

- **Mechanical efficiency**
  - **Test**
    - n=1000
      - ▲ Upper Limit ▲
    - n=2000
      - ▲ Upper Limit ▲

Functions can depend on 2 variables whereas one serves as parameter. Separate curves exist for each particular parameter value that are used to calculate function values. The parameter value is displayed on endpoint of the curve in the diagram.

With the upper limit curve you can define a recommended range, which means an area that is defined by a higher and a lower limit.

In panel **Points** right from diagram you can edit curve points of selected function. You can add new points at the end of the table – the points are automatically sorted by x values. To remove a point you have to delete either x or y value.

These buttons are enabling the user to:

- import points from file (one point per line)
- export points to file
- copy all points to clipboard
- paste points from clipboard (e.g from Excel)
- clear the table
On panel **Test** you can test the active function. Saving of values is possible by clicking **OK**-button.

### 5.3.4 Fluids

** SETTINGS | Database | Fluids  

The dialog lists all defined fluids. New ones can be added, present fluids can be altered, renamed or deleted.
In the right panel, the properties of the selected fluid can be defined. The available parameter vary depending on the medium type (compressible/incompressible).

The buttons for opening and saving offer the possibility of the exchange of fluid data between CFturbo installations.

New fluids can be added by copying and changing values of existing fluids using the plus button. Another way is to use fluids defined in the CoolProp library. To this end the button has to be used.

In the third column it is specified by whether a CoolProp definition of the fluid is available. This applies only for compressible fluids.

**Incompressible fluid** [for pumps, fans only]

Parameters are:
- density \( \rho \)
- kinematic viscosity \( \nu \)
- thermal conductivity \( \lambda \)
- heat capacity \( c_p \)
- vapor pressure \( p_v \) (only for liquids)

**Compressible fluid** [for compressors, turbines only]

Here some gas properties are required because they are used in the gas models for the descriptions of the behavior of the gases. Those parameters are:
- gas constant \( R \)
- heat capacity \( c_p \) (perfect gas)
- molar weight \( M \) alternatively for \( R \)

- critical pressure \( p_{\text{crit}} \), temperature \( T_{\text{crit}} \) and density \( \rho_{\text{crit}} \)
- acentric factor \( \omega \)
heat capacity coefficients $c_p$ (at zero pressure)

- compressibility factor $Z$

- kinematic viscosity $\nu$

- thermal conductivity $\lambda$

Currently the following gas models are implemented. They represent a relation between pressure, temperature and density (here given with its reciprocal the spec. volume $v$):

<table>
<thead>
<tr>
<th>Gas model</th>
<th>Approach</th>
<th>Annotation</th>
<th>Reference (first published)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Perfect Gas</td>
<td>$p = \frac{R \cdot T \cdot Z}{v}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Redlich-Kwong</td>
<td>$p = \frac{R \cdot T}{v-b+c} - \frac{a(T)}{v(v+b)}$</td>
<td>Each approach has its own set of coefficients $a$, $b$ and $c$.</td>
<td>Redlich, O., Kwong, J.N.S.</td>
</tr>
<tr>
<td>Aungier/Redlich-Kwong</td>
<td>$p = \frac{R \cdot T}{v-b+c} - \frac{a(T)}{v^2-2vb+b}$</td>
<td></td>
<td>Aungier, R.H.</td>
</tr>
<tr>
<td>Soave/Redlich-Kwong</td>
<td>$p = \frac{R \cdot T}{v-b+c} - \frac{a(T)}{v^2-(2v+b)}$</td>
<td></td>
<td>Soave, G.</td>
</tr>
<tr>
<td>Peng-Robinson</td>
<td>$p = \frac{R \cdot T}{v-b+c} - \frac{a(T)}{v^2-2vb+b}$</td>
<td></td>
<td>Peng, D.Y., Robinson, D.B.</td>
</tr>
<tr>
<td>CoolProp</td>
<td>see reference</td>
<td>Currently available for fluid creation only but not within the project</td>
<td><a href="http://www.CoolProp.org">www.CoolProp.org</a></td>
</tr>
</tbody>
</table>

The implemented gas property models can be tested with user defined data. Those data consists of a thermodynamic state defined by $p_1$ and $T_1$. Using these values the density $\rho_1$ and the specific heat $c_p$ will be calculated. The latter is calculated from the following approach at a pressure close to zero:

$$c_p(T) = \sum_{i=0}^{3} c_{pi} \cdot T^i.$$

Also, using a pressure $p_2$, the gas shall be compressed or expanded to an isentropic temperature $T_{2is}$ will be calculated. A second temperature $T_2$ is calculated under the assumption that the gas shall be compressed or expanded from state 1 to pressure $p_2$ with an efficiency of $\eta$. The according enthalpy and entropy differences $\Delta h$ and $\Delta s$ resp. is given too, see h-s-diagram.
### Test of Air

#### Input values
- **$p_1$**: 1 bar
- **$p_2$**: 2 bar
- **$T_1$**: 25.0 °C
- **η**: 90%

#### Properties
- **Density**: $\rho_1 = f(p_1, T_1)$, 1.1688 kg/m³
- **Specific heat**: $c_p = f(p_1, T_1)$, 1006.3 J/(kg·K)
- **Isentropic temperature**: $T_{2is} = f(p_1, p_2, T_1)$, 90.2 °C
- **Isentropic enthalpy diff.**: $\Delta h_{is} = f(p_1, p_2, T_1)$, 65594 m²/s²
- **Enthalpy difference**: $\Delta h = f(p_1, p_2, T_1, \eta)$, 72882 m²/s²
- **Temperature**: $T_2 = f(p_1, p_2, T_1, \eta)$, 97.4 °C
- **Polytropic efficiency total-total**: $\eta_{pol} = f(p_1, p_2, T_1, T_2)$, 90.856
- **Sat. temperature**: $T_s = f(p_1)$, -191.54 °C
- **Sat. pressure**: $p_s = f(T_1)$, 0 bar

---

#### Graphs
- **$\Delta h$ [m²/s²]**
- **$\Delta s$ [J/(kg·K)]**
5.3.4.1 CoolProp library

If a fluid from the CoolProp library is to be added the following dialogs will appear:

**Incompressible fluid** [for pumps, fans only]

The constant data needed for the design process will be derived at a temperature T1. Three different sets of fluids are available: pure fluids, water mixtures and 2 phase fluids. Water mixtures need the specification of the mass fraction of the added fluid. That means $x = 0$ is pure water, whereas $x = 1$ is a mixture without water.
For 2 phase fluids the liquid state shall be used, i.e. \( p_1 > p_{v\text{ triple}} \) and \( T_{\text{triple}} < T_1 < T_{\text{crit}} \). Triple point temperature as well as critical temperature are used as range limits for the vapor pressure curve, see picture below.
Compressible fluid [for compressors, turbines only]

Constant data as well as pressure and temperature dependent properties are displayed in a table. The heat capacity coefficients $c_p$ (at zero pressure) are derived at temperatures $T_1$ and $T_2$. A least square fitting algorithm is used for this purpose.
The gaseous state exists in a certain scope, i.e. $p_1 < p_v$ and $p_1 < p_{\text{triple}}$ and $T_{\text{triple}} < T_1$. Triple point temperature as well as critical temperature are used as range limits for the vapor pressure curve, see picture below. Additionally the heat capacity at zero pressure is given as function of $T_1$ and $T_2$ both based on CoolProp as well as on the polynomial with the coefficients $c_{pi}$. 

© CFturbo GmbH
5.3.4.2 Gas mixtures

Compressible fluid [for compressors, gas turbines only]

New gas mixtures can be designed on the basis of gases already defined in the compressible branch of the fluid manager. The mass and mole ratios resp. of the components will be used to determine the mixture's properties. The generated mixture can be tested with the calculate button in the upper right corner.
All mixture parameters apart from kinematic viscosity and thermal conductivity are calculated with the help of the component's mass fraction $w_i$ and parameter $f_i$:

$$f_{\text{mix}} = \sum w_i f_i$$

Mixture kinematic viscosity and thermal conductivity are determined by the component's viscosity and conductivity respectively weighted by the component's mole fraction $x_i$ and by a correction factor $\beta_{ij}(T)$. Mole fraction and mass fraction are connected by the molar weight $M_i$:

$$\sum x_i M_i = \frac{1}{\sum w_i M_i}$$

The correction factor $\beta_{ij}(T)$ is determined from the component's kinematic viscosity $v_i$ and molar weight $M_i$ according to Mason & Saxena by:

The kinematic viscosity is determined by:
\[ v(T) = \sum_i \frac{x_i \cdot v_i(T)}{\sum_j x_j \cdot \phi_j(T)} \]

The thermal conductivity is determined by:

\[ \lambda(T) = \sum_i \frac{x_i \cdot \lambda_i(T)}{\sum_j x_j \cdot \phi_j(T)} \]

### 5.3.5 Profiles

? SETTINGS | Database | Profiles 🌐

The dialog lists all defined profiles. New ones can be added, present profiles can be renamed, deleted and changed.

In the right panels, the properties of the selected profile can be defined. The available parameter vary depending on the profile type.

The buttons for opening and saving offer the possibility of the exchange of profile data between CFturbo installations.

**NACA 4 Digit**
The NACA 4 Digit wing sections are low cambered profiles. This family of profiles allows a separate modification of camber and thickness, which is especially advantageous for blade design.

The profile are defined by:

- First digit describing maximum camber as percentage of the chord.
- Second digit describing the distance of maximum camber from the airfoil leading edge in tens of percents of the chord.
- Last two digits describing maximum thickness of the airfoil as percent of the chord.

The thickness distribution is given by:

In case the profile is not cambered, its center of gravity is located at \( x/l = 42.04 \% \). This is the default position for the axial positioning in blade position and sweep.

The meanline consists of two parabola arcs, whose transition point is their apex, respectively. The point is defined by the the first two digits.
\[ y_S = \frac{1}{X_T} \left[ \frac{X_T}{I} - \left( \frac{X}{I} \right)^2 \right] \quad \text{if} \quad \frac{X}{I} \leq \frac{X_T}{I} \]

\[ y_S = \frac{1}{X_T} \left[ 1 - 2 \frac{X_T}{I} + 2 \frac{X_T}{I} \left( \frac{X}{I} \right)^2 \right] \quad \text{if} \quad \frac{X}{I} > \frac{X_T}{I} \]

In addition to the geometric properties lift coefficients and glide numbers need to be set with respect to the angle of attack.

**NACA 65 series**

The NACA 65 series is of importance for turbo-machinery because of their systematic cascade studies. In contrast to NACA 4 digit, their aerodynamic data is also known for more heavy cambered profiles.
The meanline can be calculated from a theoretical lift coefficient that is calculated from a user-defined camber angle, see Carolus p. 54, (Eq. 3.11, 3.12):

\[ c_{ll} = \frac{2\pi}{\ln(2)} \cdot \tan\left(\frac{\varphi}{4}\right) \text{ mit } \varphi = \beta_2 - \beta_1 \]

\[ y_\infty = -\frac{c_{ll}}{4\pi} \left[ \left(1 - \frac{x}{l}\right) \ln \left(1 - \frac{x}{l}\right) + \frac{x}{l} \ln \left(\frac{x}{l}\right) \right] \]

The nose radius can be modified, thickness values have to be changed in the table. One can type in a camber angle that will result in a cambered profile. This profile can be promoted to Point-based profiles by pressing convert. Please note that the skeleton design is done in the appropriate design step. Therefore the camber angle is not part of the profile description and is given here for informational and conversion reasons.

**Point-based**

![Profile interface](image)

© CFturbo GmbH
Besides NACA profiles also user-defined profiles are provided. Therefore the lower and upper side of the profile has to be known. Moreover lift coefficients and glide numbers need to be set with respect to the angle of attack.

Circular profiles are constructed by 2 circular arcs, 1 linear piece and a nose with a certain nose radius. They are described by 3 parameters: profile angle $\beta$, nose radius $r_N/l$ and a radius offset. The construction details are displayed in the sketch below:
5.4 HELP

This menu supports the user on how to use CFturbo.

The following features are available:

- **Help topics** General CFturbo online help, including help index
- **About CFturbo** Information about CFturbo (e.g. version information)
- **Check for updates** Check for updates online
- **Web Help** Show CFturbo help in a web-browser
- **Show tutorials** Show online tutorials for CFturbo
What's new
List of main new features in the current version

CFTurbo website
Open CFTurbo website in browser

5.4.1 Check for Updates

Here you can check for available updates on the CFTurbo website. Most of all this concerns the frequently released maintenance versions mainly provided for bug fixing.

The currently running version is displayed as well as the latest available for download. If an updated version is available a direct link to the download website is displayed. The download access (name + password) remains valid as long as a maintenance contract is running (time limited rental licenses include maintenance for the whole leasing period - there is no separate maintenance contract required).

Update check can be executed automatically. This can be configured in the Preferences.
5.5 Component

These context sensitive menu is used for designing the currently selected component.

A separate tab with the corresponding design steps is available for each component type:

<table>
<thead>
<tr>
<th>Component</th>
<th>Design Mode</th>
</tr>
</thead>
<tbody>
<tr>
<td>IMPELLER</td>
<td>Mean line design mode</td>
</tr>
<tr>
<td>STATOR</td>
<td>Airfoil/ Hydrofoil design mode</td>
</tr>
<tr>
<td>VOLUTE</td>
<td></td>
</tr>
</tbody>
</table>

Menu items and buttons only become active in accordance to the current design state. Each finished design steps can be opened again whereas all depending design steps and components are updated automatically. Manual removing of complete component’s design steps is possible in order to continue with CFturbo initial design (see Remove design steps).

For designing the complete geometry of a single component you have to run through all items of the appropriate menu step by step.
Alternatively all these menu items can be selected in the Meridian view using the toolbar directly on the selected component (see Meridian).

5.6 3D View

This context sensitive menu is used for handling of the 3D model. It becomes visible if the 3D Model view is currently selected. Detailed description can be found in Views/ 3D Model.

2 tabs are available:

3D MODEL

BLADES

This Menu is used for handling geometries with blades (impeller, vaned stator) in the 3D model. Because a project can contain multiple geometries with blades, these settings refer to the currently selected component.
### 5.7 Report View

This context sensitive menu is used for handling the project report.

<table>
<thead>
<tr>
<th>Report View</th>
<th>Component</th>
</tr>
</thead>
<tbody>
<tr>
<td>Datei</td>
<td>PROJECT</td>
</tr>
<tr>
<td>SETTINGS</td>
<td>HELP</td>
</tr>
<tr>
<td>STATOR</td>
<td>REPORT</td>
</tr>
</tbody>
</table>

- Save report
- Print report
- Copy to Clipboard
- General
- Components (MIN)
- Design steps
- Design steps + 1
- All information (MAX)
- Visible

Detailed description can be found in [Views/ Report](#).
Part VI
CFturbo offers 3 alternative views on the project in the central part of the main window. The view can be selected by the buttons underneath the ribbons.

- **Meridian**
  
  The diagram with the meridional view of the components gives an overview of the project and enables quick access to the components and the interfaces in between.

- **3D Model**
  
  Shows the whole project as a 3D model.

- **Report**
  
  Presents a tabular view on the project information and the parameters of the components down to design step level.

---

© CFturbo GmbH
6.1 Meridian

This view consists mainly of a diagram containing the meridional shape of all components.

Active components are displayed with their respective color, inactive components are displayed grey.

**Meridional diagram**

The diagram depicts the assembled meridional shapes of the project components and their connecting interfaces. A large arrow on the inlet of the first component illustrate the flow direction. Captions showing component name and a consecutive number are displayed as well.

The currently selected component is displayed with thick border and can be changed by mouse click on a component.
Component toolbar

If the mouse moves over the selected component the components menu is shown in compact style.

Alternatively you can use the corresponding ribbon menu (see IMPELLER/ STATOR/ VOLUTE).

Component context menu

Right clicking on the component opens its context menu, see Activate/ Rename/ Delete components.

Adding Components

Via the symbol an additional component can be added to the project at the symbols position. A menu shows the available component types and the option to import an existing one.

See Add component.
Geometric coupling

The direction of the geometric coupling between components is displayed by small symbols (see left). The coupling can be changed by moving the mouse over a coupling symbol and selecting a coupling configuration from the appearing menu.

Progression diagram

Below the meridional view, progressions of several physical quantities along the flow direction of all components can be displayed:
A  Cross section area

$c_m$  Meridional velocity

$c_u$  Circumferential velocity

c  Absolute velocity

$\alpha$  Flow angle

### 6.2  3D Model

Tab sheet **3D Model** contains the three dimensional representation of the project design state. This view has its own context sensitive ribbon tab, see **3D View** tab.

The CAD model can be exported as IGES, STEP, STL, Parasolid or BREP - see **Export** tab. For export, only the currently visible geometrical elements are considered.
Navigation

The 3D display can be influenced by **mouse**:

<table>
<thead>
<tr>
<th><strong>Rotate</strong></th>
<th>Rotation around center of visualized geometry or clicked point on a 3D-object respectively</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="3D marker" /></td>
<td>The rotation center is visualized by a 3D marker.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Zoom</strong></th>
<th>↓ Zoom (also mouse wheel) ↔ Rotation around z-axis</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th><strong>Move</strong></th>
<th>Move</th>
</tr>
</thead>
</table>

The functions can be assigned to mouse buttons via Preferences/ General.

Furthermore, the 3D-View is sensitive to mouse movement and mouse clicks in the following manner:

<table>
<thead>
<tr>
<th>Mouse movement</th>
<th>Activates highlighting of 3D-object under the cursor. When hovering on the 3D-object, a hint with its name is displayed.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Left mouse-button</td>
<td>Selects and deselects the 3D-object under the cursor, respectively. When clicking in empty space, all 3D-objects are deselected.</td>
</tr>
<tr>
<td>&lt;Crtl&gt; + left mouse-button</td>
<td>Multi-selection of 3D-objects.</td>
</tr>
<tr>
<td>Right mouse-button</td>
<td>Opens context menu with display properties for all selected 3D-objects (see Model tree (left)).</td>
</tr>
</tbody>
</table>

Menus

**Above** the 3D representation in the menus **3D Model** and **3D Model - Blades** you can find buttons which have only an optical effect but do not change the geometry model.

- Model display (top)

**Model tree**

**Left** of the 3D representation is the **Model tree**. There, all available geometry parts are listed in a tree structure, whereby they can be configured individually.
3D-Preview

In many design step dialogs a 3D-Preview of the currently designed part can be displayed via the Additional views button at the top.

The 3D-Preview behaves in the same way as the 3D Model view described above. For performance reasons, the 3D objects are displayed with medium resolution, at most.

See also:
- Problems when generating surfaces/solids
- Open/ Save design
- Data export

6.2.1 Model display (top)

3D MODEL

The following actions are available by the buttons of the 3D Model tab. They are used for visualization only and do not affect the geometry model.

General

- Copy representation to clipboard
- Print representation
- Save representation as PNG, JPG, GIF or BMP
Save representation as 3D-PDF

Visible 3D-model data is transferred to a PDF-document, enabling the user to distribute or archive a 3D-representation of CFturbo-models.

The PDF-document contains a 3D-viewer, including a model-tree as well as various 3D-capabilities (e.g. measuring, clipping, display options). It can be opened with PDF-viewers supporting 3D-content (e.g. Adobe® Acrobat® Reader, not possible in internet-browsers without certain plugins).

If the PDF-document is opened in a PDF-viewer and a static picture is shown instead of a 3D-model, then:

1) ensure using a PDF-viewer supporting 3D-content
2) ensure activating 3D-content in the settings of the PDF-viewer
3) click on the static picture to switch to the 3D-content

An example of a 3D-PDF visualized in Adobe® Acrobat® Reader DC is shown below:

Import external 3D geometry for visualization

The 3D Import enables the user to view 3D data in IGES, STEP, STL, Parasolid and BREP format or of CFturbo-projects (*.cft) e.g. for comparison with the current design or for redesigning. Geometry data is shown in the 3D Model and can be transformed and exported.

If the import consumes a lot of time, a lower resolution can be selected (see "Settings" below).
**View**

<table>
<thead>
<tr>
<th>Icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Fit view" /></td>
<td>Fit view (zoom all geometry to visible region)</td>
</tr>
<tr>
<td><img src="image" alt="Viewing direction" /></td>
<td>Viewing direction in positive or negative (&lt;0&gt;) x-axis direction</td>
</tr>
<tr>
<td><img src="image" alt="Viewing direction" /></td>
<td>Viewing direction in positive or negative (&lt;0&gt;) y-axis direction</td>
</tr>
<tr>
<td><img src="image" alt="Viewing direction" /></td>
<td>Viewing direction in positive or negative (&lt;0&gt;) z-axis direction</td>
</tr>
<tr>
<td><img src="image" alt="Reset view" /></td>
<td>Reset view (default position)</td>
</tr>
<tr>
<td><img src="image" alt="Load view" /></td>
<td>Load view from file</td>
</tr>
<tr>
<td><img src="image" alt="Save current view" /></td>
<td>Save current view to file</td>
</tr>
</tbody>
</table>

**Settings**

<table>
<thead>
<tr>
<th>Icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Switch coordinate system" /></td>
<td>Switch coordinate system on/off</td>
</tr>
<tr>
<td><img src="image" alt="Switch scale system" /></td>
<td>Switch scale system on/off</td>
</tr>
<tr>
<td><img src="image" alt="Set background color" /></td>
<td>Set background color</td>
</tr>
<tr>
<td><img src="image" alt="Uniform rotation of impeller" /></td>
<td>A uniform rotation of the impeller around the z axis can be generated, whereby the velocity can be influenced by the track bar.</td>
</tr>
<tr>
<td><img src="image" alt="Select resolution of curves and surfaces" /></td>
<td>Select resolution of curves and surfaces (affects display)</td>
</tr>
<tr>
<td><img src="image" alt="Coarse" /></td>
<td>Coarse</td>
</tr>
<tr>
<td><img src="image" alt="Middle" /></td>
<td>Middle</td>
</tr>
<tr>
<td><img src="image" alt="Fine" /></td>
<td>Fine</td>
</tr>
<tr>
<td><img src="image" alt="Define line width for points" /></td>
<td>Define line width for points</td>
</tr>
<tr>
<td><img src="image" alt="Define line width for curves" /></td>
<td>Define line width for curves</td>
</tr>
<tr>
<td><img src="image" alt="Set number of surface isocurves" /></td>
<td>Set number of surface isocurves</td>
</tr>
</tbody>
</table>

**Clipping**

A clipping plane for x=const., y=const. or z=const. can be defined and optionally displayed. The position of the clipping plane can be adjusted.
The direction of clipping (visible clipping side) can be switched.

**BLADES**

These actions are used for visualization only and do not affect model geometry.

**Please note:** The following options refer to the currently selected component of the project.

<table>
<thead>
<tr>
<th>Component</th>
<th>3D View</th>
<th>3D View</th>
</tr>
</thead>
<tbody>
<tr>
<td>STATOR</td>
<td>3D MODEL</td>
<td>BLADES</td>
</tr>
</tbody>
</table>

- **Single blade**
<table>
<thead>
<tr>
<th>Blade passage</th>
<th>All blades</th>
</tr>
</thead>
<tbody>
<tr>
<td>Display a single blade passage bordered by 2 neighboring blades.</td>
<td></td>
</tr>
</tbody>
</table>
Display all blades of the selected impeller or vaned stator.

Area

Display an approximately perpendicularly flown through area between hub, shroud and two neighboring blades for the currently selected component. The position of this area can optionally be fixed to the location of the throat area (Fix to minimum). Otherwise, it can be slided to any reasonable position within the blade to blade channel with the help of the track bar Section Position.

By pressing the button Show progression a window is opened, in which the value of the cross section is displayed in dependence on the position (see here for changing position variables) between leading edge and trailing edge. The current position as well as that of the throat area and the maximum sphere diameter are marked with special symbols. In the lower part of the window some measures for the current position are displayed.
Sphere

The sphere represents a particle with the highest possible diameter that can be conveyed through the blade passage.
6.2.2  Model tree (left)

The model tree contains all available geometry parts listed in a tree structure, whereby their visibility can be switched on or off alternatively.

All visible elements are exported, if the model is saved as IGES, STEP, STL, Parasolid or BREP - see Export tab.

Display properties

The elements selected in the model tree are highlighted in the 3D view. The selection can be cleared by pressing the <Esc> key.
The attributes can be defined by right click:

- Display mode: Wireframe, Shaded surface, Shaded surface with edges
- Material
- Color and transparency
- Check/ uncheck all sub-elements of the selected element
- Show geometric properties of selected element in extra dialog:
  - volume, density, mass, center of gravity and static moments of inertia for solids
  - area for surfaces
  - length for edges
- Export selected element as IGES, STEP, STL, Parasolid or BREP.

  Imported geometry will be exported in its transformed state (this option is not available for STL imports)

For elements in the **Imports** section only:

- removes selected element(s) from model tree and 3D view
- renames selected element inside model tree

**Model tree structure**

The model tree has 3 main sections:

1) **Section Components**

contains all components of the project with the following sub elements:

<table>
<thead>
<tr>
<th>Impeller/Stator</th>
<th>Volute</th>
</tr>
</thead>
<tbody>
<tr>
<td>Meridian</td>
<td>Spiral</td>
</tr>
<tr>
<td>Mean surface</td>
<td>Diffuser</td>
</tr>
<tr>
<td>Blade</td>
<td>Cut-water</td>
</tr>
<tr>
<td><strong>CFD setup</strong></td>
<td><strong>CFD setup</strong></td>
</tr>
</tbody>
</table>

If an element contains child elements, it can be expanded by clicking on the collapsed element symbol (_PID_).
Each single element **without** child elements can be selected (✓) or unselected (☐).

Each single element **with** child elements can have 3 states:

1. □ Hub
   - □ Points
   - □ Curve
   - □ Surface
   - The element and all child elements are selected.

2. □ Hub
   - □ Points
   - □ Curve
   - □ Surface
   - The element and not all child elements are selected.

3. □ Hub
   - □ Points
   - □ Curve
   - □ Surface
   - The element is unselected. Child elements might be selected.

An element is **visible** in the 3D view, if it is selected and all its **parent elements are also selected**.

Note: If the <Ctrl> key is pressed while selecting an element, all child elements are selected, too!

2) **Section Geometry**

contains all basic geometrical types:

- Points
- Curves
- Surfaces
- Solid faces
- Solids

This allows:

- to select all objects of a certain geometrical type. In the 3D view, only those elements become visible, whose parent elements are selected also.
- to modify the display properties of all currently visible objects of a certain geometrical type.

3) **Sections Imports and Reference designs**

These sections contain imported geometric models: [imported 3D models from neutral 3D file formats](#) and CFTurbo components of [reference projects](#).
Visibility and render properties for imported models can be modified in the same way as for models of section Components.

Selecting a model tree node of an imported model enables the 3D-transformation of the entire model. For that purpose the panel Transformation is displayed at the bottom side of the model tree. This can be used to align imported models with the project model for visual comparisons of the model shapes.

The Transformation panel allows the application of five different types of geometric transformations:

- **Translations** can be applied iteratively along the coordinate axes.
- **Rotations** can be applied iteratively around the coordinate axes.
- Uniform model **scaling** is applied in absolute (percentage) terms.
- **Mirroring** is toggled for the models coordinate system in all three coordinate directions.
- **Z-Axis alignment** applies to the selection of a single or multiple face(s) containing rotational properties (e.g. cylinder, cone) and aligns the related rotational axis collinear to the global z-axis.

To apply a transformation to the selected parts, select a transformation type, set its parameters and hit <Enter>.

The model transformation can be reset to the state which it was imported with by clicking the reset button on top right.

Useful transformations for an imported model can be saved for later use by exporting the model with its current transformation via the context menu (Export).

**Model states**

Model states contain the properties of all tree elements (except imports). Several model states can be managed via the controls above the model tree.

<table>
<thead>
<tr>
<th>Key</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>Default</td>
<td>Select existing model state</td>
</tr>
<tr>
<td>Ctrl</td>
<td>Save model state</td>
</tr>
<tr>
<td>Tab</td>
<td>Rename selected model state</td>
</tr>
<tr>
<td>+</td>
<td>Add new model state</td>
</tr>
<tr>
<td>–</td>
<td>Delete selected model state</td>
</tr>
</tbody>
</table>
The following predefined model states cannot be modified:

- "Default" The default model state
- "Default + CFD setup" The default model state with CFD setup visible
- "Flow domain solids" Only solids are visible that belong to the flow domain
- "Material domain solids" Only solids are visible that belong to the material domain
- "Component colors" Every component is displayed with the color defined in the Components view

For performance reasons, model states do not contain the state of each individual 3D object, but only to the level of distinction between different geometrical types (points, curves, surfaces). Therefore, e.g. all curves that belong to a "Curves" object share the same properties.

### 6.2.3 Problems when generating the 3D model

#### Information about 3D-Errors

If any errors occur while generating geometrical elements then the corresponding part in the model tree is marked by red color.

Furthermore, a corresponding error message is displayed in the message panel.

#### Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>3D-Error: Could not create solid.</td>
<td></td>
</tr>
<tr>
<td>Distance tolerance is too low or too high</td>
<td>Change the distance tolerance (see Model settings)</td>
</tr>
<tr>
<td>Problem</td>
<td>Possible solutions</td>
</tr>
<tr>
<td>--------------------------------------------------</td>
<td>--------------------------------------------------------</td>
</tr>
<tr>
<td>Number of data points is disadvantageous</td>
<td>Change the number of data points for the 3D model</td>
</tr>
<tr>
<td>(seldom)</td>
<td>(see Model settings)</td>
</tr>
</tbody>
</table>

Eliminating errors during surface generation

For eliminating errors during surface generation there exist the following possibilities:

- try a different number of data points for the 3D model (see Impeller or Volute-Settings)

- try a different display resolution (see Model display (top))

The pictures illustrate the possible influence of point density on the surface generation of the blade.

Surface display errors

It may occur that a surface is not displayed although it exists.
You can recognize such cases by selecting the surface in the model tree and choosing a high number of isocurves (see Model display (top)).

Normally, choosing another resolution (see Model display (top)) solves this problem.

**Slow 3D model**

If the handling of the 3D model is very slow, normally an update of the graphic card driver is helpful.

**Visualization errors**

Visualization errors and artifacts can often be resolved by updating the graphic card driver.

See also: Known problems

### 6.3 Report

The report shows the most important information about the design in a tabular style. This view has its own context sensitive ribbon tab, see Report View.

In the tree, the project information and the global setup parameters are listed prior to the components. Tree elements containing sub elements can be collapsed and expanded.
The buttons of the **Report** tab on the ribbon have the following function:

<table>
<thead>
<tr>
<th>Icon</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Save" /></td>
<td>Save report as HTML, RTF, CSV or TXT</td>
</tr>
<tr>
<td><img src="image" alt="Print" /></td>
<td>Print report</td>
</tr>
<tr>
<td><img src="image" alt="Copy" /></td>
<td>Copy the content to the clipboard. All marked rows are copied. If nothing is marked then all content is copied. Marking can be done by mouse, &lt;Ctrl&gt; &lt;A&gt; marks all. Content will be pasted in MS Word/Excel as table.</td>
</tr>
<tr>
<td><img src="image" alt="Expand" /></td>
<td>Expand nodes to several levels</td>
</tr>
</tbody>
</table>
Part VII
This chapter describes in detail the design process for all impeller type components featured in CFltubo.

The content reflects the design steps in the sequence they are encountered during the design process.

**Design steps**

- **Main dimensions**
- **Meridional contour**
- **Mean line design**
  - **Blade properties**
  - **Blade mean lines**
  - **Blade profiles**
  - **Blade edges**
- **Airfoil/ Hydrofoil design**
  - **Blade properties**
  - **Blade profiles**
  - **Blade sweep**
- **CFD setup**
- **Model settings**
- **Model finishing**
- **Remove design steps**

**Possible warnings**
### Problem

<table>
<thead>
<tr>
<th>Neighboring blades are intersecting each other.</th>
</tr>
</thead>
<tbody>
<tr>
<td>(see message)</td>
</tr>
<tr>
<td>Numerous details of the design influence the blade shape. Some examples for possible solutions:</td>
</tr>
<tr>
<td>• Modify main dimensions</td>
</tr>
<tr>
<td>• Reduce number of blades</td>
</tr>
<tr>
<td>• Reduce blade wrap angle</td>
</tr>
<tr>
<td>• Reduce blade thickness</td>
</tr>
</tbody>
</table>

### 7.1 Main dimensions

The Main Dimensions menu item is used to define main dimensions of the impeller.

**Details by impeller type**

- Centrifugal/ Mixed-flow Pump/ Fan
- Axial Pump / Fan
- Centrifugal Compressor
- Radial-inflow Gas Turbine
- Axial Gas Turbine / Compressor
- Francis Turbine

**Automatic calculation of impeller diameter depending on rotational speed**

The automatic calculations can be easily used to determine the influence of the rotational speed on the impeller diameter, e.g. to find the required rotational speed to account for a certain limited design space.

To do this, both the **Automatic** parameter estimation (page Parameters) and the **Automatic** main
Impeller dimension calculation (page **Dimensions**) must be activated. If you then change the **Alternative speed** on page **Setup**, the resulting impeller diameter can be checked on the **Dimensions** page and/or in the **Meridian** diagram.

**Possible warnings**

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Automated Main dimensions are active.</strong></td>
<td>To fix the main dimensions you could uncheck the “Automatic” calculation. Then you have to manually start the calculation if required.</td>
</tr>
<tr>
<td>Dimensions are not fixed and may adapt to input parameters.</td>
<td></td>
</tr>
<tr>
<td>Main dimensions are updated automatically if any input parameters are modified.</td>
<td></td>
</tr>
<tr>
<td><strong>Specific speed of impeller is beyond the supported range.</strong></td>
<td>Modify specific speed defined by n, Q, Δpt in the impeller <strong>Main dimensions</strong> and/or in the <strong>Global Setup</strong>.</td>
</tr>
<tr>
<td>The specific speed nq of the impeller is much too low or too high. The supported range is nq = 5 ... 500.</td>
<td>In some cases it may be necessary to select a different impeller type according to the specific speed.</td>
</tr>
<tr>
<td><strong>Specific speed of impeller is beyond the recommended range.</strong></td>
<td>Modify specific speed defined by n, Q, Δpt in the impeller <strong>Main dimensions</strong> and/or in the <strong>Global Setup</strong>.</td>
</tr>
<tr>
<td>The specific speed is lower or higher then the recommended values.</td>
<td>In some cases it may be necessary to select a different impeller type according to the specific speed.</td>
</tr>
<tr>
<td>This warning is generated for</td>
<td></td>
</tr>
<tr>
<td>• centrifugal/ mixed-flow impellers with specific speed nq &lt; 10 or nq &gt; 160</td>
<td></td>
</tr>
<tr>
<td>• axial impellers with specific speed nq &lt; 100 or nq &gt; 400</td>
<td></td>
</tr>
<tr>
<td><strong>High pre-swirl numbers are unusual.</strong></td>
<td>Adapt pre-swirl in the <strong>Global setup</strong>.</td>
</tr>
<tr>
<td>The impeller is the first component of the project and the inflow swirl is defined by the <strong>Global setup</strong>. With very high pre-swirl an impeller design might be impossible.</td>
<td></td>
</tr>
</tbody>
</table>
7.1.1 Centrifugal, Mixed-flow Pump/ Fan

The Main Dimensions menu item is used to define main dimensions of the impeller. Main Dimensions are forming the most important basis for all following design steps.

The real flow in an impeller is turbulent and three-dimensional. Secondary flows, separation and reattachment in boundary layers, cavitation, transient recirculation areas and other features may occur. Nevertheless it is useful - and it is common practice in the pump design theory - to simplify the realistic flow applying representative streamlines for the first design approach.

Employing 1D-streamline theory the following cross sections are significant in particular: suction area (index S), just before leading edge (index 0), at the beginning (index 1) and at the end of the blade (index 2) and finally behind the trailing edge (index 3).

Details

- Setup
- Parameters
- Dimensions
### Setup

On page **Setup** you can specify some basic settings.

#### General

- **Manual dimensioning**
  In manual dimensioning mode the main dimensions and blade angles are not calculated by CFturbo. All these values are user-defined input values.

- **Unshrouded**
  Design a shrouded (closed) or unshrouded (open) impeller.
  For an unshrouded impeller you have to define the **tip clearance**, optional different values at inlet and outlet.
• **Splitter blades** *(not for axial machines)*  
  Design impeller with or without splitter blades.

• **Material density**  
The material density of the impeller is an informational value that is not relevant for the hydraulic design but is used for the calculation of moments of inertia. Density values can be directly entered or selected from a list by pressing the settings button next to the input area.

• **Impeller type**  
  Beside Standard impeller type the following special impeller types are available with their specific parameters and default settings:
  - for pumps:
    * **Wastewater** You have to specify the desired number of blades used for some specific empirical correlations.
    * **Barske (low nq)** A flow factor FQ can be specified to use the so called "Enlarged Flow Method" in order to get a higher efficiency. Typical range of FQ is between 1.3 and 1.7.
  - for fans: **Squirrel cage**

• **Inflow swirl**  
  Either the outlet swirl of the upstream component will be used for the determination of the inlet swirl or the absolute inlet flow angle.

**Multi stage**
For a multi stage design the panel Multi stage options is available.

**Initial default setting**

When creating a new design the initial default settings for some important properties are displayed in the panel Initial default settings. These settings are used in further design steps and can be modified by selecting the Change settings button. Of course these default settings can be modified manually in the appropriate design steps. See Preferences: Impeller/ Stator settings for more information.

**Information**

Some design point values are displayed in the right Information panel when selecting the page Values (see Global setup).

**7.1.1.2 Parameters**

On page Parameters you have to put in or to modify parameters resulting from approximation functions in dependence on specific speed \( n_q \) or flow rate \( Q \). Separate functions exist for pumps and fans. Additionally some specific functions for waste water pumps are available. See Approximation functions.
For details of how to handle the parameter edit fields please see Edit fields with empirical functions.

Parameter and efficiency values can be handled manually or can be switched to automatic update by the checkbox on top of the page. Then the default values are used always, even after design point modifications (see Global setup).

If the automatic mode is not selected the current default values can be specified by one of the following options:

- globally by the button on top of the page
- regionally by the default button within the Parameters or Efficiency region
- individually by the default button within the input field when selected
Parameters

The panel **Parameters** allows defining alternative parameters in each case for the calculation of the following impeller main dimensions:

<table>
<thead>
<tr>
<th>for pumps</th>
<th>for fans</th>
</tr>
</thead>
<tbody>
<tr>
<td>suction diameter $d_S$</td>
<td>inlet diameter $d_1$</td>
</tr>
<tr>
<td>inlet width $b_1$</td>
<td></td>
</tr>
<tr>
<td>impeller diameter $d_2$</td>
<td></td>
</tr>
<tr>
<td>impeller width $b_2$</td>
<td></td>
</tr>
</tbody>
</table>

**For $d_S$-calculation (pumps)**

<table>
<thead>
<tr>
<th><strong>Intake coefficient $\epsilon$</strong></th>
<th><strong>Inflow angle $\beta_{\alpha}$</strong></th>
<th><strong>Minimal relative velocity $w$</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>• Ratio between meridional inflow velocity and specific energy $\epsilon = \frac{c_0}{\sqrt{2Y}}$</td>
<td>• high $\rightarrow$ smaller dimensions, lower friction losses</td>
<td>• small friction and shock losses</td>
</tr>
<tr>
<td>• 0.05…0.4 (rising with $nq$)</td>
<td>• $&lt; 20^\circ$ $\rightarrow$ prevent the risk of cavitation</td>
<td>• only if no cavitation risk !</td>
</tr>
<tr>
<td>• $(k_{m1}$ at Stepanoff)</td>
<td>• $&gt; 15^\circ$ $\rightarrow$ with regard to efficiency</td>
<td>• $f_{ds}$=1.15…1.05 standard impeller, $nq$=15…40</td>
</tr>
</tbody>
</table>
### Suction Specific Speed $n_{ss}$

$$n_{ss} = n \left[ \text{min}^{-1} \right] \frac{\sqrt{Q \left[ \text{m}^3/\text{s} \right]}}{(NPSH_R \left[ \text{m} \right])^{3/4}}$$

(European definition for illustration)

<table>
<thead>
<tr>
<th>Impeller Type</th>
<th>$u_1$</th>
<th>$Q_{min}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Standard suction impeller</td>
<td>$&lt;50$ m/s</td>
<td>160...220</td>
</tr>
<tr>
<td>Suction impeller, axial inflow</td>
<td>$&lt;35$ m/s</td>
<td>220...280</td>
</tr>
<tr>
<td>Suction impeller, cont. shaft</td>
<td>$&lt;50$ m/s</td>
<td>180...240</td>
</tr>
<tr>
<td>High pressure pump</td>
<td>$&gt;50$ m/s</td>
<td>160...190</td>
</tr>
<tr>
<td>Standard inducer</td>
<td>$&gt;35$ m/s</td>
<td>400...700</td>
</tr>
<tr>
<td>Rocket inducer</td>
<td></td>
<td>$&gt;&gt;1000$</td>
</tr>
</tbody>
</table>

### Min. NPSH

$$NPSH_R = \lambda_c \frac{c_{m1}^2}{2g} + \lambda_w \frac{w_1^2}{2g}$$

- $\lambda_c$ suction pressure coefficient for absolute velocity $c$ (inflow acceleration and losses): 1.1 for axial inflow; 1.2...1.35 for radial inflow casing
- $\lambda_w$ suction pressure coefficient for relative velocity $w$ (pressure drop at leading edge): 0.10...0.30 for standard impeller; 0.03...0.06 for inducer

### For $d_1$ Calculation (fan)

**Diameter Ratio $d_1/d_2$**

$$\frac{d_1}{d_2} = 1.25 \sqrt[5/6]{\frac{\psi}{\eta}}$$

### For $b_1$ Calculation (fan)

**Merit. Deceleration**

$$\frac{c_{m1}}{c_{mS}} \approx \frac{d_1}{4b_1} = 2.16 \alpha^{9/8} \leq 2$$

### For $d_2$-Calculation

**Work Coefficient $\Psi$**

- Dimensionless expression for the specific energy:
  
  - 0.7...1.3 centrifugal impeller
  - 0.25...0.7 mixed-flow impeller
### Impeller

<table>
<thead>
<tr>
<th>0.1 ...0.4 axial impeller</th>
</tr>
</thead>
<tbody>
<tr>
<td>• high → small $d_2$, flat characteristic curve</td>
</tr>
<tr>
<td>• low → high $d_2$, steep characteristic curve</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Specific diameter $\delta$</th>
</tr>
</thead>
<tbody>
<tr>
<td>• according to Cordier diagram (see Dimensions)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Outflow angle $\beta_3$</th>
</tr>
</thead>
<tbody>
<tr>
<td>• 6°...13°: recommended for stable performance curve (with $nq$ rising)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>$\beta_{B2} = 90^\circ$</th>
</tr>
</thead>
<tbody>
<tr>
<td>for impeller type &quot;Barske (low $nq$)&quot; only</td>
</tr>
<tr>
<td>• empirical factor $k_{d2} = 1.15 ... 1.29$</td>
</tr>
</tbody>
</table>

| $d_2 = k_{d2} \sqrt{\frac{gH}{n^2 \eta^2}}$ |

<table>
<thead>
<tr>
<th>For $b_2$-calculation</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Outlet width ratio $b_2/d_2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>• 0.04...0.30 (rising with $nq$)</td>
</tr>
</tbody>
</table>

| for pumps: |
| Mer. deceleration $c_{m3}/c_{mS}$ |
| • 0.60...0.95 (rising with $nq$) |

| for pumps: |
| Outlet coefficient $\varepsilon_2$ |
| • Ratio between meridional outlet velocity and specific energy $\varepsilon_2 = c_{m2}/\sqrt{2Y}$ |
| • 0.08...0.26 (rising with $nq$) |
| • ($k_{m2}$ at Stepanoff) |

| for fans: |
| Shroud angle $\varepsilon_{Shr}$ |

<table>
<thead>
<tr>
<th>Efficiency</th>
</tr>
</thead>
</table>
In panel **Efficiency** you have to specify several efficiencies. You have to distinguish between design relevant efficiencies and efficiencies used for information only:

**Design relevant**

- hydraulic efficiency $\eta_h$
- volumetric efficiency $\eta_v$
- tip clearance efficiency $\eta_T$
- additional hydraulic efficiency $\eta_{h^+}$ (displayed for information only, see **Global setup**)

**Information only**

- side friction efficiency $\eta_S$
- mechanical efficiency $\eta_m$
- motor efficiency $\eta_{mot}$

The additional hydraulic efficiency $\eta_{h^+}$ is used additionally for impeller dimensioning in order to compensate the flow losses.

The losses resulting in energy dissipation from the fluid form the **internal efficiency**.

$$\eta_l = \eta_h \eta_v \eta_S \eta_T \eta_{h^+}$$

Internal and mechanical efficiency form the overall efficiency (coupling efficiency) of the stage $\eta_{St}$.

When considering motor losses additionally the overall efficiency of the stage incl. motor $\eta_{St^*}$ is defined.

$$\eta_{St} = \frac{P_Q}{P_D} = \eta_l \eta_m$$  \quad P_Q: pump output, see above

$$\eta_{St^*} = \frac{P_Q}{P_D + P_{el}} = \eta_l \eta_m \eta_{mot}$$  \quad P_D: mechanical power demand (coupling/ driving power)

$$P_{el}: electrical power demand of motor$$

The following summary illustrates the single efficiencies and their classification:
The obtainable overall efficiency correlates to specific speed and to the size and the type of the impeller as well as to special design features like bypass installations and auxiliary aggregates. Efficiencies calculated by approximation functions are representing the theoretical reachable values and they should be corrected by the user if more information about the impeller or the whole pump are available.

The hydraulic efficiency (or blade efficiency) describe the energy losses within the pump caused by friction and vorticity. Friction losses mainly originate from shear stresses in boundary layers. Vorticity losses are caused by turbulence and on the other hand by changes of flow cross section and flow direction which may lead to secondary flow, flow separation, wake behind blades etc.. The hydraulic efficiency is the ratio between specific energy $Y$ and the energy transmitted by the impeller blades:

$$\eta_h = \frac{Y}{Y} \approx \sqrt{\eta} \approx 0.85...0.93$$

The volumetric efficiency is a quantity for the deviation of effective flow rate $Q$ from total flow rate inside the impeller which also includes the circulating flow within the pump casing:

(rising with impeller size)

The tip clearance efficiency is only relevant for unshrouded impellers. It contains losses due to the flow through the gap between blade tips and housing from the pressure to the suction side of the impeller.

<table>
<thead>
<tr>
<th>classification</th>
<th>efficiencies</th>
<th>Relevant for impeller design</th>
</tr>
</thead>
<tbody>
<tr>
<td>stage</td>
<td>$\eta_h$</td>
<td>yes: for energy transmission</td>
</tr>
<tr>
<td></td>
<td>hydraulic</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\eta_T$</td>
<td>tip</td>
</tr>
<tr>
<td></td>
<td>$\eta_v$</td>
<td>volumetric</td>
</tr>
<tr>
<td></td>
<td>$\eta_s$</td>
<td>side friction</td>
</tr>
<tr>
<td>stage incl. motor</td>
<td>$\eta_{mot}$</td>
<td>motor</td>
</tr>
<tr>
<td>electrical</td>
<td>$\eta_{mot}$</td>
<td>motor</td>
</tr>
</tbody>
</table>
blades. The flow losses mainly depend on the tip clearance distance $x_T$ and decrease with rising number of blades and rising blade outlet angle $\beta_2$.

$$
\eta_T = 1 - f_n A_{\text{Ratio}} \quad f_n = f \left( \eta_{q_{\text{Ratio}}} \right) \quad A_{\text{Ratio}} \approx \frac{x_T}{b_2}
$$

The side friction efficiency contains losses caused by rotation of fluid between hub/shroud and housing:

$$
\eta_S = 1 - \frac{P_S}{P} = \begin{cases} 
0.5...0.985 & \text{für } n_q < 40 \\
0.985...0.995 & \text{für } n_q > 40
\end{cases}
$$

The mechanical efficiency mainly includes the friction losses in bearings and seals:

$$
\eta_m = 1 - \frac{P_m}{P} = 0.95...0.995 
$$

(rising with impeller size)

Hydraulic and volumetric efficiency as well as the tip clearance efficiency are most important for the impeller dimensioning because of their influence to $\tilde{Y}$ and/or $\tilde{Q}$. Mechanical and side friction efficiency are affecting only the required driving power of the machine.

If the check box “Use $\eta$ for main dimensions” is set, then main dimension calculation is done on the basis of $Y_{\text{eff}} = 0.5(Y/\eta + Y)$. Otherwise $Y$ - specific work without losses - is used.

**Information**

In the right area of the register **Parameter** you can find again some calculated values for information:

<table>
<thead>
<tr>
<th>Required driving power</th>
<th>$P_D = \frac{P_Q}{\eta_{\text{St}}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Power loss</td>
<td>$P_L = P_D - P_Q = P_D \left( 1 - \eta_{\text{St}} \right)$</td>
</tr>
<tr>
<td>Internal efficiency</td>
<td></td>
</tr>
<tr>
<td>Stage efficiency</td>
<td></td>
</tr>
</tbody>
</table>
7.1.1.3 Dimensions

On page Dimensions, panel Shaft/ hub, the required shaft diameter is computed and the hub diameter is determined by the user.

The main dimensions of a designed impeller - suction diameter \( d_s \), impeller diameter \( d_2 \), outlet width \( b_2 \) - can be seen on Main dimensions panel. They can be recomputed by pressing the Calculate-button. The computation is based on “Euler’s Equation of Turbomachinery”, on the continuity equation and the relations for the velocity triangles as well as on the parameters and parameter ratios given in the tab sheets Setup and Parameters.

Individual main dimensions can be calculated separately using the button inside the value field.

You may accept the proposed values or you can modify them slightly, e.g. to meet a certain normalized diameter.

In case the checkbox Automatic is activated a new calculation will accomplished after any change of parameter. Then the manual alteration of the main dimensions is not possible.

\[
\eta_{st} = \frac{P_a}{P_{el}} = \eta_{st} \eta_{mot}
\]
Due to the Euler equation the impeller diameter $d_2$ and the blade angles $\beta_{B2}$ are coupled (see Outlet triangle). Lower $d_2$ values result in higher $\beta_{B2}$ (higher blade loading) and vice versa. For that reason the resulting average $\beta_{B2}$ value is displayed for information right beside the calculated/specified $d_2$ value.

A specific problem exists for fan impellers. If the suction diameter $d_S$ is calculated by diameter ratio $d_1/d_2$, then the hub has to be planar, i.e. hub diameter $d_H = 0$. Otherwise the empirical correlations are invalid. If the user defines a $d_H$ value deviating from 0, a warning symbol points to this problem. The solution is to select a different parameter for the calculation of the suction diameter $d_S$ (see Parameters).
You can select a value for the diameters $d_a$ from standard specifications. For that purpose you have to press the settings button right beside the input field.

The small dialog gives you the possibility to select a diameter from several standard specifications. If material, standard name and pressure range are selected the lower panel shows all diameters of the chosen standard. One diameter is highlighted as a proposal. Nominal diameter, outside diameter and wall thickness for the marked entry is displayed. Using of + and - buttons additional standard specifications and user defined diameters can be added or existing parameters can be removed from the list.

At File location the name of the file containing the diameters is shown. The file is originally called Diameter.cftdi and is located in the installation directory of CFturbo. Modifications of the list will be saved if the user is leaving the dialog window by clicking the OK-button. In case there are no write permissions the user can choose another directory to save the file. Renaming of files is possible by Save as- functionality. By clicking the Open-button a previously saved file can be opened.

### Neighboring components

In specific cases the dimensions of the neighboring components at inlet and/or outlet can be used to get exactly matching geometry.

This feature is available only for explicitly uncoupled components or side-by-side impellers.

### Information
In the right panel of any tab sheet an information panel is situated, which holds the computed variables in accordance to the actual state of design, the resulting Meridional section as well as the Cordier-Diagramm with the location of the best point. These three sections can be chosen by the appropriate soft buttons in the heading.

In the Value section the following variables are displayed for information which result from calculated or determined main dimensions:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Work coefficient</td>
<td>$\psi = \frac{Y}{u_2^{2/2}}$</td>
</tr>
<tr>
<td>Flow coefficient</td>
<td>$\varphi = \frac{Q}{\pi/4 d_2^2 u_2} = \frac{4b_2 c_m^2}{d_2^2 u_2}$</td>
</tr>
<tr>
<td>Meridional flow coefficient</td>
<td>$\varphi_m = \frac{Q}{\pi d_2 b_2 u_2} = \frac{c_m^2}{u_2}$</td>
</tr>
<tr>
<td>Specific diameter</td>
<td>$\delta = \frac{\psi^{1/4}}{\varphi_{i.t}} = 1.05 d_2 \left( \frac{Y}{Q^2} \right)^{1/4}$</td>
</tr>
<tr>
<td>Average inlet velocity</td>
<td>$\bar{\varepsilon}_{m5} = \frac{Q/\eta_v}{\pi/4 \left( d_5^2 - d_N^2 \right)}$</td>
</tr>
<tr>
<td>Average inlet velocity (net)</td>
<td>$\bar{\varepsilon}_{m5}^* = \frac{Q}{\pi/4 \left( d_5^2 - d_N^2 \right)}$</td>
</tr>
<tr>
<td>Average outlet velocity</td>
<td>$\bar{\varepsilon}_{m3} = \frac{Q/\eta_v}{\pi d_2 b_2}$</td>
</tr>
<tr>
<td>Average outlet velocity (net)</td>
<td>$\bar{\varepsilon}_{m3}^* = \frac{Q}{\pi d_2 b_2}$</td>
</tr>
<tr>
<td>Outlet width ratio</td>
<td>$b_2/d_2$</td>
</tr>
<tr>
<td>Meridional deceleration</td>
<td></td>
</tr>
<tr>
<td>Estimated axial force</td>
<td>$F_{ax} = 0.9 \rho g h \pi/4 \text{Max}(0; d_5^2 - d_N^2)$</td>
</tr>
</tbody>
</table>

Lobanoff/ Ross
<table>
<thead>
<tr>
<th>NPSH&lt;sub&gt;R&lt;/sub&gt; estimation</th>
<th>( NPSH_R(u_{t1}) = f_2 \cdot c_{m1}^2 + f_1 \cdot c_{m1} + f_0 )</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>see diagram below this table</td>
</tr>
<tr>
<td>Pfleiderer</td>
<td>( NPSH_R = \lambda_c \frac{c_{m1}^2}{2g} + \lambda_w \frac{w_1^2}{2g} )</td>
</tr>
<tr>
<td></td>
<td>with loss coefficients</td>
</tr>
<tr>
<td></td>
<td>( \lambda_c = 1.1 \ldots 1.35 ), ( \lambda_w = (0.03) 0.1 \ldots 0.3 )</td>
</tr>
<tr>
<td>G&quot;ulich</td>
<td>( NPSH_R = H \cdot \left( \frac{n_q}{n_{ss}} \right)^{4/3} )</td>
</tr>
<tr>
<td></td>
<td>or ( NPSH_R = \left( \frac{n\sqrt{Q}}{n_{ss}} \right)^{4/3} )</td>
</tr>
<tr>
<td></td>
<td>with suction specific speed ( n_{ss} = 160\ldots280 )</td>
</tr>
<tr>
<td>Stepanoff</td>
<td>( NPSH_R = \alpha \cdot H )</td>
</tr>
<tr>
<td></td>
<td>with cavitation number ( \sigma = 1.22 \cdot 10^{-3} \cdot n_q^{4/3} )</td>
</tr>
<tr>
<td>Petermann</td>
<td>( NPSH_R = \frac{1}{g} \cdot \left( n\sqrt{Q}/S_q \right)^{4/3} )</td>
</tr>
<tr>
<td></td>
<td>with suction number ( S_q = (0.2) 0.4\ldots0.6 (2.0) )</td>
</tr>
<tr>
<td>Europump</td>
<td>( NPSH_R = (0.3\ldots0.5) \cdot n\sqrt{Q} )</td>
</tr>
</tbody>
</table>
The Meridional preview is until now based on the main dimensions only.
The **Cordier** diagram can be used for checking the impeller diameter $d_z$.

See [Cordier](#).

The **Velocity triangles** are the result of a mid-span calculation and are based on the design point and the main dimensions.
7.1.2 Axial Pump / Fan

**Impeller | Main dimensions**

The Main Dimensions menu item is used to define main dimensions of the axial impeller. Main Dimensions are forming the most important basis for all following design steps.
The real flow in an impeller is turbulent and three-dimensional. Secondary flows, separation and reattachment in boundary layers, cavitation, transient recirculation areas and other features may occur. Nevertheless it is useful - and it is common practice in the pump design theory - to simplify the realistic flow applying representative streamlines for the first design approach.

Employing 1D-streamline theory the following cross sections are significant in particular: suction area (index S), just before leading edge (index 0), at the beginning (index 1) and at the end of the blade (index 2), behind the trailing edge (index 3) and at the outlet (index 4).
On page **Setup** you can specify some basic settings.

### General

- **Manual dimensioning**
  In manual dimensioning mode the main dimensions and blade angles are not calculated by CFturbo. All these values are user-defined input values.

- **Unshrouded**
  Design a shrouded (closed) or unshrouded (open) impeller. For an unshrouded impeller you have to define the **tip clearance**, optional different values at inlet and outlet.

- **Material density**
  The material density of the impeller is an informational value that is not relevant for the hydraulic design but is used for the calculation of moments of inertia. Density values can be directly entered or selected from a list by pressing the settings button next to the input area.
• Impeller type
  Beside Standard impeller type the following special impeller types are available with their specific parameters and default settings:
  - for pumps: Inducer
  - for fans: Automotive cooling

• Inflow swirl
  Either the outlet swirl of the upstream component will be used for the determination of the inlet swirl or the absolute inlet flow angle.

• Blade design mode
  Airfoil/ Hydrofoil Design according to Airfoil/Hydrofoil design theory.
  Mean line Design using Euler’s equation on mean lines.
  A completed Airfoil/ Hydrofoil design can be automatically converted to Mean line to allow more design flexibility. In contrast, when switching from Mean line design mode to Airfoil/ Hydrofoil the component has to be re-designed starting from Blade properties.

Multi stage

For a multi stage design the panel Multi stage options is available.

Initial default setting

When creating a new design the initial default settings for some important properties are displayed in the panel Initial default settings. These settings are used in further design steps and can be modified by selecting the Change settings button. Of course these default settings can be modified manually in the appropriate design steps. See Preferences: Impeller/ Stator settings for more information.

Information

Some design point values are displayed in the right Information panel when selecting the page Values (see Global setup).
7.1.2.2 Parameters Pump

On page Parameters you have to put in or to modify parameters resulting from approximation functions in dependence on specific speed \( n_q \) or flow rate \( Q \). See Approximation functions.

Parameter and efficiency values can be handled manually or can be switched to automatic update by the checkbox on top of the page. Then the default values are used always, even after design point modifications (see Global setup).

If the automatic mode is not selected the current default values can be specified by one of the following options:

- globally by the button on top of the page
regionally by the default button within the Parameters or Efficiency region

individually by the default button above the input field when selected

**Parameters**

The panel **Parameters** allows defining alternative parameters in each case for the calculation of the following impeller diameters:

<table>
<thead>
<tr>
<th>inlet</th>
<th>outlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>$d_{S1}$, $d_{H1}$</td>
<td>$d_{S2}$, $d_{H2}$</td>
</tr>
</tbody>
</table>

The following is focusing on normal axial pumps - for **inducers** special correlations are used.

**For $d_{S2}$-calculation**
Work coefficient $\psi$ (= pressure and head coefficient)

- dimensionless expression for the specific energy:
  $$\psi = \frac{Y}{\left(\frac{t_2}{2}\right)^2} \quad \text{and} \quad \psi_{\text{eff}} = \frac{Y_{\text{eff}}}{\left(\frac{t_2}{2}\right)^2}$$
- 0.7 ... 1.3 centrifugal impeller
- 0.25 ... 0.7 mixed-flow impeller
- 0.1 ... 0.6 axial impeller
- high $\rightarrow$ small $d_{S2}$, flat characteristic curve
- low $\rightarrow$ high $d_{S2}$, steep characteristic curve

Specific diameter $\delta$
- according to Cordier diagram (see Dimensions)

For $d_{H2}$ calculation

Diameter ratio $d_{H2}/d_{S2}$

$$\frac{d_{H2}}{d_{S2}} = 0.4 \ldots 0.9$$

If the check box "$\beta_{H2} = 90^\circ$" is set the diameter ratio is set to:

$$\frac{d_{H2}}{d_{S2}} = \frac{\sqrt{Y}}{u_{S2}}$$

Under the assumptions: $c_u \cdot u = Y = \text{const.}$

For $d_{S1}/d_{H1}$-calculation

Meridional velocity ratio $c_{m2}/c_{m1}$

$$\frac{c_{m2}}{c_{m1}} = 0.9 \ldots 1.1$$

Diameter ratio $d_{H1}/d_{S1}$

- strictly axial: $d_{H2} = d_{H1}$ and $d_{S2} = d_{S1}$
- const. hub: $d_{H2} = d_{H1}$
- const. mid: $d_{M2} = d_{M1}$
- const. shroud: $d_{S2} = d_{S1}$

Efficiency
In panel **Efficiency** you have to specify several efficiencies. You have to distinguish between design relevant efficiencies and efficiencies used for information only:

**Design relevant**
- hydraulic efficiency $\eta_h$
- volumetric efficiency $\eta_v$
- additional hydraulic efficiency $\eta_{h+}$ (displayed for information only, see [Global setup](#))

**Information only**
- mechanical efficiency $\eta_m$
- motor efficiency $\eta_{mot}$

The additional hydraulic efficiency $\eta_{h+}$ is used additionally for impeller dimensioning in order to compensate additional flow losses.

The losses resulting in energy dissipation from the fluid form the **internal efficiency**.

$$\eta_i = \eta_h \cdot \eta_v \cdot \eta_{h+}$$

Internal and mechanical efficiency form the overall efficiency (coupling efficiency) of the stage $\eta_{St}$.

When considering motor losses additionally the overall efficiency of the stage incl. motor $\eta_{St}^*$ is defined.

$$\eta_{St} = \frac{P_O}{P_D} = \eta_i \eta_m$$

$P_O$: pump output, see above

$P_D$: mechanical power demand (coupling/ driving power)

$$\eta_{St}^* = \frac{P_O}{P_{el}} = \eta_{St} \eta_{mot}$$

$P_{el}$: electrical power demand of motor

The following summary illustrates the single efficiencies and their classification:

<table>
<thead>
<tr>
<th>classification</th>
<th>efficiencies</th>
<th>Relevant for impeller design</th>
</tr>
</thead>
</table>

© CFturbo GmbH
The obtainable overall efficiency correlates to specific speed and to the size and the type of the impeller as well as to special design features like bypass installations and auxiliary aggregates. Efficiencies calculated by approximation functions are representing the theoretical reachable values and they should be corrected by the user if more information about the impeller or the whole pump are available.

The hydraulic efficiency (or blade efficiency) describe the energy losses within the pump caused by friction and vorticity. Friction losses mainly originate from shear stresses in boundary layers. Vorticity losses are caused by turbulence and on the other hand by changes of flow cross section and flow direction which may lead to secondary flow, flow separation, wake behind blades etc.

The volumetric efficiency is a quantity for the deviation of effective flow rate $\tilde{Q}$ from total flow rate inside the impeller $Q$:

$$\eta_v = \frac{\tilde{Q}}{Q} \approx 0.70 \ldots 0.95$$

(rising with decreasing tip clearance)

The mechanical efficiency mainly includes the friction losses in bearings and seals:

$$\eta_m = 1 - \frac{P_m}{P} \approx 0.95 \ldots 0.995$$

(rising with impeller size)

Total-total and volumetric efficiency are most important for the impeller dimensioning because of their influence to and/or . The mechanical efficiency is affecting only the required driving power of the machine.

If the check box "Use $\eta$ for main dimensions" is set, then main dimension calculation is done on the basis of $Y_{ef} = 0.5(Y/\eta + Y)$. Otherwise $Y$ - specific work without losses - is used.
Information

In the right area of the register **Parameter** you can find again some calculated values for information:

<table>
<thead>
<tr>
<th>Required driving power</th>
<th>$P_D = \frac{P_Q}{\eta_{St}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Power loss</td>
<td>$P_L = P_D - P_Q = P_D \left(1 - \eta_{St}\right)$</td>
</tr>
<tr>
<td>Internal efficiency</td>
<td>$\eta_i = \eta_h \cdot \eta_v \cdot \eta_h^+$</td>
</tr>
<tr>
<td>Stage efficiency</td>
<td>$\eta_{St} = \frac{P_Q}{P_D} = \eta_i \eta_{m}$</td>
</tr>
<tr>
<td>Stage efficiency incl. motor</td>
<td>$\eta_{St}^* = \frac{P_Q}{P_{el}} = \eta_{St} \eta_{mot}$</td>
</tr>
</tbody>
</table>

7.1.2.1 Inducer

Inducers are placed in front of centrifugal pump impellers normally in order to improve the suction performance (reduce NPSHR) of the pump.

For inducers the inlet section is the primary one. The important suction diameter $d_{s1}$ is calculated using the meridional flow coefficient $\varphi_m$:

$$\varphi_m = \frac{Q}{A_{s1} u_{s1}} = \frac{4Q}{\pi \left(\left(d_{s1}^2 - d_{sh}^2\right) \pi d_{s1} n\right)} = \frac{c_{m1}}{u_{s1}} = \tan \beta_{0S}$$

In CFturbo the so called Brumfield curve is used to estimate an appropriate $\varphi_m$ value to achieve a required level of suction performance. Input values is the suction specific speed $n_{ss}$:

(or the US definition $N_{ss}$, see Preferences/Units/Other)

The Brumfield curve can be displayed and also modified if necessary by clicking on the function button just right of the $n_{ss}$ edit field.
The $\phi_m$ value can be calculated automatically from the given $n_{ss}$ value or modified manually. There is a limit of $\phi_m = 0.06$, lower values will result in backflow at blade tip and cavitation induced flow instability.

Alternatively you can specify the rel. inlet flow angle $\beta_{0s}$ or the meridional flow coefficient $\phi_m$ directly. Furthermore the parameters for classic axial pump design could be used alternatively.

The inlet hub diameter $d_{H1}$ is calculated using the diameter ratio $\nu_1$:

$$\nu_1 = \frac{d_{H1}}{d_{S1}} = 0.2 \ldots 0.4$$

Typical for inducers is a constant tip (shroud) diameter. The hub diameter can increase from inlet to outlet slightly in order to use centrifugal effect for energy transmission. The meridional velocity ratio between inlet and outlet can be used to estimate the outlet cross section:

Alternatively the diameter ratio $\nu_2 = d_{H2}/d_{S2}$ at outlet similar to the inlet side can be used.
7.1.2.3 Parameters Fan

On page **Parameters** you have to put in or to modify parameters resulting from approximation functions in dependence on specific speed \( n_q \) or flow rate \( Q \). See **Approximation functions**.

Parameter and efficiency values can be handled manually or can be switched to automatic update by the checkbox on top of the page. Then the default values are used always, even after design point modifications (see **Global setup**).

If the automatic mode is not selected the current default values can be specified by one of the following options:

- **Set default** globally by the button on top of the page.
regionally by the default button within the Parameters or Efficiency region

individually by the default button within the input field when selected

**Parameters**

The panel Parameters allows defining alternative parameters in each case for the calculation of the following impeller diameters:

<table>
<thead>
<tr>
<th>Inlet</th>
<th>outlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>$d_{S1}$</td>
<td>$d_{H1}$</td>
</tr>
<tr>
<td>$d_{S2}$</td>
<td>$d_{H2}$</td>
</tr>
</tbody>
</table>

For $d_{S2}$ calculation

- dimensionless expression for the specific energy:
  - Work coefficient $\psi$ (= pressure and head coefficient)
    - $0.7 \ldots 1.3$ centrifugal impeller
    - $0.25 \ldots 0.7$ mixed-flow impeller
    - $0.1 \ldots 0.6$ axial impeller
  - high $\rightarrow$ small $d_{S2}$, flat characteristic curve
  - low $\rightarrow$ high $d_{S2}$, steep characteristic curve
Specific diameter $\delta$
- according to Cordier diagram (see Dimensions)

Flow coefficient $\varphi$
- dimensionless flow rate
  $$\varphi = \frac{Q}{\pi/4 d_2^2 u_2}$$
- high $\rightarrow$ small $d_{s2}$, flat characteristic curve
- low $\rightarrow$ high $d_{s2}$, steep characteristic curve

For $d_{h2}$ calculation

Diameter ratio $d_{h2}/d_{s2}$
Can be estimated under assumption $\beta_2 = 90^\circ$ @ hub and $c_{u \cdot u} = \text{const.}$ by:

$$\frac{d_{h2}}{d_{s2}} = \sqrt{\frac{\varphi}{2}}$$

For $d_{s1}/d_{h1}$-calculation

Meridional velocity ratio $c_{m2}/c_{m1}$
- $c_{m2}/c_{m1} = 0.9 ... 1.1$

Diameter ratio $d_{h1}/d_{s1}$
- strictly axial: $d_{h2} = d_{h1}$ and $d_{s2} = d_{s1}$
- const. hub: $d_{h2} = d_{h1}$
- const. mid: $d_{m2} = d_{m1}$
- const. shroud: $d_{s2} = d_{s1}$

Efficiency

In panel Efficiency you have to specify several efficiencies. You have to distinguish between design relevant efficiencies and efficiencies used for information only:

Design relevant
- total-total efficiency $\eta_{tt}$
- volumetric efficiency $\eta_v$
- additional total-total efficiency $\eta_{tt}^+$ (displayed for information only, see Global setup)

**Information only**
- mechanical efficiency $\eta_m$
- motor efficiency $\eta_{mot}$

The additional total-total efficiency $\eta_{tt}^+$ is used for impeller dimensioning in order to compensate additional flow losses.

The losses resulting in energy dissipation from the fluid form the internal efficiency.

$$\eta_i = \eta_{tt} \cdot \eta_v \cdot \eta_{tt}^+$$

Impeller and mechanical efficiency form the overall efficiency (coupling efficiency) of the stage $\eta_{St}^-$.

When considering motor losses additionally the overall efficiency of the stage incl. motor $\eta_{St}^*$ is defined.

$$\eta_{St} = \frac{P_Q}{P_D} = \eta_i \eta_m$$

$P_Q$: power output, see above
$P_D$: mechanical power demand (coupling/driving power)

$$\eta_{St}^* = \frac{P_Q}{P_{el}} = \eta_{St} \eta_{mot}$$

$P_{el}$: electrical power demand of motor

The following summary illustrates the single efficiencies and their classification:

<table>
<thead>
<tr>
<th>classification</th>
<th>efficiencies</th>
<th>Relevant for impeller design</th>
</tr>
</thead>
<tbody>
<tr>
<td>stage</td>
<td>impeller</td>
<td></td>
</tr>
<tr>
<td>$\eta_{tt}$</td>
<td>additional total-total</td>
<td>yes: for energy transmission</td>
</tr>
<tr>
<td>$\eta_{tt}^+$</td>
<td>total-total</td>
<td></td>
</tr>
<tr>
<td>$\eta_V$</td>
<td>volumetric</td>
<td>yes: for flow rate</td>
</tr>
</tbody>
</table>
The obtainable overall efficiency correlates to specific speed and to the size and the type of the impeller as well as to special design features like bypass installations and auxiliary aggregates. Efficiencies calculated by approximation functions are representing the theoretical reachable values and they should be corrected by the user if more information about the impeller or the whole pump are available.

The hydraulic efficiency (or blade efficiency) describe the energy losses within the pump caused by friction and vorticity. Friction losses mainly originate from shear stresses in boundary layers. Vorticity losses are caused by turbulence and on the other hand by changes of flow cross section and flow direction which may lead to secondary flow, flow separation, wake behind blades etc.

The volumetric efficiency is a quantity for the deviation of effective flow rate \( \tilde{Q} \) from total flow rate inside the impeller \( \tilde{Q} \):

\[
\eta_v = \frac{Q}{\tilde{Q}} \approx 0.70 \ldots 0.95
\]

(rising with decreasing tip clearance)

The mechanical efficiency mainly includes the friction losses in bearings and seals:

\[
\eta_m = 1 - \frac{P_m}{P} \approx 0.95 \ldots 0.995
\]

(rising with impeller size)

Total-total and volumetric efficiency are most important for the impeller dimensioning because of their influence to \( \tilde{Y} \) and/or \( \tilde{Q} \). The mechanical efficiency is affecting only the required driving power of the machine.

If the check box "Use \( \eta \) for main dimensions" is set, then main dimension calculation is done on the basis of \( Y_{\text{eff}} = 0.5(Y/\eta + Y) \). Otherwise \( Y \) - specific work without losses - is used.

**Information**

In the right area of the register **Parameter** you can find again some calculated values for **information**:
### Required driving power

\[ P_D = \frac{P_Q}{\eta_{St}} \]

### Power loss

\[ P_L = P_D - P_Q = P_D \left( 1 - \eta_{St} \right) \]

### Internal efficiency

\[ \eta_{im} = \eta_{it} \eta_{V} \]

### Stage efficiency

\[ \eta_{St} = \frac{P_Q}{P_D} = \eta_{im} \eta_{St} \]

### Stage efficiency incl. motor

\[ \eta_{St}^* = \frac{P_Q}{P_{el}} = \eta_{St} \eta_{mot} \]

### Assembly

<table>
<thead>
<tr>
<th>Pressure difference</th>
<th>Remark</th>
</tr>
</thead>
<tbody>
<tr>
<td>[ \Delta p_u = \Delta p_{u,imp} - \left( \zeta_D + \zeta_{GV} \right) \frac{P_{c2}}{2} c_{m2}^2 ]</td>
<td>inline installation, resistance coefficients according to Wallis</td>
</tr>
<tr>
<td>[ \zeta_D = 0.1 \cdot \left( 2 \left( \frac{d_{H2}}{d_{S2}} \right)^2 - \left( \frac{d_{H2}}{d_{S2}} \right)^4 \right) ]</td>
<td></td>
</tr>
<tr>
<td>[ \zeta_{GV} = 0.1 \cdot \left( 1 - \frac{c_{u2}^2}{c_{m2}^2} \right) ]</td>
<td></td>
</tr>
</tbody>
</table>

| \[ \Delta p_u = \Delta p_{u,imp} - \left( \zeta_D + \zeta_{GV} \right) \frac{P_{c2}}{2} c_{m2}^2 \] | inline installation, Carnot type diffuser, resistance coefficients according to Wallis | B |
| \[ \zeta_D = \left( 1 - \left( -0.5 \frac{d_{H2}}{d_{S2}} + 0.95 \right) \right) \cdot \left( 2 \frac{d_{H2}}{d_{S2}} \right) \] | |
| \[ \zeta_{GV} = 0.1 \cdot \left( 1 - \frac{c_{u2}^2}{c_{m2}^2} \right) \] | |

| \[ \Delta p_u = \Delta p_{u,imp} - \left( \zeta_D + \zeta_{GV} \right) \frac{P_{c2}}{2} c_{m2}^2 \] | inline installation, swirl assumed to dissipate in duct, pseudo total-to-total pressure rise, resistance coefficients according to Wallis | C |
| \[ \zeta_D = \left( 1 - \left( -0.5 \frac{d_{H2}}{d_{S2}} + 0.95 \right) \right) \cdot \left( 2 \frac{d_{H2}}{d_{S2}} \right) \] | |
| \[ \zeta_{GV} = 0.1 \cdot \left( 1 - \frac{c_{u2}^2}{c_{m2}^2} \right) \] | |
Impeller

\[ \Delta p_{\text{ts}} = \Delta p_{u,a} - \zeta \frac{\rho}{2} c_a^2 \]
\[ c_a = c_{m2} \cdot \left( 1 - \left( \frac{d_{S2}}{d_{S1}} \right)^2 \right) \]

\[ \Delta p_{\text{ts}} = \Delta p_{u,b} - \zeta \frac{\rho}{2} c_a^2 \]
\[ c_a = c_{m2} \cdot \left( 1 - \left( \frac{d_{S2}}{d_{S1}} \right)^2 \right) \]

\[ \Delta p_{\text{ts}} = \Delta p_{u,\text{Imp}} - \zeta_{GV} \frac{\rho}{2} c_{m2}^2 - \frac{\rho}{2} c_{u2}^2 \]
\[ \zeta_{GV} = 0.1 \cdot \left( 1 - \frac{c_{u2}^2}{c_{m2}^2} \right) \]

\[ \Delta p_{\text{ts}} = \Delta p_{u,\text{Imp}} - \frac{\rho}{2} c_{m2}^2 - \frac{\rho}{2} c_{u2}^2 \]

7.1.2.4 Dimensions

The main dimensions of a designed impeller - suction diameter \( d_{S1} \) and \( d_{H1} \) and outlet diameter \( d_{S2} \) and \( d_{H2} \) - can be seen on Main dimensions panel. They can be recomputed by pressing the Calculate-button. The computation is based on "Euler's Equation of Turbomachinery", on the continuity equation and the relations for the velocity triangles as well as on the parameters and parameter ratios given in the tab sheets Setup and Parameters.

Individual main dimensions can be calculated separately using the button inside the value field.

You may accept the proposed values or you can modify them slightly, e.g. to meet a certain normalized diameter.
In case the checkbox **Automatic** is activated a new calculation will accomplished after any change of parameter. Then the manual alteration of the main dimensions is not possible.
**Neighboring components**

In specific cases the dimensions of the neighboring components at inlet and/or outlet can be used to get exactly matching geometry.

This feature is available only for explicitly **uncoupled** components or side-by-side impellers.

**Information**

In the right panel of any tab sheet an **information** panel is situated, which holds the computed variables in accordance to the actual state of design, the resulting **Meridional section** as well as the **Cordier-Diagramm** with the location of the best point. These three sections can be chosen by the appropriate soft buttons in the heading.
In the **Value** section the following variables are displayed for information which result from calculated or determined main dimensions:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>Work coefficient</td>
<td>$\Psi = \frac{Y}{u_{2s}^2/2}$</td>
</tr>
<tr>
<td>Flow coefficient</td>
<td>$\varphi = \frac{Q}{\pi/4 , d_{2s}^2 u_{2s}}$</td>
</tr>
<tr>
<td>Meridional flow coefficient</td>
<td>$\varphi_{m2} = \frac{\pi}{4} \left( \frac{d_{2s}^2 - d_{2h}^2}{d_{2h}^2} \right) \frac{\varphi_2}{u_2} = \frac{c_{m2}}{u_2}$</td>
</tr>
<tr>
<td>Specific diameter</td>
<td>$\delta = \frac{\psi^{1/4}}{\varphi_{1/2}} = 1.05 d_{2s} \left( \frac{Y}{\varphi_{1/2} \varphi_{2}} \right)^{1/4}$</td>
</tr>
<tr>
<td>Average inlet velocity</td>
<td>$c_{m1} = \frac{Q}{\pi/4 (d_{S1}^2 - d_{H1}^2)}$</td>
</tr>
<tr>
<td>Inlet abs. circ. velocity component</td>
<td>$c_{u1}$</td>
</tr>
<tr>
<td>Inlet relative velocity</td>
<td>$w_1$</td>
</tr>
<tr>
<td>Average outlet velocity</td>
<td>$c_{m2} = \frac{Q}{\pi/4 (d_{S2}^2 - d_{H2}^2)}$</td>
</tr>
<tr>
<td>Outlet circ. velocity component</td>
<td>$c_{u2} = \left( Y - u_1 c_{u1} \right)$</td>
</tr>
<tr>
<td>Outlet relative velocity</td>
<td>$w_2$</td>
</tr>
<tr>
<td>Meridional velocity ratio</td>
<td>$c_{m2} / c_{m1}$</td>
</tr>
<tr>
<td>Relative velocity ratio</td>
<td>$w_2 / w_1$</td>
</tr>
<tr>
<td>Axial force (thrust)</td>
<td>$F_{ax} = p_{t2} \cdot A_2 - p_{t1} \cdot A_1$</td>
</tr>
<tr>
<td>NPSH$_R$ estimation</td>
<td><strong>Lobanoff/ Ross</strong></td>
</tr>
<tr>
<td></td>
<td>$NPSH_R (u_{t1}) = f_2 \cdot c_{m1}^2 + f_1 \cdot c_{m1} + f_0$</td>
</tr>
</tbody>
</table>
see diagram below this table

<table>
<thead>
<tr>
<th>Impeller</th>
<th>Equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pfleiderer</td>
<td>[ NPSH_R = \lambda_c \frac{cm^2}{2g} + \lambda_w \frac{w^2}{2g} ]</td>
</tr>
<tr>
<td>with loss coefficients ( \lambda_c = 1.1 \ldots 1.35, \lambda_w = (0.03) 0.1 \ldots 0.3 )</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Gülich</th>
<th>Equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>( NPSH_R = H \cdot \left( \frac{n_q}{n_{ss}} \right)^{4/3} ) or ( NPSH_R = (n\sqrt{Q}/n_{ss})^{4/3} )</td>
<td></td>
</tr>
<tr>
<td>with suction specific speed ( n_{ss} = 160 \ldots 280 )</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Stepanoff</th>
<th>Equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>( NPSH_R = \sigma \cdot H )</td>
<td></td>
</tr>
<tr>
<td>with cavitation number ( \sigma = 1.22 \cdot 10^{-3} \cdot n_q^{4/3} )</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Petermann</th>
<th>Equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>( NPSH_R = \frac{1}{g} \cdot \left( \frac{n\sqrt{Q}}{S_q} \right)^{4/3} )</td>
<td></td>
</tr>
<tr>
<td>with suction number ( S_q = (0.2) 0.4 \ldots 0.6 (2.0) )</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Europump</th>
<th>Equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>( NPSH_R = (0.3 \ldots 0.5) \cdot n\sqrt{Q} )</td>
<td></td>
</tr>
</tbody>
</table>
The Meridional preview is until now based on the main dimensions only.
The **Cordier** diagram can be used for checking the impeller diameter $d_2$.

See [Cordier](#).
The Velocity triangles are the result of a mid-span calculation and are based on the design point and the main dimensions.
7.1.3 Centrifugal Compressor

The Main Dimensions menu item is used to define main dimensions of the impeller. Main Dimensions are forming the most important basis for all following design steps.
The real flow in a compressor impeller is turbulent and three-dimensional. Secondary flows, separation and reattachment in boundary layers, transient recirculation areas and other features may occur. Nevertheless it is useful - and it is common practice in the compressor design theory - to simplify the realistic flow applying representative streamlines for the first design approach.

Employing 1D-streamline theory the following cross sections are significant in particular: suction area (index S), just before leading edge (index 0), at the beginning (index 1) and at the end of the blade (index 2) and finally behind the trailing edge (index 3).

Details

- Setup
- Parameters
- Dimensions
7.1.3.1 Setup

On page Setup you can specify some basic settings.

General

- **Manual dimensioning**
  In manual dimensioning mode the main dimensions and blade angles are not calculated by CFturbo. All these values are user-defined input values.

- **Unshrouded**
  Design a shrouded (closed) or unshrouded (open) impeller. For an unshrouded impeller you have to define the **tip clearance**, optional different values at
inlet and outlet.

- **Splitter blades**
  Design impeller with or without splitter blades.

- **Material density**
  The material density of the impeller is an informational value that is not relevant for the aerodynamic design but is used for the calculation of moments of inertia. Density values can be directly entered or selected from a list by pressing the settings button next to the input area.

- **Inflow swirl**
  Either the outlet swirl of the upstream component will be used for the determination of the inlet swirl or the absolute inlet flow angle.

**Multi stage**

For a multi stage design the panel **Multi stage options** is available.

**Initial default setting**

When creating a new design the initial default settings for some important properties are displayed in the panel **Initial default settings**. These settings are used in further design steps and can be modified by selecting the **Change settings** button. Of course these default settings can be modified
manually in the appropriate design steps. See Preferences: Impeller/ Stator settings for more information.

Information

Some design point values are displayed in the right Information panel when selecting the page Values (see Global setup).

7.1.3.2 Parameters

On page Parameters you have to put in or to modify parameters resulting from approximation functions in dependence on specific speed $n_q$ or flow rate $Q$ (see Approximation functions).

For details of how to handle the parameter edit fields please see Edit fields with empirical functions.
Parameter and efficiency values can be handled manually or can be switched to automatic update by the checkbox on top of the page. Then the default values are used always, even after design point modifications (see Global setup).

If the automatic mode is not selected the current default values can be specified by one of the following options:

- globally by the button on top of the page
- regionally by the default button within the Parameters or Efficiency region
- individually by the default button within the input field when selected

**Parameters**

The panel Parameters allows defining alternative values in each case for the calculation of the following impeller main dimensions:

- suction diameter \( d_s \)
- impeller diameter \( d_2 \)
- impeller width \( b_2 \)

For \( d_2 \)-calculation
Work coefficient $\Psi$  
(= pressure and head coefficient)
- dimensionless expression for the specific enthalpy $\Delta h_{s} = Y$ and $\Delta h = Y_{\text{eff}}$ resp.
  
  $$\Psi = \frac{\Delta h_{s}}{u_{2}^{2}/2} \quad \text{and} \quad \Psi = \frac{\Delta h}{u_{2}^{2}/2}$$
- high $\rightarrow$ small $d_{2}$, flat characteristic curve
  low $\rightarrow$ high $d_{2}$, steep characteristic curve

Flow coefficient $\varphi$
- dimensionless flow rate
  $$\varphi = \frac{Q}{\pi/4 d_{2}^{2} u_{2}}$$
  0.01 narrow centrifugal impeller, untwisted blades
  0.15 mixed-flow impeller, twisted blades

Specific diameter $\delta$
- according to Cordier diagram (see Dimensions $\Delta h$)

Machine Mach number $M_{a_{u}}$
- dimensionless peripheral speed of impeller related to total inlet speed of sound
  $$M_{a_{u}} = \frac{u_{2}}{a_{t,5}}$$

Peripheral speed $u_{2}$
- Limiting values due to strength as a function of the material

For $b_{2}$-calculation

Outlet width ratio $b_{2}/d_{2}$
- 0.01…0.15 (with $n q$ rising)

Meridional flow coefficient $\varphi_{m}$
- dimensionless flow rate
  $$\varphi_{m} = \frac{Q_{2}}{\pi d_{2} b_{2} u_{2}} = \frac{c_{2m}}{u_{2}}$$
  0.10…0.50 (with $n q$ rising)

For $d_{1}$-calculation (optional)

Diameter ratio $d_{1}/d_{2}$
- $d_{1}/d_{2} = 0.3…0.8$
Relative deceleration $w_2/w_1$

$w_2/w_1 > 0.7$ or $f(b_2/d_2)$

For $b_1$-calculation (optional)

Meridional deceleration $c_{m2}/c_{m1}$

$c_{m2}/c_{m1} = 0.8...1.25$

for $d_S$-calculation

| Meridional deceleration $c_{m1}/c_{mS}$ or $c_{m2}/c_{mS}$ | $c_{m1}/c_{mS} = 0.9...1.1$
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$c_{m2}/c_{mS}$</td>
<td>$c_{m2}/c_{mS} = 0.7...1.3$</td>
</tr>
</tbody>
</table>

Relative inlet flow angle $\beta_S$

$\beta_S = \arctan \frac{c_{mS}}{w_{uS}} = \arctan \frac{c_{mS}}{u_S - c_{uS}} \approx 30^\circ$

Relative inlet Mach number $M_{wS}$

$M_{wS} = \frac{w_S}{a_S} = \sqrt{\frac{c_{mS}^2 + w_{uS}^2}{a_S}} \leq 0.75...0.85$

Diameter ratio $d_S/d_2$

$dS/d_2 = 0.65...0.8$

The relative inlet Mach number can be implemented in a certain range only. The lower limit is given by the fact that small values for $dS$ (high meridional velocity $c_{mS}$) as well as high values for $dS$ (high rotational speed $u_S$ and therefore $w_{uS}$) result in an increasing relative velocity $w_S$. Due to the square root equation of $M_{wS}$, two different values of $dS$ are possible. For certain boundary conditions, a minimal relative velocity and therefore a minimal relative inlet Mach number is existing always.
In this context it's important to know that the fluid density is dependent on the velocity and therefore on the geometrical dimensions.

**Efficiency**

In panel **Efficiency** you have to specify several efficiencies. You have to distinguish between design relevant efficiencies and efficiencies used for information only:

**Design relevant**
- total-total efficiency $\eta_{tt}$
- volumetric efficiency $\eta_v$
- additional total-total efficiency $\eta_{tt}^+$ (displayed for information only, see [Global setup](#))

**Information only**
- mechanical efficiency $\eta_m$
- motor efficiency $\eta_{mot}$

The additional total-total efficiency $\eta_{tt}^+$ is used for impeller dimensioning in order to compensate additional flow losses.

The losses resulting in energy dissipation from the fluid form the **internal efficiency**.
Impeller and mechanical efficiency form the overall efficiency (coupling efficiency) of the stage \( \eta_{St} \).

When considering motor losses additionally the overall efficiency of the stage incl. motor \( \eta_{St}^* \) is defined.

\[
\eta_{St} = \frac{P_Q}{P_D} = \eta_m \quad \text{P}_Q: \text{output power, see above}
\]

\[
\eta_{St} = \frac{P_Q}{P_{el}} = \eta_{mot} \quad \text{P}_{el}: \text{electrical power demand of motor}
\]

The following summary illustrates the single efficiencies and their classification:

<table>
<thead>
<tr>
<th>classification</th>
<th>efficiencies</th>
<th>Relevant for impeller design</th>
</tr>
</thead>
<tbody>
<tr>
<td>stage</td>
<td>( \eta_C )</td>
<td>additional casing: yes: for energy transmission</td>
</tr>
<tr>
<td></td>
<td>( \eta_t )</td>
<td>total-total</td>
</tr>
<tr>
<td>impeller</td>
<td>( \eta_V )</td>
<td>volumetric: yes: for flow rate</td>
</tr>
<tr>
<td></td>
<td>( \eta_m )</td>
<td>mechanical: no: for overall information only</td>
</tr>
<tr>
<td>stage incl. motor</td>
<td>electrical</td>
<td>( \eta_{mot} ): motor</td>
</tr>
</tbody>
</table>
between the actual specific energy $Y$ and the energy transmitted by the impeller blades without any losses:

$$\eta_{tt} = \frac{Y}{\overline{Y}}$$

The volumetric efficiency is a quantity for the deviation of effective flow rate $Q$ from total flow rate inside the impeller $\overline{Q}$ which also includes the circulating flow within the casing:

$$\eta_v = \frac{Q}{\overline{Q}} \approx 0.93...0.99$$

(rising with impeller size)

The mechanical efficiency mainly includes the friction losses in bearings and seals:

$$\eta_m = 1 - \frac{P_m}{P} \approx 0.95...0.995$$

(rising with impeller size)

Impeller efficiency and volumetric efficiency are most important for the impeller dimensioning because of their influence to $\overline{Q}$ and/or $\overline{Y}$. The mechanical efficiency is affecting only the required driving power of the machine.

If the check box "Use $\eta$ for main dimensions" is set, then main dimension calculation is done on the basis of $\Delta h = 0.5(\Delta h_{is}/\eta + \Delta h_{is})$. Otherwise $\Delta h_{is}$ - the isentropic specific enthalpy - is used.

**Information**

In the right panel of the tab sheet **Parameter** you can find again some calculated values for information:

<table>
<thead>
<tr>
<th>Requirement</th>
<th>Formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required driving power</td>
<td>$P_D = \frac{P_Q}{\eta_{St}}$</td>
</tr>
<tr>
<td>Power loss</td>
<td>$P_L = P_D - P_Q = P_D (1 - \eta_{St})$</td>
</tr>
<tr>
<td>Internal efficiency</td>
<td></td>
</tr>
<tr>
<td>Stage efficiency</td>
<td></td>
</tr>
</tbody>
</table>
### Stage efficiency incl. motor

\[ \eta_{st} = \frac{p_{0}}{p_{el}} = \eta_{st} \eta_{mot} \]

### Total-to-static efficiency

\[ \eta_{ts} = \frac{\pi_{t}^{\kappa-1} \left( 1 - \frac{c_{2}^{2}}{2c_{p}T_{s}} \right) - 1}{\tau_{t} - 1} \]

(perfect gas model)

### Polytropic efficiency

\[ \eta_{p} = \frac{n}{n-1} \left( \frac{\kappa}{\kappa-1} \right) \]

\( n \) .. polytropic exponent
\( \kappa \) .. isentropic exponent

---

### 7.1.3.3 Dimensions

On page **Dimensions**, panel **Shaft/ hub**, the required shaft diameter is computed and the hub diameter is determined by the user.

→ **Shaft/Hub**
The main dimensions of a designed impeller - suction diameter $d_s$, impeller diameter $d_2$, outlet width $b_2$ - can be seen on Main dimensions panel. They can be recomputed by pressing the Calculate-button. The computation is based on “Euler’s Equation of Turbomachinery”, on the continuity equation and the relations for the velocity triangles as well as on the parameters and parameter ratios given in the tab sheets Setup and Parameters.

Individual main dimensions can be calculated separately using the button inside the value field.

You may accept the proposed values or you can modify them slightly, e.g. to meet a certain normalized diameter.

In case the checkbox Automatic is activated a new calculation will accomplished after any change of parameter. Then the manual alteration of the main dimensions is not possible.
Due to the Euler equation the impeller diameter $d_2$ and the blade angles $\beta_{B_2}$ are coupled (see Outlet triangle $\Delta B_2$). Lower $d_2$ values result in higher $\beta_{B_2}$ (higher blade loading) and vice versa. For that reason the resulting average $\beta_{B_2}$ value is displayed for information right beside the calculated/ specified $d_2$ value.

**Neighboring components**

In specific cases the dimensions of the neighboring components at inlet and/ or outlet can be used to get exactly matching geometry.
This feature is available only for explicitly uncoupled components or side-by-side impellers.

**Information**

In the right panel of any tab sheet an information panel is situated, which holds the computed variables in accordance to the actual state of design, the resulting Meridional section as well as the Cordier-Diagramm with the location of the best point. These three sections can be chosen by the appropriate soft buttons in the heading.

In the Value section the following variables are displayed for information which result from calculated or determined main dimensions:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>Work coefficient</td>
<td>$\psi = \frac{Y}{u_2^2/2} = 0.6\ldots1.5$</td>
</tr>
<tr>
<td>Flow coefficient</td>
<td>$\varphi = \frac{Q}{\pi/4 \ d_2^2 u_2} = 0.01\ldots0.5$</td>
</tr>
<tr>
<td>Meridional flow coefficient</td>
<td>$\varphi_m = \frac{Q_2}{\pi d_2 b_2 u_2} = \frac{c_{m2}}{u_2} = 0.1\ldots0.5$</td>
</tr>
<tr>
<td>Specific diameter</td>
<td>$\delta = \frac{\psi^{1/4}}{\varphi_{1/2}} = 1.05 d_2 \left(\frac{Y}{Q_{IS}^2}\right)^{1/4}$</td>
</tr>
<tr>
<td>Tangential force coefficient</td>
<td>$c_t = \frac{\psi}{\eta_t \ \varphi_m} = 3\ldots6$</td>
</tr>
<tr>
<td>Outlet width ratio</td>
<td>$b_2/d_2 = 0.01\ldots0.15$</td>
</tr>
<tr>
<td>Diameter ratio</td>
<td>$d_S/d_2$</td>
</tr>
<tr>
<td>Inlet Mach number</td>
<td></td>
</tr>
<tr>
<td>Outlet Mach number</td>
<td>Outlet Mach number: [ Ma_{c,2} = \frac{1}{\sqrt{\frac{a_{1,2}}{c_2}^2 - \frac{k-1}{2}}} \leq 1 ] (perfect gas model)</td>
</tr>
<tr>
<td>---------------------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Degree of Reaction</td>
<td>Degree of Reaction: [ R = 1 - \frac{c_2^2}{2Y} ]</td>
</tr>
<tr>
<td>thermodynamic values for</td>
<td>thermodynamic values for: [ \rho, p, T, c_m, c_u, w, u, p_t, p_i, T_t ]</td>
</tr>
<tr>
<td>impeller inlet (cross section S)</td>
<td>impeller outlet (cross section 2)</td>
</tr>
</tbody>
</table>

In the impeller outlet (cross section 2) two different total pressure values are given: \( p_{1,2} \) and \( p_{12\text{imp}} \).

This is based upon the following assumption: All non-rotating components of the project are considered as being loss-less. An additional efficiency can be defined in the global setup. Additional losses connected to this additional efficiency are considered within the impeller. That is to say, straight after the trailing edge an adiabatic expansion takes place reducing the total pressure of the impeller \( p_{12\text{imp}} \) to the value at the inlet of the next non-rotating component \( p_{1,2} \). In accordance to the following diagram these two different pressure values are calculated (assuming perfect gas behavior) by:

\[
p_{1,2} = p_{11}\left(\frac{h_{12s}}{h_{11}}\right)^{\frac{x}{k-1}} \quad \text{and} \quad p_{12\text{imp}} = p_{11}\left(\frac{h_{12\text{imp}}}{h_{11}}\right)^{\frac{x}{k-1}} = p_{11}\left(1 + \frac{\Delta h_{ls}}{h_{11} \cdot n_m}\right)^{\frac{x}{k-1}}
\]
All thermodynamic values are calculated on the basis of the total pressure $p_{t2\text{imp}}$ in cross section 2.

If a stator is located prior the impeller, boundary conditions for the determination of the main dimensions will be calculated on the basis of the thermodynamic state at the outlet of this stator. In case of an undefined thermodynamic state at this location the inlet boundary conditions (i.e. total pressure and temperature as well as swirl) will be taken from the global setup and a warning is generated.

It might be that for the chosen configuration of global setup and main dimensions a reasonable thermodynamic state cannot be calculated. This may be the case if e.g. the mass flow is too high for the chosen cross sections. Then again a warning is generated.

The Meridional preview is based on the until now designed main dimensions.
The **Cordier** diagram can be used for checking the impeller diameter $d_2$. See [Cordier](#).
Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thermodynamic state cannot be calculated at inlet/outlet. Possible reason: choked flow. Consider change of main dimensions or global setup.</td>
<td>Increase the dimensions (width, diameters etc.) or change the Global setup (e.g. decrease mass flow, increase pressure, decrease temperature).</td>
</tr>
<tr>
<td>The dimensions might be too tight for the specified mass flow and inlet conditions. In other words the mass flow is higher than the choked mass flow rate for the particular inlet condition and the respective cross section.</td>
<td></td>
</tr>
</tbody>
</table>
7.1.4 Radial-inflow Gas Turbine

The Main Dimensions menu item is used to define main dimensions of the rotor. Main Dimensions are forming the most important basis for all following design steps.

The real flow in a turbine rotor is turbulent and three-dimensional. Secondary flows, separation and reattachment in boundary layers, transient recirculation areas and other features may occur. Nevertheless it is useful - and it is common practice in the turbine design theory - to simplify the realistic flow applying representative streamlines for the first design approach.

Employing 1D-streamline theory the following cross sections are significant in particular: area just before leading edge (index 0), at the beginning (index 1) and at the end of the blade (index 2) and finally behind the trailing edge (index 3).

The cross section (S) is situated at the suction side in the connection flange of the component following the turbine.

The design of the main dimensions has to be made in a strict order. This will be secured by the following: One step within the design has to be finished completely before the next can be accomplished. That is to say, the changeability of a tab sheet will be disabled by CFturbo until all necessary parameters have been specified.
7.1.4.1 Setup

On page Setup one can specify some basic settings.

**General**

- **Manual dimensioning**
  In manual dimensioning mode the main dimensions and blade angles are not calculated by CFturbo. All these values are user-defined input values.

- **Unshrouded**
  Design a shrouded (closed) or unshrouded (open) rotor.
  For an unshrouded rotor you have to define the **tip clearance**, optional different values at inlet and outlet.
• **Splitter blades**  
  Design the rotor with or without splitter blades.

• **Material density**  
  The material density of the impeller is an informational value that is not relevant for the aerodynamic design but is used for the calculation of moments of inertia. Density values can be directly entered or selected from a list by pressing the settings button next to the input area.

• **Inflow swirl**  
  Either the outlet swirl of the upstream component will be used for the determination of the inlet swirl or the relative inlet flow angle $\beta_1$ or the degree of reaction (the latter only if total-to-static pressure ratio $\pi_{ts}$ is specified, see [global setup](#)).

**Initial default setting**

When creating a new design the initial default settings for some important properties are displayed in the panel **Initial default settings**. These settings are used in further design steps and can be modified by selecting the **Change settings** button. Of course these default settings can be modified manually in the appropriate design steps. See [Preferences: Impeller/ Stator settings](#) for more information.

**Information**

Some design point values are displayed in the right **Information** panel when selecting the page **Values** (see [Global setup](#)).
7.1.4.2 Parameters

On page Parameters one has to put in or to modify parameters resulting from approximation functions in dependence on specific speed \( n_q \) (see Approximation functions).

Parameter and efficiency values can be handled manually or can be switched to automatic update by the checkbox on top of the page. Then the default values are used always, even after design point modifications (see Global setup).

For details of how to handle the parameter edit fields please see Edit fields with empirical functions.
If the automatic mode is not selected the current default values can be specified by one of the following options:

- **globally** by the button on top of the page
- **regionally** by the default button within the **Parameters** or **Efficiency** region
- **individually** by the default button within the input field when selected

### Parameters

The panel **Parameters** allows defining alternative values in each case for the calculation of the following rotor main dimensions:

- suction diameter \( d_s \)
- rotor diameter \( d_1 \)
- inlet width \( b_1 \)

For details of how to handle the parameter edit fields please see [Edit fields with empirical functions](#).

The diameter coefficient is a parameter for the Balje diagram but not a design parameter. With its help suggestions for isentropic velocity ratio total-static \( \nu_{ts} \) as well as for the inlet width ratio \( b_1/d_1 \) can be get (only if total-to-static pressure ratio \( \pi_{ts} \) is specified, see [global setup](#)).

One of the following parameters has to be specified for the calculation of the rotor diameter \( d_1 \).
### Work coefficient $\psi$

($=$ pressure and head coefficient)

- Dimensionless expression of the specific enthalpy
  \[ \psi = \frac{\Delta h_{ls}}{u_{i}^2/2} \quad \text{and} \quad \psi = \frac{\Delta h}{u_{i}^2/2} \]
- Big $\rightarrow$ Small $d_{i}$
- Small $\rightarrow$ Big $d_{i}$
- Guideline $\sim 2$

### Flow coefficient $\varphi_{m}$

- Dimensionless mass flow
  \[ \varphi_{m} = \frac{c_{m1}}{u_{1}} \]
- In accordance to Cordier-Diagramm $^{[30]}$

### Tangential force coefficient

$\mathbf{c}_{1} = \frac{\psi}{\varphi_{m}}$

- Coefficient of a flow force pointing in tangential direction
  - 3 ... 4 Francis high-speed turbine
  - 4 ... 8 Normal-speed turbine
  - 8 ... 10 Low-speed turbine

### Coefficient ratio

$\mathbf{c}_{R} = \frac{\psi}{\varphi_{m}^2}$

- Ratio of work to the square of the meridional speed
  - 6 ... 10 Francis high-speed turbine
  - 10 ... 12 Normal-speed turbine
  - 12 ... 30 Low-speed turbine

### Isentropic velocity ratio total-static

$\mathbf{v}_{ts} = \frac{u}{c_{0}}$

- $u$: Peripheral velocity at inlet
- $c_{0}$: Spouting velocity
  \[ c_{0} = \sqrt{2 \cdot \Delta h_{ls}} \]

Defaults by Balje diagram $^{[36]}$ (only if total-to-static pressure ratio $\pi_{ts}$ is specified, see global setup) $^{[31]}$

Between the work coefficient $\psi$, the relative flow angle $\beta_{1}$ and the tangential force coefficient $\psi/\varphi_{m}$, there is the following relation:

At a relative flow angle of $\beta_{1} = 90^\circ$ the work coefficient becomes $\psi = 2$. In this case the work coefficient should not be chosen as a design parameter in the tab sheet Parameters. Otherwise one has no influence on the meridional flow coefficient and therefore meridional flow, see last equation.
One of the following parameters has to be specified for the calculation of the rotor inlet width $b_1$.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Specification</th>
</tr>
</thead>
<tbody>
<tr>
<td>Meridional flow coefficient</td>
<td>Default according to equation above:</td>
</tr>
<tr>
<td>$\varphi_m = \frac{c_m}{u}$</td>
<td>$\varphi_m = \left( \frac{1}{\psi} - \frac{1}{\psi^2} \right) \cot(\beta_i)$</td>
</tr>
<tr>
<td>Inlet width ratio $b_1/d_1$</td>
<td>Defaults by Balje diagram (only if total-to-static pressure ratio $\pi_{ts}$ is specified, see global setup)</td>
</tr>
</tbody>
</table>

For all further geometric variables guess values have to be given:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter ratio $d_2/d_1$</td>
<td>$\sim 0.5$</td>
</tr>
<tr>
<td>Meridional acceleration $c_{m2}/c_{m1}$</td>
<td>1.005..1.05</td>
</tr>
<tr>
<td>Meridional acceleration (suction side) $c_{mS}/c_{m2}$ or Diameter ratio $d_S/d_1$</td>
<td>1.005..1.05 or $\sim 0.7$</td>
</tr>
<tr>
<td>Diameter ratio $d_H/d_S$</td>
<td>$\sim 0.3$</td>
</tr>
</tbody>
</table>

There are three modes for the definition of the hub diameter $d_H$:

- Direct input in the tab sheet Dimensions (check box "Calculate hub diameter" deactivated)
- Combo box option "Diameter ratio $d_H/d_S$": automatic calculation with $d_H = d_H/d_S \times d_S$.
- Combo box option "Diameter ratio $d_2/d_1$": automatic calculation with $d_H = d_2/d_1 \times d_S$. Here the diameter ratio $d_H/d_S$ will be adjusted in a way that the guideline of the geometrical ratios will be met.
With option “diameter ratio $d_2/d_1$,” for the $d_3$-calculation the option “Diameter ratio $d_2/d_1$” is not available.

**Efficiency**

In the group **Efficiency** the following efficiencies need to be given:

**Design relevant**

- Rotor efficiency $\eta_{tt}$ (total-total: pressure ratio $\tau_{tt}$ specified in global setup) or
- Rotor efficiency $\eta_{ts}$ (total-static: pressure ratio $\tau_{ts}$ specified in global setup)

**Information only**

- Mechanical efficiency $\eta_m$

Internal and mechanical efficiency form the overall efficiency (coupling efficiency):

$$\eta_{tt} = \frac{P_D}{P_Q} = \eta_{tt} \eta_m$$

$P_Q$: (isentropic) Rotor power

$P_D$: Power output (coupling/ driving power)

The rotor efficiency (or blade efficiency) $\eta_{tt}$ describes the energy losses within the turbine caused by friction and vorticity. Friction losses mainly originate from shear stresses in boundary layers. Vorticity losses are caused by turbulence and on the other hand by changes of flow cross section and flow direction which may lead to secondary flow, flow separation, wake behind blades etc.. The rotor efficiency is the ratio between the actual specific work $\hat{Y}$ and the specific work at loss less transmission:

$$\eta_{tt} = \frac{\hat{Y}}{\tilde{Y}}$$

The mechanical efficiency mainly includes the friction losses in bearings and seals:

$$\eta_m = 1 - \frac{P_m}{P} \approx 0.95...0.995$$

(rising with impeller size)

If the check box **Use $\eta$ for main dimensions** is set, then main dimension calculation is done on the basis of $\Delta h = \Delta h_{is} \cdot \eta$. Otherwise $\Delta h_{is}$ - the isentropic specific enthalpy - is used.
Information

In the right panel of the tab sheet Parameter some variables are displayed for Information:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>Actual Power $P_D$</td>
<td>$P_D = P_Q \cdot \eta_{ttSt}$</td>
</tr>
<tr>
<td>Power loss $P_L$</td>
<td>$P_L = P_Q - P_D$</td>
</tr>
<tr>
<td>Flow $Q$</td>
<td>calculated with total density in the outlet: $Q_t = \frac{m}{\rho_{12}}$</td>
</tr>
<tr>
<td>Total pressure inlet $p_{t1}$</td>
<td>$p_{t1} = \pi p_{t2}$</td>
</tr>
<tr>
<td>Pressure ratio total-total</td>
<td>$\pi_{tt}$</td>
</tr>
<tr>
<td>Pressure ratio total-static</td>
<td>$\pi_{ts}$</td>
</tr>
<tr>
<td>Stage efficiency total-total</td>
<td>$\eta_{ttSt}$</td>
</tr>
<tr>
<td>Isentropic efficiency total-static</td>
<td>$\eta_{ts}$</td>
</tr>
<tr>
<td>Polytropic efficiency</td>
<td>$\eta_p = \frac{\kappa}{n - 1}$</td>
</tr>
</tbody>
</table>

In general for cost reasons single-stage & single-intake machines are preferred covering a range of about $10 < n_q < 400$. In exceptional cases it may become necessary to design a rotor for extremely low specific speed values ($n_q < 10$). These rotors are characterized by large rotor diameters and low rotor widths. The ratio of free flow cross section area to wetted surfaces becomes unfavorable and is causing high frictional losses. To prevent this one may increase either rotational speed $n$ or mass flow rate $\dot{m}$ if possible. An alternative solution could be the design of a multi-stage turbine reducing the pressure drop of a single-stage. If especially high specific speed values ($n_q > 400$) do occur one can reduce rotational speed $n$ or mass flow rate $\dot{m}$ if feasible. Another option would be to operate several single-stage turbines - having a lower $n_q$ - in parallel.
Please note: CFturbo® is preferably used between $10 < n_q < 150$ – radial and mixed-flow rotors.

7.1.4.3 Dimensions

In the panel **Shaft**, the required shaft diameter is computed.

→ **Shaft/ Hub**

The main dimensions of a rotor - suction diameter $d_{S}$, hub diameter $d_{H}$, rotor diameter $d_{r}$ and inlet width $b_{1}$ - can be seen on the tab sheet **Dimensions**. They can be recomputed by pressing the **Calculate** button within the panel **Main dimensions**. The computation is based on “Euler's Equation of Turbomachinery”, on the continuity equation and the relations for the velocity triangles as well as on the parameters and parameter ratios given in the tab sheets **Setup** and **Parameters**.

Individual main dimensions can be calculated separately using the button inside the value field.

One may accept the proposed values or can modify them slightly, e.g. to meet a certain normalized diameter.

In case the checkbox **Automatic** is activated a new calculation will accomplished after any change of parameter. Then the manual alteration of the main dimensions is not possible.
Neighboring components

In specific cases the dimensions of the neighboring components at inlet and/or outlet can be used to get exactly matching geometry.

This feature is available only for explicitly uncoupled components or side-by-side impellers.

Information
In the right panel of any tab sheet an information panel is situated, which holds the computed variables in accordance to the actual state of design, the resulting Meridional section as well as the Cordier-Diagramm with the location of the best point. These three sections can be chosen by the appropriate soft buttons in the heading.

In the information section of the tab sheet Dimensions the following variables are displayed for Information:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>Work coefficient</td>
<td>$\psi = \frac{\Delta h_{vis} \cdot \eta_{It}}{u_i^2 / 2}$</td>
</tr>
<tr>
<td>Flow coefficient</td>
<td>$\varphi = \frac{Q}{\pi / 4 \cdot d_1^2 \cdot u_1}$</td>
</tr>
<tr>
<td>Meridional flow coefficient</td>
<td>$\varphi_m = \frac{Q_1}{\pi \cdot d_1 \cdot b_1 \cdot u_1} = \frac{c_m}{u_1}$</td>
</tr>
<tr>
<td>Specific diameter</td>
<td>$\delta = 1.054 \cdot d_1 \frac{\Delta h_{vis}^{1/4}}{Q^{1/2}}$</td>
</tr>
<tr>
<td>Specific speed $n_q$</td>
<td>$n_q = n \left[ \min^{-1} \right] = \sqrt[3]{\frac{Q \cdot \frac{m^3}{s}}{\frac{m^2}{s^2} \cdot \frac{1}{g}}}$ points to machine type and general shape of rotor</td>
</tr>
<tr>
<td>Inlet pressure, density and temperature</td>
<td>$p_1, T_1, \rho_1, p_{1t}, T_{1t}, \rho_{1t}$ static and total values</td>
</tr>
<tr>
<td>Inlet velocities</td>
<td>$c_1, c_{u1}, c_{m1}, w_1$</td>
</tr>
<tr>
<td>Peripheral speed at inlet</td>
<td>$u_i = \pi \cdot d_1 \cdot n$</td>
</tr>
<tr>
<td>Machine Mach Number</td>
<td>$M_t = \frac{u_1}{a_1}$</td>
</tr>
<tr>
<td>Blade width at inlet</td>
<td></td>
</tr>
<tr>
<td>Outlet pressure, density and temperature</td>
<td>$p_2, T_2, \rho_2, p_{2t}, T_{2t}, \rho_{2t}$ static and total values</td>
</tr>
<tr>
<td>Outlet velocities</td>
<td>$c_2, c_{u2}, c_{m2}, w_2$</td>
</tr>
<tr>
<td>Parameter</td>
<td>Formula</td>
</tr>
<tr>
<td>-----------------------------------------------</td>
<td>----------------------------------------------</td>
</tr>
<tr>
<td>Peripheral speed at outlet</td>
<td>( u_2 = \pi \cdot d_2 \cdot n )</td>
</tr>
<tr>
<td>Outlet Mach Number</td>
<td>( M_2 = \frac{c_2}{a_2} )</td>
</tr>
<tr>
<td>Mean diameter at outlet</td>
<td>( d_2 = \left( \frac{d_2}{d_1} \right) d_1 )</td>
</tr>
<tr>
<td>Width at outlet</td>
<td>( b_2 = 1.025 \frac{\dot{m}}{\pi d_2 \cdot c_{m2} \cdot \rho_2} )</td>
</tr>
<tr>
<td>Ratio Width-diameter at inlet</td>
<td>( b_2/d_1 )</td>
</tr>
<tr>
<td>Diameter ratio</td>
<td>( d_2/d_{2\text{min}} ) with:</td>
</tr>
<tr>
<td></td>
<td>( d_{2\text{min}} = \sqrt{\frac{1}{2} \left( d_{s2}^2 - d_{H2}^2 \right)} )</td>
</tr>
<tr>
<td>Ratio radius-width at outlet</td>
<td>( \frac{r_{s2} - r_{H2}}{b_2} = \frac{d_{s2} - d_{H2}}{2 \cdot b_2} )</td>
</tr>
<tr>
<td>Isentropic velocity ratio</td>
<td>( v_{ts} = \frac{u_1}{\sqrt{2\Delta h_{ts}}} )</td>
</tr>
</tbody>
</table>

The guidelines given in the last column of the last three rows, should be matched within the design.

It might be that for the chosen configuration of global setup and main dimensions a reasonable thermodynamic state cannot be calculated. This may be the case if e.g. the mass flow is too high for the chosen cross sections. Then a warning is generated.

The Meridional preview is based on the main dimensions designed until this point.
The **Cordier** diagram can be used for checking the impeller diameter $d_2$.

See [Cordier](#).
The Velocity triangles are the result of a mid-span calculation and are based on the design point and the main dimensions.
Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thermodynamic state cannot be calculated at inlet/outlet. Possible reason: choked flow. Consider change of main dimensions or global setup.</td>
<td>Increase the dimensions (width, diameters etc.) or change the Global setup (e.g. decrease mass flow, increase pressure, decrease temperature).</td>
</tr>
<tr>
<td>The dimensions might be too tight for the specified mass flow and inlet conditions. In other words the mass flow is higher than the choked mass flow rate for the particular inlet condition and the respective cross section.</td>
<td>Increase the dimensions (width, diameters etc.) or change the Global setup (e.g. decrease mass flow, increase pressure, decrease temperature).</td>
</tr>
<tr>
<td>β1 differs by more than 25° from the flow imposed by the upstream component.</td>
<td></td>
</tr>
<tr>
<td>Problem</td>
<td>Possible solutions</td>
</tr>
<tr>
<td>------------------------------------------------------------------------</td>
<td>-------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>( \beta_1 ) differs by more than 25° from ( \beta_1 \text{mean} ).</td>
<td></td>
</tr>
<tr>
<td>( \beta_1 ) differs by more than 25° from the flow imposed by the degree of reaction.</td>
<td>Change the dimensions (width, diameters etc.) or change the Global setup (e.g. mass flow, pressure ratio) or change inlet swirl in the Setup by either of parameters given on the left hand side.</td>
</tr>
</tbody>
</table>

The dimensions and the design point yield an estimated average relative inlet flow angle \( \beta_1 \) that differs by more than 25° from the relative inlet flow angle imposed by the either of

- swirl of the upstream component
- \( \beta_1 \text{mean} \)
- degree of reaction \( R \)

see Setup.

7.1.5 Axial Gas Turbine / Compressor

? Rotor | Main dimensions

The Main Dimensions menu item is used to define main dimensions of the axial rotor. Main Dimensions are forming the most important basis for all following design steps.

The real flow in the impeller/rotor is turbulent and three-dimensional. Secondary flows, separation and reattachment in boundary layers, transient recirculation areas and other features may occur. Nevertheless it is useful - and it is common practice in the turbine design theory - to simplify the realistic flow applying representative streamlines for the first design approach.

Details

- Setup
- Parameters Gas Turbine
Employing 1D-streamline theory the following cross sections are significant in particular: just before leading edge (index 0), at the beginning (index 1) and at the end of the blade (index 2), behind the trailing edge (index 3) and at the outlet (index 4).

**Possible warnings**

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solution</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet boundary conditions do not fit previous component.</td>
<td>Adjust the stator geometry (dimensions or blade angles) or change the Global setup (e.g. decrease mass flow).</td>
</tr>
</tbody>
</table>

If a stator is located prior the impeller or rotor, boundary conditions for the determination of the main dimensions will be calculated on the basis of the thermodynamic state at its outlet. In case of an undefined thermodynamic state at this location the inlet boundary conditions (i.e. total pressure and temperature) will be taken from the global setup.
7.1.5.1 Setup

On page **Setup** one can specify some basic settings.

![Setup screen capture](image)

**General**

- **Manual dimensioning**
  
  In manual dimensioning mode the main dimensions and blade angles are not calculated by CFturbo. All these values are user-defined input values.

- **Unshrouded**
  
  Design a shrouded (closed) or unshrouded (open) impeller. For an unshrouded impeller you have to define the **tip clearance**, optional different values at inlet and outlet.

- **Material density**
  
  The material density of the impeller is an informational value that is not relevant for the aerodynamic design but is used for the calculation of moments of inertia. Density values can be directly entered or selected from a list by pressing the settings button next to the input area.
• **Impeller type**  
  Turbine only: select either **Standard** or **Rocket engine** rotor type.

• **Inflow swirl**  
  Either the outlet swirl of the upstream component will be used for the determination of the inlet swirl or the absolute inlet flow angle or the degree of reaction (the latter only if total-to-static pressure ratio \( \pi_{ts} \) is specified, see global setup). If the absolute inlet flow angle was chosen it is to be specified whether the inlet flow is sub or super sonic.

• **Blade design mode**  
  Currently design mode **Airfoil** is only available for compressors whereas **Mean line** (using Euler's equation on mean lines) is available for compressors and turbines.

**Multi stage**

For a multi stage design the panel **Multi stage options** is available.

**Initial default setting**

When creating a new design the initial default settings for some important properties are displayed in the panel **Initial default settings**. These settings are used in further design steps and can be modified by selecting the **Change settings** button. Of course these default settings can be modified manually in the appropriate design steps. See **Preferences: Impeller/ Stator settings** for more information.

**Information**

Some design point values are displayed in the right **Information** panel when selecting the page **Values** (see **Global setup**).
7.1.5.2 Parameters Gas Turbine

On page **Parameters** one has to put in or to modify parameters resulting from approximation functions in dependence on specific speed $n_q$ (see **Approximation functions**).
For details of how to handle the parameter edit fields please see Edit fields with empirical functions.

Parameter and efficiency values can be handled manually or can be switched to automatic update by the checkbox on top of the page. Then the default values are used always, even after design point modifications (see Global setup).

If the automatic mode is not selected the current default values can be specified by one of the following options:

- globally by the button on top of the page
- regionally by the default button within the Parameters or Efficiency region
Parameters

The panel **Parameters** allows defining alternative parameters in each case for the calculation of the following impeller diameters:

<table>
<thead>
<tr>
<th>Inlet</th>
<th>Outlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>$d_{S1}$, $d_{H1}$</td>
<td>$d_{S2}$, $d_{H2}$</td>
</tr>
</tbody>
</table>

The diameter coefficient is a parameter for the **Balje diagram** but not a design parameter. With its help suggestions for isentropic velocity ratio total-static $\nu_{ts}$ as well as for the inlet width ratio $b_1/d_1$ or the diameter ratio $d_{H}/d_{S}$ can be get (only if total-to-static pressure ratio $\pi_{ts}$ is specified, see **global setup**).

One of the following parameters has to be specified for the calculation of the mean inlet diameter $0.5(d_{S1}+d_{H1})$.

| Isentropic velocity ratio total-total | $u$: Peripheral velocity at inlet |
Impeller

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$v_{it} = \frac{u}{c_{t0}}$</td>
<td>$c_{t0}$: spouting velocity&lt;br&gt;$c_{t0} = \sqrt{2 \cdot \Delta h_{tt}}$&lt;br&gt;(only if total-to-total pressure ratio $\pi_{tt}$ is specified, see global setup)</td>
</tr>
<tr>
<td>Isentropic velocity ratio total-static $v_{ts} = \frac{u}{c_0}$</td>
<td>$u$: Peripheral velocity at inlet&lt;br&gt;$c_0$: spouting velocity&lt;br&gt;$c_0 = \sqrt{2 \cdot \Delta h_{ts}}$&lt;br&gt;Defaults by Balje diagram (only if total-to-static pressure ratio $\pi_{ts}$ is specified, see global setup)</td>
</tr>
</tbody>
</table>

One of the following parameters has to be specified for the calculation of the rotor inlet width $b_1$.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter ratio $d_H/d_s$</td>
<td>Inlet hub diameter $d_{H1}$&lt;br&gt;$d_{H1} = \frac{d_H}{d_s} d_{S1}$</td>
</tr>
<tr>
<td>Inlet width ratio $h/d_{S1}$</td>
<td>$h$: inlet width&lt;br&gt;Defaults by Balje diagram (only if total-to-static pressure ratio $\pi_{ts}$ is specified, see global setup)</td>
</tr>
</tbody>
</table>

The outlet section can be calculated with:

| Meridional velocity ratio $c_{m2}/c_{m1}$ | 0.9..1.1<br>- strictly coaxial $d_{H2} = d_{H1}$ and $d_{S2} = d_{S1}$<br>- coaxial @Hub $d_{H2} = d_{H1}$<br>- coaxial @Mid-span $d_{m2} = d_{m1}$<br>- coaxial @Shroudd $d_{S2} = d_{S1}$ |

**Efficiency**

In the group **Efficiency** the following efficiencies need to be given:

**Design relevant**
• Rotor efficiency \( \eta_{tt} \) (total-total: pressure ratio \( \tau_{tt} \) specified in global setup) or

• Rotor efficiency \( \eta_{ts} \) (total-static: pressure ratio \( \tau_{ts} \) specified in global setup)

\[
\eta_{ts} = \frac{\Delta h_{ts}}{\Delta h_{sis}}
\]

**Information only**

• Mechanical efficiency \( \eta_m \)

Internal and mechanical efficiency form the overall efficiency (coupling efficiency):

\[
\eta_{it|si} = \frac{P_D}{P_Q} = \eta_{it} \eta_{im}
\]

\( P_Q \): (isentropic) Rotor power  
\( P_D \): Power output (coupling/ driving power)

The rotor efficiency (or blade efficiency) \( \eta_{tt} \) describes the energy losses within the turbine caused by friction and vorticity. Friction losses mainly originate from shear stresses in boundary layers. Vorticity losses are caused by turbulence and on the other hand by changes of flow cross section and flow direction which may lead to secondary flow, flow separation, wake behind blades etc.. The rotor efficiency is the ratio between the actual specific enthalpy difference and the ideal (isentropic) specific enthalpy difference at loss less transmission:

\[
\eta_{tt} = \frac{\Delta h_{ts}}{\Delta h_{sis}}
\]

The mechanical efficiency mainly includes the friction losses in bearings and seals:

\[
\eta_m = 1 - \frac{P_L}{P} \approx 0.95...0.995,
\]

rising with impeller size.

**Information**

In the right panel of the tab sheet **Parameter** some variables are displayed for **Information**:

| actual Power \( P_D \) | \( P_D = P_Q \cdot \eta_{it|si} \) |
|-------------------------|-----------------------------|
| Power loss \( P_L \)    | \( P_L = P_Q - P_D \)        |
Flow $Q_1$ calculated with total density in the outlet:

$$Q_1 = \frac{\dot{m}}{\rho_{12}}$$

Pressure ratio total-total $\pi_{tt}$

Pressure ratio total-static $\pi_{ts}$

Efficiency total-total $\eta_{tt}$

Efficiency total-static $\eta_{ts}$

Polytropic efficiency

$$\eta_p = \frac{\kappa - 1}{\kappa} \frac{n}{n-1}$$

(n .. polytropic exponent
$\kappa$ .. isentropic exponent)

In general for cost reasons single-stage & single-intake machines are preferred covering a range of about $10 < nq < 400$. If especially high specific speed values ($nq > 400$) do occur one can reduce rotational speed $n$ or mass flow rate $\dot{m}$ if feasible. Another option would be to operate several single-stage turbines - having a lower $nq$ - in parallel.
7.1.5.3 Parameters Compressor

On page Parameters put in or modify parameters resulting from approximation functions in dependence on specific speed \( n_q \) or other parameters. See Approximation functions.

Parameter and efficiency values can be handled manually or can be switched to automatic update by the checkbox on top of the page. Then the default values are used always, even after design point modifications (see Global setup).

If the automatic mode is not selected the current default values can be specified by one of the following options:

- Set default globally by the button on top of the page
regionally by the default button within the **Parameters** or **Efficiency** region

individually by the default button within the input field when selected

### Parameters

The panel **Parameters** allows defining alternative parameters in each case for the calculation of the following impeller diameters:

<table>
<thead>
<tr>
<th>Inlet</th>
<th>outlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>(d_{S1}, d_{H1})</td>
<td>(d_{S2}, d_{H2})</td>
</tr>
</tbody>
</table>

For \(d_{S2}\) calculation

<table>
<thead>
<tr>
<th>Work coefficient (\psi)</th>
<th>* dimensionless expression for the specific energy:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.1 ...0.6 axial impeller</td>
</tr>
<tr>
<td></td>
<td>* high (\rightarrow) small (d_{S2}); flat characteristic curve</td>
</tr>
<tr>
<td></td>
<td>low (\rightarrow) high (d_{S2}); steep characteristic curve</td>
</tr>
</tbody>
</table>
Diameter coefficient $\delta$ according to Cordier diagram (see Dimensions).

For $d_{H2}$ calculation

<table>
<thead>
<tr>
<th>Diameter ratio $d_{H2}/d_{S2}$</th>
<th>Can be estimated under assumption $\beta_2 = 90^\circ$ @ hub and $c_u \cdot u = \text{const.}$ by: $d_{H2} = d_{S2} \dfrac{\sqrt{\psi}}{\sqrt{2}}$</th>
</tr>
</thead>
</table>

For $d_{S1}/d_{H1}$-calculation

<table>
<thead>
<tr>
<th>Meridional velocity ratio $c_{m2}/c_{m1}$</th>
<th>$\dfrac{c_{m2}}{c_{m1}} = 0.9 \ldots 1.1$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter ratio $d_{H1}/d_{S1}$</td>
<td>$\dfrac{d_{H1}}{d_{S1}} = 0.4 \ldots 0.9$</td>
</tr>
<tr>
<td>strictly axial $d_{H2} = d_{H1}$ and $d_{S2} = d_{S1}$</td>
<td></td>
</tr>
<tr>
<td>const. hub $d_{H2} = d_{H1}$</td>
<td></td>
</tr>
<tr>
<td>const. mid $d_{M2} = d_{M1}$</td>
<td></td>
</tr>
<tr>
<td>const. shroud $d_{S2} = d_{S1}$</td>
<td></td>
</tr>
</tbody>
</table>

### Efficiency

In panel Efficiency you have to specify several efficiencies. You have to distinguish between design relevant efficiencies and efficiencies used for information only:

**Design relevant**

- total-total efficiency $\eta_{tt}$

**Information only**

- mechanical efficiency $\eta_m$
- motor efficiency $\eta_{mot}$

Impeller and mechanical efficiency form the overall efficiency (coupling efficiency) of the stage $\eta_{St}$. 
When considering motor losses additionally the overall efficiency of the stage incl. motor $\eta_{\text{St}}^*$ is defined.

$$\eta_{\text{St}} = \frac{P_\Omega}{P_D} = \eta_e \eta_m$$

$P_\Omega$: power output, see above

$P_D$: mechanical power demand (coupling/ driving power)

$$\eta_{\text{St}}^* = \frac{P_\Omega}{P_{\text{el}}} = \eta_{\text{St}} \eta_{\text{mot}}$$

$P_{\text{el}}$: electrical power demand of motor

The following summary illustrates the single efficiencies and their classification:

<table>
<thead>
<tr>
<th>classification</th>
<th>efficiencies</th>
<th>Relevant for impeller design</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stage</td>
<td>$\eta_{\text{tt}}$ total-total</td>
<td>yes: for energy transmission</td>
</tr>
<tr>
<td></td>
<td>$\eta_m$ mechanical</td>
<td>no: for overall information only</td>
</tr>
<tr>
<td>stage incl. motor</td>
<td>electrical $\eta_{\text{mot}}$ motor</td>
<td></td>
</tr>
</tbody>
</table>

The obtainable overall efficiency correlates to specific speed and to the size and the type of the impeller as well as to special design features like bypass installations and auxiliary aggregates. Efficiencies calculated by approximation functions are representing the theoretical reachable values and they should be corrected by the user if more information about the impeller or the whole pump are available.

The total-total efficiency describe the energy losses within the compressor. Friction losses mainly originate from shear stresses in boundary layers. Vorticity losses are caused by turbulence and on the other hand by changes of flow cross section and flow direction which may lead to secondary flow, flow separation, wake behind blades etc. This impeller efficiency is the ratio between the ideal (isentropic) specific enthalpy difference at loss less transmission and the actual specific enthalpy difference:

The volumetric efficiency is a quantity for the deviation of effective flow rate $Q$ from total flow rate inside the impeller:
The mechanical efficiency mainly includes the friction losses in bearings and seals:

\[ \eta = 1 - \frac{P_m}{P} \approx 0.95 \ldots 0.995 \] (rising with impeller size)

Total-total and volumetric efficiency are most important for the impeller dimensioning because of their influence to \( \tilde{\dot{V}} \) and/or \( \tilde{\dot{Q}} \). The mechanical efficiency is affecting only the required driving power of the machine.

If the check box "Use \( \eta \) for main dimensions" is set, then main dimension calculation is done on the basis of \( Y_{\text{eff}} = 0.5(Y/\eta + Y) \). Otherwise \( Y \) - specific work without losses - is used.

### Information

In the right area of the register **Parameter** you can find again some calculated values for information:

<table>
<thead>
<tr>
<th></th>
<th>[ p_d = \frac{P_Q}{\eta_{\text{St}}} ]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required driving power</td>
<td>[ p_L = p_d - p_Q = p_d \left( 1 - \eta_{\text{St}} \right) ]</td>
</tr>
<tr>
<td>Power loss</td>
<td>[ \eta_t = \eta_{tt} ]</td>
</tr>
<tr>
<td>Internal efficiency</td>
<td>[ \eta_{\text{St}} = \frac{P_Q}{P_D} = \eta_t \eta_{\text{m}} ]</td>
</tr>
<tr>
<td>Stage efficiency</td>
<td>[ \eta_{\text{St}} = \frac{P_Q}{P_{\text{el}}} = \eta_{\text{St}} \eta_{\text{mot}} ]</td>
</tr>
</tbody>
</table>

### 7.1.5.4 Dimensions

The main dimensions of a rotor - inlet diameter \( d_{S1} \) and \( d_{H1} \) and outlet diameter \( d_{S2} \) and \( d_{H2} \) - can be seen on **Main dimensions** panel. They can be recomputed
by pressing the **Calculate**-button. The computation is based on “Euler's Equation of Turbomachinery”, on the continuity equation and the relations for the velocity triangles as well as on the parameters and parameter ratios given in the tab sheets **Setup** and **Parameters**.

Individual main dimensions can be calculated separately using the button inside the value field.

You may accept the proposed values or you can modify them slightly, e.g. to meet a certain normalized diameter.

In case the checkbox **Automatic** is activated a new calculation will accomplished after any change of parameter. Then the manual alteration of the main dimensions is not possible.
Neighboring components

In specific cases the dimensions of the neighboring components at inlet and/ or outlet can be used to get exactly matching geometry.

This feature is available only for explicitly uncoupled components or side-by-side impellers.

Information

In the right panel of any tab sheet an information panel is situated, which holds the computed variables in accordance to the actual state of design, the resulting Meridional section as well as the Cordier-Diagramm with the location of the best point. These three sections can be chosen by the appropriate soft buttons in the heading.
In the information section of the tab sheet **Dimensions** the following variables are displayed for **Information**:

<table>
<thead>
<tr>
<th>Equation</th>
<th>variables</th>
</tr>
</thead>
<tbody>
<tr>
<td>Work coefficient</td>
<td>$\psi = \frac{\Delta h_{tiss} \cdot \eta_{tt}}{u_1^2 / 2}$</td>
</tr>
<tr>
<td>Flow coefficient</td>
<td>$\varphi = \frac{Q_{t1}}{\pi / 4 \cdot d_1^2 u_1}$</td>
</tr>
<tr>
<td>Meridional flow coefficient</td>
<td>$\varphi_m = \frac{Q_1}{4 \left( d_{iss}^2 - d_{it}^2 \right) u_1}$</td>
</tr>
<tr>
<td>Specific diameter</td>
<td>$\delta = 1.054 \cdot d_{iss} \cdot \frac{\Delta h_{tiss}^{1/4}}{Q_{t1}^{1/2}}$</td>
</tr>
<tr>
<td>Inlet pressure, density and temperature</td>
<td>$p_1, T_1, \rho_1, p_{1t}, T_{1t}, \rho_{1t}$ static and total values</td>
</tr>
<tr>
<td>Inlet velocities</td>
<td>$c_{1}, c_{u1}, c_{m1}, w_{1}, u_{1}$</td>
</tr>
<tr>
<td>Absolute Mach Number (inlet &amp; outlet)</td>
<td>$M_c = \frac{c}{a}$</td>
</tr>
<tr>
<td>Relative Mach Number (inlet &amp; outlet)</td>
<td>$M_w = \frac{w}{a}$</td>
</tr>
<tr>
<td>Machine tip Mach Number (inlet &amp; outlet)</td>
<td>$M u S = \frac{u S}{a}$</td>
</tr>
<tr>
<td>Outlet pressure, density and temperature</td>
<td>$p_2, T_2, \rho_2, p_{2t}, T_{2t}, \rho_{2t}$ static and total values</td>
</tr>
<tr>
<td>Outlet velocities</td>
<td>$c_{2}, c_{u2}, c_{m2}, w_{2}, u_{2}$</td>
</tr>
<tr>
<td>Isentropic velocity ratio</td>
<td>$v_{Is} = \frac{u_1}{\sqrt{2 \Delta h_{tiss}}}$ only turbines</td>
</tr>
<tr>
<td>Pressure rise coefficient</td>
<td>$c_p = \frac{p_2 - p_1}{1/2 \rho_1 w_1^2}$ only compressor</td>
</tr>
<tr>
<td>Suction capacity</td>
<td>$s_p = \frac{\dot{m} \cdot \omega_2}{\rho_1 a_{1t}^3}$ only compressor</td>
</tr>
</tbody>
</table>
The Meridional preview is based on the main dimensions designed until this point.

The Cordier diagram can be used for checking the impeller diameter $d_2$.

See Cordier [ref].
The **Velocity triangles** are the result of a mid-span calculation and are based on the design point and the main dimensions.
Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thermodynamic state cannot be calculated at inlet/outlet. Possible reason: choked flow. The cu-specification will not be available.</td>
<td>Increase the dimensions (diameters etc.) or change the Global setup (e.g. decrease mass flow, increase pressure, decrease temperature).</td>
</tr>
<tr>
<td>The dimensions might be too tight for the specified mass flow and inlet conditions. In other words the mass flow is higher than the choked mass flow rate for the particular inlet conditions and the respective cross section.</td>
<td></td>
</tr>
</tbody>
</table>
7.1.6  Francis Turbine

The Main Dimensions menu item is used to define main dimensions of the runner. Main Dimensions are forming the most important basis for all following design steps.

The real flow in a Francis turbine runner is turbulent and three-dimensional. Secondary flows, separation and reattachment in boundary layers, transient recirculation areas and other features may occur. Nevertheless it is useful - and it is common practice in the turbine design theory - to simplify the realistic flow applying representative streamlines for the first design approach.

Employing 1D-streamline theory the following cross sections are significant in particular: area just before leading edge (index 0), at the beginning (index 1) and at the end of the blade (index 2), behind the trailing edge (index 3) and at the outlet (index 4).

Details
- Setup
- Parameters
- Dimensions

The design of the main dimensions has to be made in a strict order. This will be secured by the following: One step within the design has to be finished completely before the next can be accomplished. That is to say, the changeability of a tab sheet will be disabled by CFturbo until all necessary parameters have been specified.
7.1.6.1 Setup

On page **Setup** one can specify some basic settings.

### General

- **Manual dimensioning**
  In manual dimensioning mode the main dimensions and blade angles are not calculated by CFturbo. All these values are user-defined input values.

- **Unshrouded**
  Design a shrouded (closed) or unshrouded (open) runner.
  For an unshrouded runner one must define the **tip clearance**, optional different values at inlet and outlet.

- **Splitter blades**
  Design the runner with or without splitter blades.

- **Material density**
  The material density of the impeller is an informational value that is not relevant for the hydrodynamic design but is used for the calculation of moments of inertia. Density values can
be directly entered or selected from a list by pressing the settings button next to the input area.

- **Inflow swirl**
  For the determination of the inlet swirl either of the following options can be used:
  - outlet swirl of the upstream component,
  - absolute inlet flow angle $\alpha_1$,
  - relative inlet flow angle $\beta_1$,
  - outlet swirl velocity $c_{u2}$ in accordance to the Euler equation: $g \cdot H = c_{u1} \cdot u_1 - c_{u2} \cdot u_2$.

**Initial default setting**

When creating a new design the initial default settings for some important properties are displayed in the panel **Initial default settings**. These settings are used in further design steps and can be modified by selecting the **Change settings** button. Of course these default settings can be modified manually in the appropriate design steps. See **Preferences: Impeller/ Stator settings** for more information.

**Information**

Some design point values are displayed in the right **Information** panel when selecting the page **Values** (see **Global setup**).
7.1.6.2 Parameters

On page **Parameters** one has to put in or to modify parameters resulting from approximation functions in dependence on specific speed \( n_q \) (see **Approximation functions**).

Parameter and efficiency values can be handled manually or can be switched to automatic update by the checkbox on top of the page. Then the default values are used always, even after design point modifications (see **Global setup**).

If the automatic mode is not selected the current default values can be specified by one of the following options:

- **Set default** globally by the button on top of the page
regionally by the default button within the **Parameters** or **Efficiency** region

individually by the default button within the input field when selected

---

**Parameters**

The panel **Parameters** allows defining alternative parameters in each case for the calculation of the following impeller diameters:

<table>
<thead>
<tr>
<th>Inlet</th>
<th>Outlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>$d_{S1}$, $d_{H1}$, $b_{1}$</td>
<td>$d_{S2}$, $d_{H2}$</td>
</tr>
</tbody>
</table>

---

For $d_{S1}$-calculation
Work coefficient $\psi$

- dimensionless expression for the specific energy:
  $\psi = Y/\left(\frac{u_Y^2}{2}\right)$ and $\psi = Y_{\text{eff}}/\left(\frac{u_Y^2}{2}\right)$

For $d_{H1}$ calculation

| Diameter ratio $d_{H1}/d_{S1}$ | $d_{H1}/d_{S1} = 0.4...1.0$ |

For $d_{S2}$ calculation

| Diameter ratio $d_{S2}/d_{S1}$ | $d_{S2}/d_{S1} = 0.5...1.1$ |

For $b_1$ calculation

| width ratio $b_1/d_{S1}$ | $b_1/d_{S1} = 0.01...0.35$ |

**Efficiency**

In panel **Efficiency** you have to specify several efficiencies. You have to distinguish between design relevant efficiencies and efficiencies used for information only:

**Design relevant**

- hydraulic efficiency $\eta_h$
- volumetric efficiency $\eta_v$

**Information only**

- mechanical efficiency $\eta_m$

The losses resulting in energy dissipation from the fluid form the **internal efficiency**.

Internal and mechanical efficiency form the overall efficiency (coupling efficiency) of the stage $\eta_{St}$.
The following summary illustrates the single efficiencies and their classification:

<table>
<thead>
<tr>
<th>classification</th>
<th>efficiencies</th>
<th>Relevant for impeller design</th>
</tr>
</thead>
<tbody>
<tr>
<td>stage: internal</td>
<td>( \eta_h^{+} )</td>
<td>additional hydraulic</td>
</tr>
<tr>
<td></td>
<td>( \eta_h )</td>
<td>hydraulic</td>
</tr>
<tr>
<td></td>
<td>( \eta_v )</td>
<td>volumetric</td>
</tr>
<tr>
<td></td>
<td>( \eta_m )</td>
<td>mechanical</td>
</tr>
</tbody>
</table>

The obtainable overall efficiency correlates to specific speed and to the size and the type of the runner as well as to special design features like bypass installations and auxiliary aggregates. Efficiencies calculated by approximation functions are representing the theoretical reachable values and they should be corrected by the user if more information about the impeller or the whole turbine are available.

The hydraulic efficiency (or blade efficiency) describe the energy losses within the turbine caused by friction and vorticity. Friction losses mainly originate from shear stresses in boundary layers. Vorticity losses are caused by turbulence and on the other hand by changes of flow cross section and flow direction which may lead to secondary flow, flow separation, wake behind blades etc.

The volumetric efficiency is the ratio of the flow \( \dot{Q} \) striking the runner and the flow through the turbine \( Q \):

\[
\eta_v = \frac{\dot{Q}}{Q}
\]

The mechanical efficiency mainly includes the friction losses in bearings and seals:

\[
\eta_m = 1 - \frac{P_m}{P}
\]

Hydraulic and volumetric efficiency are most important for the runner dimensioning because of their influence to \( \dot{Y} \) and/or \( \dot{Q} \). The mechanical efficiency is affecting only the actual output power of the machine.
**Information**

In the right area of the register **Parameter** you can find again some calculated values for information:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>Torque</td>
<td>$M = \frac{P_Q}{\omega}$</td>
</tr>
<tr>
<td>Actual power output</td>
<td>$P_D = P_Q \cdot \eta_{St}$</td>
</tr>
<tr>
<td>Power loss</td>
<td>$P_L = P_O - P_D$</td>
</tr>
<tr>
<td>Internal efficiency</td>
<td>$\eta_I = \eta_h \cdot \eta_v \cdot \eta_h^+$</td>
</tr>
<tr>
<td>Stage efficiency</td>
<td>$\eta_{St} = \frac{P_D}{P_O} = \eta_h \cdot \eta_m$</td>
</tr>
</tbody>
</table>

### 7.1.6.3 Dimensions

On page **Dimensions**, panel **Shaft/ hub**, the required shaft diameter is computed and the hub diameter is determined by the user.

>

The main dimensions of a designed runner - inlet and outlet diameters $d_{S1}$, $d_{H1}$, and $d_{S2}$, $d_{H2}$, and inlet width $b_1$ - can be seen on **Main dimensions** panel. They can be recomputed by pressing the **Calculate**-button. The computation is based on "Euler's Equation of Turbomachinery", on the continuity equation and the relations for the velocity triangles as well as on the parameters and parameter ratios given in the tab sheets **Setup** and **Parameters**.

Individual main dimensions can be calculated separately using the button inside the value field.

You may accept the proposed values or you can modify them slightly, e.g. to meet a certain normalized diameter.
In case the checkbox **Automatic** is activated a new calculation will accomplished after any change of parameter. Then the manual alteration of the main dimensions is not possible.

**Neighboring components**

In specific cases the dimensions of the neighboring components at inlet and/ or outlet can be used to get exactly matching geometry.

This feature is available only for explicitly **uncoupled** components or side-by-side impellers.

**Information**

In the right panel of any tab sheet an **information** panel is situated, which holds the computed variables in accordance to the actual state of design, the resulting **Meridional section** as well as the **Cordier-Diagram** with the location of the best point. These three sections can be chosen by the appropriate soft buttons in the heading.
In the **Value** section the following variables are displayed for information which result from calculated or determined main dimensions:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>Work coefficient</td>
<td>$\psi = \frac{Y}{u_i^2/2}$</td>
</tr>
<tr>
<td>Flow coefficient</td>
<td>$\varphi_t = \frac{Q}{\pi/4 \cdot d_1^2 \cdot u_1}$</td>
</tr>
<tr>
<td>Meridional flow coefficient</td>
<td>$\varphi_m = \frac{c_{m1}}{u_1}$</td>
</tr>
<tr>
<td>Specific diameter</td>
<td>$\delta = \frac{\psi^{1/4}}{\varphi_t^{1/2}} = 1.05d_1 \left( \frac{Y}{Q^2} \right)^{1/4}$</td>
</tr>
<tr>
<td>Average inlet velocity</td>
<td>$\bar{c}<em>{m1} = \frac{Q/\eta</em>\nu}{\pi/4 \cdot (d_{S1}^2 - d_{H1}^2)}$</td>
</tr>
<tr>
<td>Average inlet velocity (net)</td>
<td>$\bar{c}<em>{m1}^* = \frac{Q}{\pi/4 \cdot (d</em>{S1}^2 - d_{H1}^2)}$</td>
</tr>
<tr>
<td>Average outlet velocity</td>
<td>$\bar{c}<em>{m2} = \frac{Q/\eta</em>\nu}{\pi/4 \cdot (d_{S2}^2 - d_{H2}^2)}$</td>
</tr>
<tr>
<td>Average outlet velocity (net)</td>
<td>$\bar{c}<em>{m2}^* = \frac{Q}{\pi/4 \cdot (d</em>{S2}^2 - d_{H2}^2)}$</td>
</tr>
<tr>
<td>Outlet width ratio</td>
<td>$b_2/d_2$</td>
</tr>
<tr>
<td>Meridional velocity ratio</td>
<td>$\bar{c}<em>{m2}/\bar{c}</em>{m1}$</td>
</tr>
</tbody>
</table>

The *Meridional preview* is until now based on the main dimensions only.
The Cordier diagram can be used for checking the impeller diameter $d_z$.

See Cordier for more information.
The Velocity triangles are the result of a mid-span calculation and are based on the design point and the main dimensions.
7.1.7 Kaplan Turbine

? Runner | Main dimensions

The Main Dimensions menu item is used to define main dimensions of the runner. Main Dimensions are forming the most important basis for all following design steps.
The real flow in a Kaplan turbine runner is turbulent and three-dimensional. Secondary flows, separation and reattachment in boundary layers, transient recirculation areas and other features may occur. Nevertheless it is useful - and it is common practice in the turbine design theory - to simplify the realistic flow applying representative streamlines for the first design approach.

Employing 1D-streamline theory the following cross sections are significant in particular: suction area (index S), just before leading edge (index 0), at the beginning (index 1) and at the end of the blade (index 2), behind the trailing edge (index 3) and at the outlet (index 4).

The design of the main dimensions has to be made in a strict order. This will be secured by the following: One step within the design has to be finished completely before the next can be accomplished. That is to say, the changeability of a tab sheet will be disabled by CFturbo until all necessary parameters have been specified.
7.1.7.1 Setup

On page Setup one can specify some basic settings.

General

- **Manual dimensioning**
  In manual dimensioning mode the main dimensions and blade angles are not calculated by CFTurbo. All these values are user-defined input values.

- **Unshrouded**
  Design a shrouded (closed) or unshrouded (open) runner.
  For an unshrouded runner one must define the **tip clearance**, optional different values at inlet and outlet.

- **Material density**
  The material density of the impeller is an informational value that is not relevant for the hydrodynamic design but is used for the calculation of moments of inertia. Density values can be directly entered or selected from a list by pressing the settings button next to the input area.
**Inflow swirl**
For the determination of the inlet swirl either of the following options can be used:

- outlet swirl of the upstream component,
- absolute inlet flow angle \( \alpha_1 \),
- outlet swirl velocity \( c_{u2} \) in accordance to the Euler equation:
  \[ g \cdot H_n = c_{u1} \cdot u_1 - c_{u2} \cdot u_2. \]

**Initial default setting**

When creating a new design the initial default settings for some important properties are displayed in the panel **Initial default settings**. These settings are used in further design steps and can be modified by selecting the **Change settings** button. Of course these default settings can be modified manually in the appropriate design steps. See **Preferences: Impeller/ Stator settings** for more information.

**Information**

Some design point values are displayed in the right **Information** panel when selecting the page **Values** (see **Global setup**).
7.1.7.2 Parameters

On page Parameters one has to put in or to modify parameters resulting from approximation functions in dependence on specific speed \( n_q \) (see Approximation functions).

Parameter and efficiency values can be handled manually or can be switched to automatic update by the checkbox on top of the page. Then the default values are used always, even after design point modifications (see Global setup).

If the automatic mode is not selected the current default values can be specified by one of the following options:

- globally by the button on top of the page
- regionally by the default button within the Parameters or Efficiency region

For details of how to handle the parameter edit fields please see Edit fields with empirical functions.
Parameters

The panel Parameters allows defining alternative parameters in each case for the calculation of the following impeller diameters:

<table>
<thead>
<tr>
<th>inlet</th>
<th>outlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>$d_{S1}$, $d_{H1}$</td>
<td>$d_{S2}$, $d_{H2}$</td>
</tr>
</tbody>
</table>

For $d_{S1}$-calculation

- Work coefficient $\psi$: dimensionless expression for the specific energy:
  \[ \psi = \frac{Y}{u_1^2 / z} \]
- Specific diameter $\delta$: according to Cordier diagram (see Dimensions)
For $d_{H1}$ calculation

Diameter ratio $d_{H1}/d_{S1}$

\[
\frac{d_{H1}}{d_{S1}} = 0.7...0.15
\]

For $d_{S2}/d_{H2}$-calculation

<table>
<thead>
<tr>
<th>Meridional velocity ratio $c_{m2}/c_{m1}$</th>
<th>Strictly axial</th>
<th>$d_{H2} = d_{H1}$ and $d_{S2} = d_{S1}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Const. hub</td>
<td>$d_{H2} = d_{H1}$</td>
<td></td>
</tr>
<tr>
<td>Const. mid</td>
<td>$d_{M2} = d_{M1}$</td>
<td></td>
</tr>
<tr>
<td>Const. shroud</td>
<td>$d_{S2} = d_{S1}$</td>
<td></td>
</tr>
</tbody>
</table>

Efficiency

In panel Efficiency you have to specify several efficiencies. You have to distinguish between design relevant efficiencies and efficiencies used for information only:

**Design relevant**

- hydraulic efficiency $\eta_h$

**Information only**

- mechanical efficiency $\eta_m$

The losses resulting in energy dissipation from the fluid form the internal efficiency.

\[
\eta_I = \eta_h \cdot \frac{\eta_m}{\eta_h}
\]

Internal and mechanical efficiency form the overall efficiency (coupling efficiency) of the stage $\eta_{St}$.

\[
\eta_{St} = \frac{P_D}{P_Q} = \eta_I \cdot \eta_m
\]

$P_D$: Runner power

$P_Q$: Power output (coupling)

The following summary illustrates the single efficiencies and their classification:
The obtainable overall efficiency correlates to specific speed and to the size and the type of the runner as well as to special design features like bypass installations and auxiliary aggregates. Efficiencies calculated by approximation functions are representing the theoretical reachable values and they should be corrected by the user if more information about the impeller or the whole turbine are available.

The hydraulic efficiency (or blade efficiency) describe the energy losses within the turbine caused by friction and vorticity. Friction losses mainly originate from shear stresses in boundary layers. Vorticity losses are caused by turbulence and on the other hand by changes of flow cross section and flow direction which may lead to secondary flow, flow separation, wake behind blades etc.

The mechanical efficiency mainly includes the friction losses in bearings and seals:

\[ \eta_m = 1 - \frac{P_m}{P} \]

Hydraulic and volumetric efficiency are most important for the runner dimensioning because of their influence on \( \bar{Y} \) and/or \( \bar{Q} \). The mechanical efficiency is affecting only the actual output power of the machine.

**Information**

In the right area of the register **Parameter** you can find again some calculated values for information:

<table>
<thead>
<tr>
<th>Torque</th>
<th>( M = \frac{P_Q}{\omega} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Actual power output</td>
<td>( P_D = P_Q \cdot \eta_{St} )</td>
</tr>
<tr>
<td>Power loss</td>
<td>( P_L = P_Q - P_D )</td>
</tr>
<tr>
<td>Internal efficiency</td>
<td>( \eta_I = \eta_h \cdot \eta_h^+ )</td>
</tr>
</tbody>
</table>
Stage efficiency

\[ \eta_{St} = \frac{P_D}{P_Q} = \eta_I \cdot \eta_m \]

7.1.7.3 Dimensions

The main dimensions of a designed runner - suction diameter \( d_{s1} \) and \( d_{h1} \), and outlet diameter \( d_{s2} \) and \( d_{h2} \) - can be seen on Main dimensions panel. They can be recomputed by pressing the Calculate-button. The computation is based on "Euler's Equation of Turbomachinery", on the continuity equation and the relations for the velocity triangles as well as on the parameters and parameter ratios given in the tab sheets Setup and Parameters.

Individual main dimensions can be calculated separately using the button inside the value field.

You may accept the proposed values or you can modify them slightly, e.g. to meet a certain normalized diameter.

In case the checkbox Automatic is activated a new calculation will accomplished after any change of parameter. Then the manual alteration of the main dimensions is not possible.
**Neighboring components**

In specific cases the dimensions of the neighboring components at inlet and/or outlet can be used to get exactly matching geometry.

<table>
<thead>
<tr>
<th>Get</th>
<th>Inlet</th>
<th>Outlet</th>
<th>from neighboring component</th>
</tr>
</thead>
</table>

This feature is available only for explicitly uncoupled components or side-by-side impellers.

**Information**

In the right panel of any tab sheet an information panel is situated, which holds the computed variables in accordance to the actual state of design, the resulting Meridional section as well as the Cordier-Diagram with the location of the best point. These three sections can be chosen by the appropriate soft buttons in the heading.

In the Value section the following variables are displayed for information which result from calculated or determined main dimensions:
<table>
<thead>
<tr>
<th></th>
<th>Formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>Work coefficient</td>
<td>$\psi = \frac{Y}{u_1^2/2}$</td>
</tr>
<tr>
<td>Flow coefficient</td>
<td>$\varphi_t = \frac{Q}{\eta/4 , d_1^2 , u_1}$</td>
</tr>
<tr>
<td>Meridional flow coefficient</td>
<td>$\varphi_m = \frac{c_{m1}}{u_1}$</td>
</tr>
<tr>
<td>Specific diameter</td>
<td>$\delta = \psi^{3/4} , \varphi_{t/2} = 1.05 , d_1 \left( \frac{Y}{Q^2} \right)^{1/4}$</td>
</tr>
<tr>
<td>Average inlet velocity</td>
<td>$\bar{c}<em>{m1} = \frac{Q / \eta_p}{\eta/4 \left( d</em>{S1}^2 - d_{H1}^2 \right)}$</td>
</tr>
<tr>
<td>Average inlet velocity (net)</td>
<td>$\bar{c}<em>{m1}^* = \frac{Q}{\eta/4 \left( d</em>{S1}^2 - d_{H1}^2 \right)}$</td>
</tr>
<tr>
<td>Average outlet velocity</td>
<td>$\bar{c}<em>{m2} = \frac{Q / \eta_p}{\eta/4 \left( d</em>{S2}^2 - d_{H2}^2 \right)}$</td>
</tr>
<tr>
<td>Average outlet velocity (net)</td>
<td>$\bar{c}<em>{m2}^* = \frac{Q}{\eta/4 \left( d</em>{S2}^2 - d_{H2}^2 \right)}$</td>
</tr>
<tr>
<td>Outlet width ratio</td>
<td>$b_2/d_2$</td>
</tr>
<tr>
<td>Meridional velocity ratio</td>
<td>$\bar{c}<em>{m2} / \bar{c}</em>{m1}$</td>
</tr>
</tbody>
</table>

The **Meridional preview** is until now based on the main dimensions only.
The **Cordier** diagram can be used for checking the impeller diameter $d_2$.

See **Cordier**. 

<table>
<thead>
<tr>
<th>Values</th>
<th>Meridian</th>
<th>Cordier</th>
<th>Velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>$N_0$</td>
<td>$n_0$</td>
<td>$d_2$</td>
<td>$V$</td>
</tr>
<tr>
<td>500</td>
<td>10</td>
<td>500</td>
<td>50</td>
</tr>
<tr>
<td>1000</td>
<td>20</td>
<td>1000</td>
<td>100</td>
</tr>
<tr>
<td>2000</td>
<td>50</td>
<td>2000</td>
<td>200</td>
</tr>
<tr>
<td>3000</td>
<td>100</td>
<td>3000</td>
<td>1000</td>
</tr>
</tbody>
</table>

![Cordier Diagram](image)

![Diagram](image)

Specific diameter $d = \Psi^{3/2} \cdot \frac{Q}{n} = 1.05 \cdot \frac{Q}{n^{3/4}}$
The **Velocity triangles** are the result of a mid-span calculation and are based on the **design point** and the main dimensions.

<table>
<thead>
<tr>
<th>Values</th>
<th>Meridian</th>
<th>Cordier</th>
<th>Velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>absolute (c)</td>
<td>relative (w)</td>
<td>velocity @ mid-span</td>
<td></td>
</tr>
</tbody>
</table>

![Velocity triangles diagram](image)

© CFturbo GmbH
7.1.8  Material density

In the table a material with its density can be selected. The list can be extended or reduced by + and - button respectively. A confirmation of the selected value is done by pressing the OK-button.

At File location the file containing material properties is shown. The file is originally called Density.cftst and is located in the installation directory of CFturbo. Modifications of the list will be saved if the user is leaving the dialog window by clicking the OK-button. In case there are no write permissions the user can choose another directory to save the file.

Renaming of files is possible by Save as-functionality. By clicking the Open-button a previously saved file can be opened.
7.1.9 Multi stage

If more than 1 impeller or rotor shall be contained in the project this can be defined in the panel Multi stage options on page Setup.

The design point (head, pressure difference etc.) can be distributed amongst the impellers using the power splitting. The energy goal used for the design of the selected impeller (index $i$) is determined by:

$$
E_i = \sum e_i \cdot E
$$

where the capital $E$ may either be head, specific work or pressure difference resp. The lower case $e_i$ is the ratio describing the power splitting for the selected impeller. This ratio is to be defined by the track bar in the panel Multi stage options or can be directly specified underneath.
In case more than 1 rotor is contained in the project the design point (Power output, pressure ratio) can be distributed amongst the rotors using the power splitting. The energy goal used for the design of the selected rotor (index i) is determined by:

\[ P_i = e_i \cdot P_{\text{Global}} \]

where the \( P \) is the actual power output. The lower case \( e_i \) is the ratio describing the power partitioning for the selected rotor. In case a pressure ratio \( \pi \) has been specified in the Global setup the pressure ratio used for the design of the selected rotor is determined by:

\[ \pi_i = \frac{\pi}{\prod_{j \neq i} \pi_j} \]

### Contra rotating, Alternative speed

The impeller may be defined with both a different rotating direction as well with a rotating speed different from the specifications set in the Global Setup. To this end the respective check box has to be set. If alternative speed is checked, an edit box appears in where the desired values has to be defined.

By activating the Automatic parameters and the Automatic main dimensions calculation, each change of the rotational speed results in an update of the main dimensions. This allows the rotational speed to be used to obtain a certain impeller diameter.
7.1.10 Shaft/Hub

Dimensioning of the shaft diameter is made under application of strength requirements. It is a result of torque \( M = \frac{P}{\omega} \) to be transmitted by the shaft and the allowable torsional stress \( \tau \) of the material.

You can directly enter allowable stress or select the value from a list by pressing settings button right beside the input area.

In a small dialog window you can see some materials and its allowable stress. The basis of these values is the allowable torsional stress of the material. Due to the fact that the shaft is additionally loaded by bending in the same order of magnitude, a safety factor of 10 ... 15 is used typically, which leads to the relatively low stress values (see References: Steinhilper/ Röper).

The list can be extended or reduced by button. You can confirm selected value by pressing the OK-button.

At File location the file containing material properties is shown. The file is originally called Stress.cftst and is located in the installation directory of CFturbo. Modifications of the list will be saved if the user is leaving the dialog window by clicking the OK-button. In case there are no write permissions the user can choose another directory to save the file. Renaming of files is possible by Save as-functionality. By clicking the Open-button a previously saved file can be opened.

To consider a higher load, e.g. due to operating conditions away from the design point, a safety factor \( SF \) may be specified leading to a modified proposed shaft diameter \( d \).

\[
d \geq \sqrt[3]{\frac{8\nu QY \cdot SF}{\pi^2 \nu \eta}}
\]

The hub diameter \( d_H \) is usually selected as small as possible and depends on the kind of connection of hub and shaft.
The Cordier diagram is based on an intensive empirical analysis of proved turbomachinery using extensive experimental data.

At the top right you can switch between different coordinate axes:

- **relative**: Specific diameter $\delta$ vs. Specific speed $\sigma$ (or $nq$ or $Ns$); original Cordier diagram
- **absolute**: Impeller/Rotor diameter $d$ vs. Rotational speed $n$; valid for fixed design point values

Additionally, straight lines for the work coefficient $\psi$ and the flow coefficient $\phi$ are displayed.

The Cordier diagram is displayed in [Global setup](#) and the [Main dimensions](#).
7.1.12 Balje

The **Balje diagram** is based on the work of O. E. Balje. For given specific speed and diameter coefficient as well as degree of reaction the diagram delivers suggestions for:

- Width ratio $h/d_S = f(nq, \delta)$,
- Isentropic velocity ratio total-static $u/c_0 = f(nq, \delta, h/d)$,
- Absolute flow angle $\alpha^* \_1 = f(\delta, h/d, \gamma)$,
- Relative flow angle $\beta^* \_1 = f(nq, \delta, h/d, \alpha^* \_1)$,
- Rotor velocity ratio $\psi_R = w_2/w_{2id} = f(nq, \delta, h/d, \alpha^* \_1, \beta^* \_1)$,
- Efficiency total-static $\eta_{ts} = f(nq, \delta, h/d, \gamma, \beta^* \_1, \psi_R)$.

The density ratio $\gamma = \rho_1/\rho_2$ is determined by the solution of a set of equations (Euler, continuity, etc.). Balje's angle notation is displayed in the following image:

The Balje diagram is displayed in the **Main dimensions**. If Energy transmission = total pressure ratio $\pi_{tt}$, see **Global setup**, the diagram is only for information and diagram parameters can be defined on top of the diagram (left image).

If Energy transmission = total-to-static pressure $\pi_{tst}$, the diagram is used to make proposals for $u/c_0$, $h/d$ and $\eta_{ts}$, which are design parameters. On the second tab parameters derived from the Balje diagram in the design point (by specific speed, diameter coefficient and degree of reaction) are displayed for information.
### 7.2 Meridional contour

The design of the meridional contour is the second important step to design the impeller.
Meridional design is divided in 3 parts:

- **Primary flow path**
  This contains the design of the primary flow path. Necessary for the following design steps.

- **Hub/Shroud materials** (optionally)
  This contains the design of hub and/or shroud material solids. This is an optional part focusing on stress analysis.

- **Secondary flow path** (optionally)
  This contains the design the secondary flow path behind hub and/or shroud. This is an optional part focusing on detailed flow analysis.

**Possible warnings**

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solution</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet hub diameter: deviations between Meridional Contour and Main Dimensions are larger than 0.1%</td>
<td>Adjust either the main dimensions or the imported curve.</td>
</tr>
<tr>
<td>The difference between the hub diameter and the corresponding geometric size in the meridian is</td>
<td></td>
</tr>
<tr>
<td>Problem</td>
<td>Possible solution</td>
</tr>
<tr>
<td>---------</td>
<td>------------------</td>
</tr>
<tr>
<td>too large. This is possible for imported polylines only.</td>
<td></td>
</tr>
<tr>
<td><strong>Inlet shroud diameter: deviations between Meridional countours and main dimension are larger than 0.1%</strong></td>
<td>Adjust either the main dimensions or the imported curve.</td>
</tr>
<tr>
<td>The difference between the suction diameter and the corresponding geometric size in the meridian is too large. This is possible for imported polylines only.</td>
<td></td>
</tr>
<tr>
<td><strong>Outlet diameter: deviations between Meridional Contour and Main Dimensions are larger than 0.1%</strong></td>
<td>Adjust either the main dimensions or the imported curve.</td>
</tr>
<tr>
<td>The difference between the impeller diameter and the corresponding geometric size in the meridian is too large. This is possible for imported polylines only.</td>
<td></td>
</tr>
<tr>
<td><strong>Outlet width: deviations between Meridional Contour and Main Dimensions are larger than 0.1%</strong></td>
<td>Adjust either the main dimensions or the imported curve.</td>
</tr>
<tr>
<td>The difference between the outlet width and the corresponding geometric size in the meridian is too large. This is possible for imported polylines only.</td>
<td></td>
</tr>
<tr>
<td><strong>Hub/ Shroud contour has discontinuities inside blade region.</strong></td>
<td>Adjust hub resp. shroud contour and apply smoothness at connectors who are inside blade region.</td>
</tr>
<tr>
<td>The hub resp. shroud contour is divided into sub-curves who are not connected smoothly in blade region.</td>
<td></td>
</tr>
<tr>
<td><strong>Angle between hub/ shroud contour and inlet/ outlet is not recommended.</strong></td>
<td>Manipulate hub/shroud contour or move inlet/outlet to change the current angle to inlet/outlet.</td>
</tr>
<tr>
<td>The current angle between hub/shroud contour and inlet/outlet can cause problems in Model finishing.</td>
<td>Contour may contain extremely small and unnecessary parts which should be removed.</td>
</tr>
<tr>
<td><strong>Hub contour intermittently touching z-axis (r=0) is not supported.</strong></td>
<td>Avoid hub regions at r = 0 internally or split geometry into different components.</td>
</tr>
<tr>
<td>The hub curve is touching the z-axis internally. Before and behind it the radius is greater than 0 creating a complete constriction.</td>
<td></td>
</tr>
<tr>
<td><strong>Meridional contour has an invalid topology. (inside out)</strong></td>
<td></td>
</tr>
</tbody>
</table>
The sense of circulation of the closed wire containing the inlet, shroud, outlet and hub curve in this specific order is counter-clockwise.

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solution</th>
</tr>
</thead>
<tbody>
<tr>
<td>The sense of circulation of the closed wire containing the inlet, shroud, outlet and hub curve in this specific order is counter-clockwise.</td>
<td>Manipulate meridional curves to guarantee a clockwise sense of circulation or adjust inlet and outlet in main dimensions.</td>
</tr>
</tbody>
</table>

7.2.1 Primary flow path

Content of this section is split into these topics:

- Hub, Shroud
- Leading, Trailing edge
- Meridional flow calculation

Graphical elements can be manipulated not only by the computer mouse per drag and drop but also by using context menus. To this end a right click on the appropriate element is necessary. Doing so the mode of the leading edge can be changed as well as the coordinates of Bezier points for example.

There are some reasonable constraints when working in simplified modes e.g. the inclination angle of the trailing edge can only be set when hub and shroud are in Bezier mode both.

**Display Options**

In the Display Options panel some graphical representations can be activated for illustration:
Area circles
used for calculation of cross section area

Neighboring components
on inlet and outlet side are displayed for information.

Max. curvature
point is displayed on hub and shroud curves

Use the buttons above the diagram to zoom the current meridional shape only or the entire geometry.
7.2.1.1 Hub, Shroud

**Design Mode**

There are two different options to define hub and shroud contours.

- **Hub, Shroud** Direct design of the two contours
- **Middle** Design of center line; the contours result from given cross section distribution between suction (dS) and outlet (d2) cross sections

**Hub, Shroud**

In the first case, hub and shroud can be designed separately.

**Middle**

In the second case, only the geometric center line of the flow channel will be modified. The contours result from specifying a relative cross section distribution. It may either be linear or could be loaded from a file using the Progression dialog.

The first value of each line is the relative meridional coordinate $x$ along the center line, with $x=0$ at the inlet cross-section and $x=1$ at the outlet cross-section. The second value is the relative cross section $A_{rel}$, which allows to compute the related absolute value:

$$A = A_{in} + A_{rel} (A_{out} - A_{in})$$

The cross section is used to determine the meridional width $b$ vertical to the flow direction.

This strategy is mainly suitable for mixed-flow impellers, it’s suboptimal for centrifugal impellers with relative sharp direction change from axial to radial.

**Axial dimensions**

The axial start position as well as the axial extension of the meridional shape can be specified in the **Axial dimensions** area on the right. Both can also be modified interactively in the graphics.

**Curve mode**
Contour curves can be designed as:

- **Bezier curve**
  The curve is defined by the position of the Bezier points.
  ➔ Details

- **Circular Arc + Straight line**
  The curve consists of a circular arc and a straight line.
  ➔ Details

- **Straight line**
  The contour is defined by a straight line between start and endpoint.

- **Polyline**
  The curve is fixed and cannot be modified interactively. Import of point sets from file is possible (Load polyline). Note, imported point sets are re-sampled on an interpolating cubic spline using 100 points. This may lead to deviations to original polyline, especially in case of low point counts.

Centrifugal fan impellers are designed simply by arc and line by default (Circular Arc + Straight line), all other impeller types in Bezier mode (Bezier curve).
On the endpoints of hub and shroud the complete geometry can be shifted optionally (Shift complete geometry). Hence the geometry can be positioned on a specific axial position.

7.2.1.1.1 Bezier

**Bezier curves**

Hub and Shroud are represented by 4th order Bezier curves. This is the default and most flexible curve mode.

The curve is determined by five Bezier points.
Points 0 and 4 are defining the endpoints of the curves while the other three points determining the shape of the curve. The middle point (2) can be moved without any restrictions whereas points 1 and 3 have only one degree of freedom. Point 1 is only movable on the straight line between points 0 and 2, point 3 between point 2 and 4. Therefore no curvature is occurring at the end of the curves. In conjunction with a continuous curvature gradient small velocity gradients can be expected. The two straight lines are defining the gradients in the end points of the curves.
Bezier point 2 can be limited in its mobility by the curve context menu option **Limit stop**. As a result the axial and radial position is limited in the area between the curve endpoints 0 and 4.

The above mentioned coupling between the Bezier points can be switched on or off by the curve context menu option **Coupled Bezier points**.

Start angle (line 0-1 or 0-1-2) and end angle (line 3-4 or 2-3-4) can be fixed optionally by the curve context menu option **Fixed start angle** or **Fixed end angle**. A fixed angle is illustrated by a dotted line instead a dashed one and by a triangular marker on the curve endpoint.

**Tangential connection**

In Bezier mode a tangential connection to neighboring components (impeller or stator) can be switched on or off using the icon beside the the first or last Bezier point:
Primary design

For an automatic primary design of the contours the following values are used:

- **Main dimensions**: $d_H$, $d_S$, $d_2$, $b_2$
- Inclination angle $g$ of trailing edge to horizontal (see Approximation functions)
- Inclination angle $e$ of hub and shroud to vertical (see Approximation functions)
- Axial extension: pumps, fans according to a) (Guelich), turbines according to b) (Lindner), compressor according to c) (Aungier). In some cases where the hub diameter $d_H$ is quite small compared to the impeller diameter $d_2$, for compressors the average of a) and b) is applied instead of c).
b) \( \Delta z = (d_{1/2} - d_1)^2 \)

c) \( \Delta z = d_2 \left( 0.014 + 0.023 \cdot \frac{d_0}{d_1} + 1.58 \cdot \varphi \right) \)

Point 1 is primarily placed at 3/4 of the axial distance of points 0 and 2, point 3 at 2/3 of the radial distance of points 2 and 4.

The manipulation of the contours can be achieved by shifting the positions of the Bezier points. As an alternative, the position of Bezier points can be realized by input of numerical values (see Graphical dialogs). Trailing edge can be rotated by moving Bezier points 4. If <Ctrl> key is pressed simultaneously the whole trailing edge can be moved in axial direction with constant inclination angle (change axial extension). Inclination angle of trailing edge can be numerically determined by clicking the right mouse button on it.

In the design process for the meridional contours the user should try to create curvatures which are as steady as possible in order to minimize local decelerations. The maximum values of the curvature should be as low as possible and should entirely disappear at the end of the contours. These requirements are met very well by Bezier curves showing the above mentioned limitations. Local cross section \( 2\pi rb \) should grow from the suction to the impeller diameter as uniformly as possible.

If using simple polyline for hub and/or shroud - e.g. for imported meridional geometries - this curve can be converted to a Bezier curve. Thus, it’s possible to make systematic modifications of existing geometries.

7.2.1.1.2 Circular Arc + Straight line

Hub and shroud are represented by the segment of a circle and a tangential straight line. The radius of the segment is defined by Point 1. The points 0 and 2 are defining the axial position of the meridional contour.
For an automatic primary design of the contours the following values are used:

- **Dimensions**: \( d_1, d_2, b_1, b_2 \)
- Radius of the circle segment \( R: 14\% \) of \( d_s \)

The manipulation of the contours can be achieved by shifting the positions of the points. As an alternative the position of points can be realized by input of numerical values. By moving points 0 or 2 the whole geometry can be moved in axial direction.
Impeller

- $d_2$
- $b_2$
- $\varepsilon_{shr}$
- $d_s$
- $b_1$
- $d_H$

© CFturbo GmbH
The design of hub and shroud can be expanded optionally. Therefore additional curves can be added on inlet and outlet side in order to design complex contour curves.

The additional inlet and outlet curves can be switched to any curve type (Bezier, Circular, Straight, Polyline) by their own popup menu.

**Tangential transition**

The tangential transition between neighboring curves can be switched on or off using the icon beside the the first or last Bezier point:
7.2.1.2 Leading, Trailing edge

Leading and trailing edge contour can be designed as:

- **Bezier curve**
  The Leading edge is defined by the position of the Bezier points.

- **Straight**
  The Leading edge is a straight connecting line between the endpoints on hub and shroud.

- **$r = \text{constant}$**
  The Leading edge runs on constant radius, i.e. parallel to rotational axis.

- **$z = \text{constant}$**
  The Leading edge runs on constant axial coordinate, i.e. perpendicular to rotational axis.

The trailing edge can not be designed, if [Trailing edge fixed on outlet](#).

The position of the meridional blade leading edge on hub and shroud can be defined by its axial ($z$), radial ($r$) or relative position (rel.) optionally.

In case of **Splitter blades** each leading edge can be designed individually.

The radial-inflow gas turbine rotors and compressor impellers have straight leading edges by default, in case of radial-inflow gas turbines $r = \text{constant}$ additionally.

Leading edge can be designed as a straight line by selecting **Straight** in the context menu of the curve (controlled by 2 Bezier points). Additionally the edge can be strictly axial or radial ($z = \text{const.}$ or $r = \text{const}$, controlled by 1 Bezier point).

For centrifugal impellers having $n_q \geq 10...30$ the leading edge is often designed parallel to the z-axis. As the trailing edge is parallel to the axis too for such applications 2D-curved blades can be created. At higher specific speed $n_q$ or due to strength reasons the leading edge often is extended into the impeller suction area. Various diameters result in different leading edge blade angles - therefore 3D-curved blades are created. This leads to better performance curves, higher efficiencies and improved suction capacity for pumps.

The position of the leading edge should be chosen in a way that the energy transmission should be about equal on all meridional flow surfaces. A criterion is the approximately equal static moment \( S = \int r \, dx \) of the meridional streamlines on hub and shroud between leading and trailing edge. In the **Static moment** section the corresponding numerical values are displayed. Both ends of the leading edge should be perpendicular to the meridional contours of hub and shroud if possible. To obtain equal static moments on hub and shroud the trailing edge is often not parallel to axial direction - particularly at higher specific speeds (mixed-flow impellers).
The leading edge can be designed by a 4th order Bezier curve. Regarding the Bezier points, the properties are similar to the hub/shroud curves. The only difference is the manipulation of the end points, which are located on the hub/shroud curves always. The position of the leading edge always appears at the same relative position in a primary CFturbo design but this not mean to be a suggestion.
Leading/Trailing edge fixed on ...

The leading/trailing edge is fixed on meridional inlet/outlet and can not be designed.

Uncheck this option to detach the leading/trailing edge from meridional inlet/outlet and design its position and shape independently.

7.2.1.3 Meridional flow calculation

Meridional flow visualization based on potential flow theory is available optionally. The result of meridional flow calculation can be displayed using the button top right of the diagram.

The display can be configured using the corresponding menu (all options can be combined).
Grid

After each change of the meridional contour a new computational grid is calculated.

Extensions are added to the inlet and outlet in order to ease the setup of the boundary conditions.

Streamlines

meridional streamlines (lines with constant values of the stream function)

equal mass flow fraction between neighboring streamlines
Impeller

**cm-lines**
iso lines of const. meridional velocity $c_m$

**cm-surfaces**
iso surfaces of const. meridional velocity $c_m$

*(scaling is displayed below the diagram)*

**Blade correction**
If activated, the blockage effect of blade thickness is considered for flow calculation.

**Vectors**
Vectors of meridional velocity $c_m$
Stream function \( \psi \)

Within the meridian the equation for stream function \( \psi \) will be solved. For an incompressible fluid this equation is in cylindrical co-ordinates \((z, r)\):

\[
\frac{\partial^2 \psi}{\partial z^2} + \frac{\partial^2 \psi}{\partial r^2} - \frac{1}{r} \frac{\partial \psi}{\partial r} = 0.
\]

For a compressible fluid the equation looks like:

\[
\left[ 1 - \frac{1}{a^2} \left( \frac{\partial \psi}{\partial z} \right)^2 \right] \frac{\partial^2 \psi}{\partial z^2} + \left[ 1 - \frac{1}{a^2} \left( \frac{\partial \psi}{\partial r} \right)^2 \right] \frac{\partial^2 \psi}{\partial r^2} + \left[ -\frac{2}{a^2} \frac{\partial \psi}{\partial z} \frac{\partial \psi}{\partial z} \right] - \frac{1}{r} \frac{\partial \psi}{\partial r} = 0,
\]

where \( a \) is the sonic speed defined by:

\[
a = \sqrt{\kappa \cdot R \cdot Z \cdot T}.
\]

Hub and shroud are representing stream lines where at in and outlet there is a certain stream function distribution chosen. This is done in accordance to the mass flow imposed by the global setup [103].

Calculation grid and solution scheme

The equation is solved using a finite-difference-method (FDM) on a computational grid, which will be generated using an elliptic grid generation. For more information about the used computational techniques refer to e.g. Anderson et al [103].

Results

The meridional velocity component can be calculated by the axial velocity component:

\[
c_z = \frac{f_R \cdot p_n \cdot \frac{\partial \psi}{\partial r}}{r \cdot \rho},
\]

by the radial velocity component:

\[
\text{with:}
\]
\[ c_m = \sqrt{c_r^2 + c_f^2}. \]

\( r_n \) and \( \rho_n \) are reference radius and density respectively. In case of incompressible fluids the density is constant throughout the flow domain and the according term in the equations is discarded.

**Example**

On the basis of the updated grid the equation for stream function is solved and lines with constant values of the stream function and of the meridional velocity are displayed.

**Annotation**

Due to the potential flow theory the given solution is only a rough estimation of the real meridional flow. One has to bear in mind that friction is not considered as well as the no slip boundary condition at hub and shroud. For detailed flow analysis CFD-techniques for solving the entire set of Navier-Stokes-Equations has to be used. Also the solution scheme implemented (FDM) may not always find a solution for every combination of design point and meridional contour.

Singularities will occur if the solution domain has radii close to zero. Then at those locations some artefacts might exist in the meridional velocity contours.

For compressible fluids it is necessary that the flow regime in the entire domain has to be far away from transonic conditions. Otherwise the equation will not have solution.

**7.2.2 Hub/Shroud materials**

Meridional material solids can be designed by selecting the "Hub/Shroud materials" page on top left and activate the feature on the right side. Initial contours will be visible, which can be manipulated using the control points and the context menu of the curve.
Some diagram display settings are available using the button bottom left.

After the model finishing, the meridional material parts and the blade solid are connected to a single solid geometry, "Material domain".

**Material design**
Hub & Shroud materials can be designed by manipulating their meridional contours. The feature is available if the following conditions are fulfilled:

- **Hub**: Only available for existing hub contour
- **Shroud**: Only available for shrouded impellers

By activating one of the material solids a default contour is shown. This contour is represented by a *wire* connecting the endpoints of meridional hub or shroud contour and can be modified interactively using its graphical elements. To provide complex contours the *wire* can consist of various *edges* which can be configured independently from each other. They are separated visually by *squared control points*. In contrast *circular control points* belong to *edges* and are connected by a *dashed control polygon*. Apart from the endpoints of the wire, all control points are moveable.

**Context menus of contour wire**

Besides drag & drop of control points *context menus* of graphical objects are alternative means to modify the contour. They are accessible via right-clicking and contain useful tools for the graphical object underneath.

- **Edit point**: Opens a small panel to set coordinates of control point.
- **Merge curve**: Transforms connector into inner control point by merging control polygons. Note that this option is only available if neighboring curves are from the same edge type.
Meridional contour of primary flow path

- **Set offset curve**: Resets the complete wire using a user-defined offset. This offset is not kept when changing the meridional contour.

Edge types of contour wire

An edge of a contour wire can be one of the following type:

- **Bezier curve**
  The curve is defined by the position of the Bezier points.
  ➔ Details

- **Line segment curve**
  The curve consists of straight lines and rounded corners. (optional)
  ➔ Details

Snapping

To simplify contour design snapping of points on coordinates and lines can be used. This mode can be chosen by clicking on the right corner below of the diagram and selecting one of the snapping types:

Snapping types

- **Point**: Currently moved point snaps to x and y ordinates of other points related to the wire. This also includes ordinates of its start position.

- **Line**: Currently moved point snaps to lines defined by all pairs of nearby points along the wire. This also includes the two lines crossing its start position.

Note that snapping can be deactivated temporarily by pressing **Shift** while dragging.

Possible warnings
<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solution</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hub/Shroud material contour has invalid edges.</td>
<td>Corner radius of the invalid (red) curve has to be reduced. This could be achieved by dragging the circle center. The curve is valid when it is no longer drawn in red.</td>
</tr>
</tbody>
</table>

When using **Line segment curves** with rounded corners the defined radius could be too large. This can result in sharp edges and very small segments.

7.2.2.1 **Bezier curve**

**Context menus**

Options of the context menu for the graphical object of a **Beziér curve** are listed below:
Control point

- Remove Beziér point: Removes selected control point from polygon.

- Edit point: Opens a small panel to set coordinates of control point.

- Split polygon: Divides control polygon into two control polygons which in turn defines two Beziér curves. This process can be reverted using Merge curve of the connector.

Control polygon

- Insert Beziér point: Inserts a new Beziér control point into control polygon

Curve menu

- Increase Beziér point count: Increases the number of control points without changing the shape of the curve. Note that all inner points will be rearranged. Note that this function naturally increases the degree of the curve.

- Remove inner control points: Removes all inner control points of the control polygon so that the Beziér curve reduces to a line.

- Save polyline: Saves 100 curve points in a file.

- Split curve: Subdivides the Beziér curve into two curves while keeping the shape of the original curve. Note that inner points will be rearranged and that splitting can not be reverted.

- Offset curve: Applies an offset to the curve underneath. Splitting curve before can increase accuracy of the resulting offset curve.
7.2.2.2 Line segment curve

Context menus

Options of the context menu for the graphical object of a Line segment curve are listed below:

Control point

- **Remove polygonal point**: Removes selected control point from polygon.
- **Edit point**: Opens a small panel to set coordinates of control point and fillet radius.
- **Split polygon**: Divides control polygon into two control polygons which in turn defines two Line segment curves. This process can be reverted using **Merge curve** of the connector.

Circle center

- **Edit radius**: Edit fillet radius directly.

Curve

- **Insert polygonal point**: Inserts a new control point.
- **Remove inner control points**: Removes all inner control points of the control polygon so that Line segment curve reduces to a line.
- **Save polyline**: Saves 100 curve points in a file.
- **Split curve**: Subdivides the Line segment curve into two curves while trying to keep the shape of the original curve.
- **Offset curve**: Applies an offset to the curve underneath.

7.2.3 Secondary flow path

Secondary flow path for impellers can be designed by selecting the "Secondary flow path" page on top left and activate the feature on the right side. Initial contours will be visible, which can be manipulated using the control points and the context menu of the curve (see **Hub/Shroud**).
Endpoints of the "secondary flow path" on the material solid are constrained to the contour. The context menu of those points provide "Edit z value" and "Edit r value".

After the model finishing the meridional primary and secondary flow paths are connected to a single solid geometry, "Fluid domain".

Hub and Shroud casing can be designed by manipulating their meridional contours. The feature is available if the following prerequisites are fulfilled:

- Hub/Shroud materials must be activated for this impeller
- Neighboring static components have to exist
- An offset between impeller and neighboring components is required

Depending on settings of main dimensions, some parts of the contour can become inactive, e.g. for unshrouded impellers the shroud is already defined by primary flow path.

The secondary flow path is defining the real geometry. In contrast to this, a simplified virtual geometry can be specified and exported separately (see virtual geometry).

Possible warnings
### Problem

**Missing static component at inlet/outlet. Both, upstream and downstream component to impeller are necessary.**

There has to be a component at the impeller's inlet/outlet for designing a Secondary flow path. Add a component to the impeller's inlet/outlet.

### 7.2.4 Additional views

The following information can be displayed in the meridional contour dialog using the "Additional views" button:

- **Informational values**
  - Minimal curvature radius on hub and shroud (position is marked on the hub and shroud curves)
  - Static moment $S$ from leading to trailing edge on hub and shroud (see below)
  - Angle $\epsilon$ in the hub and shroud end points measured to the horizontal direction
  - Angle $\gamma_{LE}$ of leading edge on hub and shroud measured to the horizontal direction
  - Axial extension $\Delta z$ of hub and shroud
  - Radial extension $\Delta r$ of hub and shroud
  - Angle $\gamma_{TE}$ of trailing edge measured to the horizontal direction
Default axial extension $\Delta z_D$ from inlet shroud to outlet midline (defined for centrifugal impellers only)

Maximal axial extension $\Delta z_M$ of complete meridional shape

Maximal radial extension $\Delta r_M$ of complete meridional shape

Axial blade overlapping $z_B$ of shroud blade area onto hub blade area in $z$-direction

LE distance $b_1$ from LE at hub to LE at shroud

LE circle $b_1$ as diameter of a circle inside the meridional contour at LE position

LE diameter $d_1$ at intersection of LE and midline

Diameter ratio $d_1/d_z$

LE diameter $d_{1,ave}$ as average of hub and shroud diameter at LE

3D-Preview

3D model of the currently designed meridional shape.

The meridian contains hub and shroud as well as a circular projection of the blade in a plane.

Curvature progression

Curvature progression along hub and shroud curve. The progression should be as smooth as possible avoiding hard peaks.

Static moment

© CFturbo GmbH
The static moment is the integral of the curve length \( x \) in the blade area multiplied by the radius \( r \):

\[
S = \int_{x_1}^{x_2} rdX
\]

It should be similar for hub and shroud end points.

**Area section**

Progression of the cross section area between hub and shroud.

Local maximum or minimum should be avoided.

**Cm progression**

Progression of the meridional velocity \( c_m \) along the meridional streamlines.

→ see Meridional flow calculation

### 7.3 Mean line design

The design of the blade's geometry is made in four steps in this design mode:

1. **Blade properties**
2. **Blade mean lines**
3. **Blade profiles**
4. **Blade edges**

#### 7.3.1 Blade properties

Definition of blade properties is made in three steps:

1. **Blade setup**
2. **Span definition**
(3) **Blade angles**

**Absolute and relative flow**

![Velocity triangles diagram]

- **Absolute velocity** $\vec{c}$
- **Relative velocity** $\vec{\dot{w}}$
- **Rotational speed** $\vec{\dot{u}} = \omega \cdot \vec{r}$
- $\vec{\dot{c}} = \vec{\dot{u}} + \vec{\dot{w}}$

Fundamental kinematic equation of Turbomachinery

**Velocity triangles**

<table>
<thead>
<tr>
<th>Radial impeller</th>
<th>Axial impeller</th>
</tr>
</thead>
<tbody>
<tr>
<td>$c_w$: determining energy transmission</td>
<td>$c_m$: determining flow rate</td>
</tr>
</tbody>
</table>
Specification of number of blades

<table>
<thead>
<tr>
<th>Number of blades</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
</tr>
</tbody>
</table>

Usual number of blades are:

- **Pump**
  - 3 ... 7
  - Wastewater: 1 ... 3
  - Barske (low nq): 12 ... 24
  - Inducer: 1 ... 3

- **Fan**
  - 6 ... 10
  - Squirrel cage: 30 ... 60

- **Compressor**
  - Depending on blade exit angle $\beta_2$:
    - 12 for $\beta_2 \approx 30^\circ$
    - 16 for $\beta_2 \approx 45^\circ...60^\circ$
    - 20 for $\beta_2 \approx 70^\circ...90^\circ$

- **Radial-inflow gas turbine**
  - 12 ... 20

- **Axial gas turbine**
  - 30 .. 70 (100)

- **Axial compressor**
  - 20 .. 40

- **Francis turbine**
  - 6 .. 16
Many blades - causing low blade loading - are related to higher friction losses. By choosing of fewer blades - leading to a higher blade loading - the hydraulic losses may rise due to increased secondary flow and stronger deviation between blade and flow direction.

The recommended number of blades according to Pfleiderer is displayed as a hint at the information image \([ \text{for centrifugal & mixed-flow pumps, fans, compressors only}]\):

\[
z = k_z \frac{d_2 + d_1 \sin \beta_1 + \beta_2}{d_2 - d_1} \quad \frac{\beta_1 + \beta_2}{2}
\]

with \(k_z = 6.5 \ldots 8.0\) for compressors, else \(5.0 \ldots 6.5\).

The recommended number of blades using the Zweifel work coefficient is displayed as a hint at the information image \([ \text{for axial gas turbines only}]\):

\[
z = 2 \cdot \pi \frac{d_{av}}{\psi \cdot \Delta z} \left(\tan(90^\circ - \alpha_1) \cdot \tan(90^\circ - \alpha_2) \cdot \cos^2(90^\circ - \alpha_2)\right)
\]

with \(\Delta z\) the axial chord length and \(d_{av}\) the average impeller diameter.

The Zweifel work coefficient is in the range of \(\psi = 0.75 \ldots 1.15\) and is specified in the approximation functions \([\pi \theta]\).

**Splitter linked to Main blade**

If the impeller has splitter blades then the shape of the splitter can be linked to the main blade optionally. If linked the splitter blades are truncated main blades. Otherwise the splitter blade can be designed completely independent.

<table>
<thead>
<tr>
<th>Number of blades</th>
<th>1</th>
<th>14</th>
<th>Splitter linked to Main blade</th>
</tr>
</thead>
</table>

**Information**

In the right panel some information are displayed which result from calculated or determined values:
(1) Velocity triangles

The velocity triangles of inflow and outflow are displayed.

Continuous lines represent flow velocities on hub (blue) and shroud (green).

Velocities directly before and behind blade area are displayed by dashed lines to show the influence of blockage in the flow domain.

Furthermore the blade angles are displayed by thick lines in order to see the incidence angle on the leading edge and the flow deviation caused by slip velocity on trailing edge.

(2) Values

Numerical values of velocity components and flow angles are displayed in a table. A short description is at mouse cursor too:

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>z</td>
<td>Axial position</td>
</tr>
<tr>
<td>d</td>
<td>Diameter</td>
</tr>
<tr>
<td>αF</td>
<td>Angle of absolute flow to circumferential direction</td>
</tr>
<tr>
<td>βF</td>
<td>Angle of relative flow to circumferential direction</td>
</tr>
<tr>
<td>u</td>
<td>Circumferential velocity</td>
</tr>
<tr>
<td>cm</td>
<td>Meridional velocity (c_m = w_m)</td>
</tr>
<tr>
<td>cu</td>
<td>Circumferential component of absolute velocity</td>
</tr>
<tr>
<td>cr</td>
<td>Radial component of absolute velocity</td>
</tr>
<tr>
<td>ca</td>
<td>Axial component of absolute velocity</td>
</tr>
<tr>
<td>c</td>
<td>Absolute velocity</td>
</tr>
<tr>
<td>w_u</td>
<td>Circumferential component of relative velocity: w_u + c</td>
</tr>
<tr>
<td>w</td>
<td>Relative velocity</td>
</tr>
<tr>
<td>τ</td>
<td>Obstruction by blades (see below)</td>
</tr>
<tr>
<td>i</td>
<td>Incidence angle: i = β_1B - β_1</td>
</tr>
<tr>
<td>δ</td>
<td>Deviation angle: δ = β_2B - β_2</td>
</tr>
<tr>
<td>w_2/w_1</td>
<td>Deceleration ratio of relative velocity</td>
</tr>
<tr>
<td>c_2/c_1</td>
<td>Absolute velocity ratio</td>
</tr>
<tr>
<td>Δα_F</td>
<td>Abs. deflection angle: Δα_F = α_{F2} - α_{F1}</td>
</tr>
<tr>
<td>Δβ_F</td>
<td>Rel. deflection angle: Δβ_F = β_{F2} - β_{F1}</td>
</tr>
<tr>
<td>φ=Δβ_B</td>
<td>Blade camber angle: φ = Δβ_B = β_{B2} - β_{B1}</td>
</tr>
</tbody>
</table>
7.3.1.1 Blade setup

Table: Default blade angles for the optimal Free-form 3D blade shape is displayed compared to the currently specified/calculated angles. Deviations from default values are marked in red color. Default blade angles are calculated based on
- Shockless inflow for β₁ at blade leading edge
- Euler equation for β₂ at blade trailing edge

Some blade angle values result from mean line constraints for simple blade shapes.

For some simplified blade shapes the blade angles of some sections result from the mean line design - see \textit{Blade angles/“Auto”}.

If the mean line design already exists in the component then these dependent angles are calculated automatically for information, otherwise the table cells remain empty.
(1) Selection of desired Blade shape

**3D types:** blade is curved in 3D

<table>
<thead>
<tr>
<th>Free-form 3D</th>
<th>Ruled surface 3D</th>
</tr>
</thead>
</table>
### Radial elements 3D

### Helical 3D

#### 2D (axial) types:
blade is curved in 2D when looking in axial direction

| Free-form 2D (axial) | Circular 2D (axial) |
2D (radial) types: blade is curved in 2D when looking in radial direction

<table>
<thead>
<tr>
<th>Free-form 2D (radial)</th>
<th>Straight 2D (radial)</th>
</tr>
</thead>
</table>

The initial blade shape depends on the machine type and can be customized in the Impeller preferences.

<table>
<thead>
<tr>
<th>PUMP</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Centrifugal &amp; Mixed-flow</td>
<td>Free-form 3D</td>
</tr>
</tbody>
</table>
Only the Free-form 3D blade shape provides complete flexibility, all other types result in limitations in blade angle specification and mean line design.

In case of Ruled surface 3D blade shape and linked splitter blades the linkage can be specified in more detail. See Ruled Surface blade.

**Limitations**

<table>
<thead>
<tr>
<th>Blade shape</th>
<th>Impeller type</th>
<th>Meridional shape</th>
<th>Splitter blades</th>
</tr>
</thead>
<tbody>
<tr>
<td>3D</td>
<td>Free-form 3D</td>
<td>(no limitations)</td>
<td></td>
</tr>
<tr>
<td>Radial &amp; Mixed-flow</td>
<td>Radial elements 3D</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Axial</td>
<td>Free-form 3D</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Radial elements 3D</td>
<td></td>
<td>not available</td>
</tr>
<tr>
<td>Helical 3D</td>
<td>axial impellers only</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
(2) Defining the blade thickness values at leading and trailing edge in panel Blade thickness s

Blade thickness can be important for the blade angle calculation due to the blockage effect and flow acceleration.

By different thickness on hub and shroud side a tapering to the blade tip can be designed. Initial thickness values are based on **empirical functions**\(^\text{[23]}\).

2 impeller types have special thickness requirements:

- **Waste water pumps** have very high thickness values at leading edge to avoid solid attachments (10% of \(d_2\) for 1 blade, 5% of \(d_2\) for more blades). The rest of the blade has smaller thickness of 30% relative to the max. thickness at leading edge.

- **Inducer pumps** have very low thickness values at leading edge to improve suction performance: 6%...10% of normal blade thickness.

**Blade thickness mode**

In general, it's a controversial issue to consider blade blockage effect for blade angle calculation or not and it in which way. Exactly at blade edge the thickness is 0 due to the rounding of the blade edge. Immediately after the blade leading edge (or before the blade trailing edge) the blade is blocking the flow in a certain manner. This blockage is dependent on the blade thickness, the blade angle and the blade angle distribution and which is hence a
rather complex with respect to the blade geometry. One can consider the blockage either by the following thickness modes:

- **Tangential**: the blade thickness is projected tangentially \( \sigma = s / \sin(\beta_{Bl}) \).

- **Orthogonal**: the blade thickness is not projected at all \( \sigma = s \).

- **None**: the blade thickness is not considered \( \sigma = 0 \).

These options will have an influence on the calculation of the meridional velocity component \( c_m \) and therefore on the blade angle calculation when pressing button **Calculate \( \beta_B \)** or if the checkbox **Automatic** is selected. Beyond it, it will influence the meridional flow calculation too.

**(3) Specification of incidence angle on blade leading edge (deviation from shockless inflow) on panel \( \beta_1 \): Incidence**

<table>
<thead>
<tr>
<th>Pump, Fan, Compressor</th>
<th>Turbine</th>
</tr>
</thead>
<tbody>
<tr>
<td>from ratio ( Q ) for shockless inflow / ( Q ) for max. efficiency</td>
<td>fully automatic by theory of <strong>WIESNER</strong> adapted by <strong>Aungier</strong></td>
</tr>
<tr>
<td>( R_Q = Q_{\text{Shockless}} / Q_{\text{BEP}} )</td>
<td>or</td>
</tr>
<tr>
<td>directly by incidence angle ( i )</td>
<td>directly by incidence angle ( i )</td>
</tr>
<tr>
<td>( (R_Q=100% \text{ or } i=0^\circ \text{ for shockless inflow}) )</td>
<td>( (i=0^\circ \text{ for shockless inflow}) )</td>
</tr>
<tr>
<td>or</td>
<td></td>
</tr>
<tr>
<td>from ratio of incidence angle ( i / \beta_B )</td>
<td>( i_{\text{rel}} = i / \beta_B )</td>
</tr>
</tbody>
</table>

For **inducer pumps** there is an additional check if the incidence is > 1° even for high flow rates (overload) to prevent pressure side cavitation.

**Squirrel cage fans** have high incidence typically resulting in blade inlet angles \( \beta_{1B} \approx 80^\circ \).

[ **Pump, Fan, Compressor impellers only** ]

**(4) Estimation of slip velocity in panel \( \beta_2 \): slip**
You have to use one of the following slip models:

<table>
<thead>
<tr>
<th>Slip model theory</th>
<th>Hints</th>
</tr>
</thead>
<tbody>
<tr>
<td>GÜLICH/ WIESNER</td>
<td>closed empirical model, extended Wiesner model</td>
</tr>
<tr>
<td>AUNGIER/ WIESNER</td>
<td>closed empirical model</td>
</tr>
<tr>
<td>VON BACKSTROEM</td>
<td>closed empirical model</td>
</tr>
<tr>
<td>PFLEIDERER</td>
<td>input of coefficient a</td>
</tr>
<tr>
<td>User-defined</td>
<td>manual selection of angular deviation $\beta_2 - \beta_z$ resp. velocity ratio $c_{u_2} / c_{u_2,\infty}$</td>
</tr>
<tr>
<td>Specific definitions</td>
<td>specific slip models for specific impeller types</td>
</tr>
</tbody>
</table>

Using the button **Show calculation details** provides specific information about the $\beta B_2$ calculation.

### Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Blade blockage factor of ... is larger than warning level of ... at leading/ trailing edge at hub/ shroud.</strong></td>
<td></td>
</tr>
<tr>
<td>Blade thickness $s$ is blocking a significant part of the flow passage $u = \pi d / \text{number of blades}$. The blockage factor is calculated as $F = s / u$.</td>
<td>Reduce number of blades and/or blade thickness.</td>
</tr>
<tr>
<td><strong>Blade blockage factor of ... is outside the valid range of 0...1 at leading/ trailing edge at hub/ shroud.</strong></td>
<td></td>
</tr>
<tr>
<td>Blade thickness is blocking the flow passage completely at the specified position.</td>
<td>Reduce number of blades and/or blade thickness.</td>
</tr>
<tr>
<td><strong>Blade number different than initially defined.</strong></td>
<td></td>
</tr>
<tr>
<td>[Wastewater pumps only]</td>
<td>It makes no sense to use other number of blades for main dimension calculation and blade design itself.</td>
</tr>
</tbody>
</table>
### Problem

Before modifying the number of blades here one should adapt the number in **Main dimensions**, update the empirical parameters and the main dimension.

### Possible solutions

**Mean lines (except hub) may be extrapolated.**

(*Free-form 2D* blade shape only)

The hub is the master mean line for "Free-form 2D" blade shape. For this blade shape the geometry of all other mean lines is designed automatically in such way that it is exactly overlapping the hub mean line if viewing in z-direction. The resulting blade shape is two-dimensional.

If the other curves have points with higher radius at trailing edge/ lower radius at leading edge than the last/ first hub point (sloping meridional edge), then these curves have to be extrapolated.

Use axis parallel (const. radius) or slightly sloping meridional leading/ trailing edge.

**Leading edge**: The shroud point should have higher or equal radius than the hub point.

**Trailing edge**: The shroud point should have lower or equal radius than the hub point.

**Blade shape [Radial Elements 3D]: requires the maximum Z-extension of the meridional blade area to be defined on the Hub.**

The hub is the master mean line for "Radial elements 3D" blade shape. The geometry of all other mean lines is designed automatically in such way that it forms a blade consisting of **radial fibers**. The resulting blade shape is three-dimensional.

If the other curves have points with lower z-values at leading edge/ higher z-value at trailing edge than the first/last hub point, these curves have to be extrapolated. In this case the blade would have a bad quality in the extrapolated region.

Use radial (const. axial position) or sloping meridional leading/ trailing edge.

**Leading edge**: The shroud leading edge should have a higher or equal axial position compared to the hub.

**Trailing edge**: The shroud trailing edge should have a lower or equal axial position compared to the hub.

**"Ruled surface" blades may export low quality surfaces when using two mean lines only.**

(*"Ruled surface 3D" blade shape only*)

Impeller with splitter blades can have wavy blade surface if only 2 blade profile sections are used.

Increase the number of blade profile sections (page "Blade angles").

**"Straight 2D (axial)" blades not possible for selected combination of meridional leading edge contour and blade angle.**
<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>The hub mean line is the master mean line. All other mean lines are</td>
<td><strong>Leading edge</strong> ( \beta_B ) The point on shroud should be moved to a higher radius.</td>
</tr>
<tr>
<td>adapted automatically in order to overlap the hub mean line if viewing</td>
<td><strong>( \beta_B )</strong> Blade angle should be increased.</td>
</tr>
<tr>
<td>in z-direction.</td>
<td></td>
</tr>
<tr>
<td>If the other mean lines are extended they will be extrapolated</td>
<td></td>
</tr>
<tr>
<td>automatically. For specific combinations of meridional leading edge</td>
<td></td>
</tr>
<tr>
<td>and blade angles ( \beta_B ) an extrapolation is impossible.</td>
<td></td>
</tr>
<tr>
<td>Leading edge ( \beta_B ) The point on shroud should be moved to a</td>
<td></td>
</tr>
<tr>
<td>higher radius.</td>
<td></td>
</tr>
<tr>
<td><strong>( \beta_B )</strong> Blade angle should be increased.</td>
<td></td>
</tr>
<tr>
<td><strong>Circular + Free-form 2D (axial)</strong> blades not possible for selected</td>
<td>Construction of circular arc is not possible for given parameters. Therefor calculation of blade is</td>
</tr>
<tr>
<td>distance and angle combination.</td>
<td>blocked</td>
</tr>
<tr>
<td>Modify ( \alpha_{LE} ) or ( \alpha_{LE} ) for this blade shape. For further information see <strong>Compound blade shapes</strong></td>
<td></td>
</tr>
<tr>
<td>Extrapolation of &quot;Circular + Free-form 2D (axial)&quot; blades not possible</td>
<td></td>
</tr>
<tr>
<td>for secondary spans.</td>
<td></td>
</tr>
<tr>
<td>The minimal inner radius for the secondary spans is limited by the</td>
<td>Try to reduce effect of extrapolation by adjusting meridional <strong>Leading edge</strong> or change parameters</td>
</tr>
<tr>
<td>circular arc (design curve) defined by ( \alpha_3 ) and ( a_3 ).</td>
<td>defining the circular arc (design curve) of this blade shape. For further information see <strong>Compound blade shapes</strong></td>
</tr>
<tr>
<td>&quot;Straight 2D (axial)&quot; blades not possible for selected combination of</td>
<td></td>
</tr>
<tr>
<td>meridional trailing edge contour and blade angle.</td>
<td></td>
</tr>
<tr>
<td>The blade angle is too small or too large - therefore designing a</td>
<td><strong>Trailing edge</strong> ( \alpha_{LE}/\beta_{LE} ) The edge should be moved to a higher radius.</td>
</tr>
<tr>
<td>&quot;Straight 2D&quot; blade shape is impossible.</td>
<td><strong>( \alpha_{LE}/\beta_{LE} )</strong> Blade angle should be increased.</td>
</tr>
</tbody>
</table>
## Problem

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;Straight 2D (radial)&quot; blades not possible for selected combination of meridional leading edge contour and blade angle</td>
<td>Blade angles not within a valid range.</td>
</tr>
<tr>
<td>Projection of the designed mean line onto the other spans fails for this blade shape.</td>
<td>Blade angle should be specified within the recommended range.</td>
</tr>
</tbody>
</table>
Ruled surface blades are used especially to enable flank milling for manufacturing. The mean surface is generated by spatial movement of a straight line.

When using splitter blades that are linked to main blade then this linkage can be specified in more detail.
You can choose between the following options:

**Exact (adjusts main blade):** The blade geometry of the splitter is forced to be equal to its main blade. Therefore, the leading edge of the splitter needs to be a ruling of the main blade. Due to the flexible choice of the splitter leading edge, this option requires a readjustment of the main blade.

**Mean lines only:** The blade geometry of the splitter is designed using the mean lines of the main blade. The advantage of this option is a higher flexibility in design of a curved leading edge of the splitter. (depends on the number of mean lines)

The following pictures illustrate the combination of different options (splitter is rotated into the main blade for illustration):

- **Splitter linked to Main Blade**
- **2 spans**
- **Exact (adjusts main blade)**

Main and Splitter are using identical rulings. The splitter leading edge is influencing the rulings and therefore the main blade.
- **Splitter linked to Main Blade**
- **2 spans**
- **Mean lines only**

Main and splitter are using their own rulings. The splitter is guided by the hub and shroud mean lines of the main blade only. The resulting splitter shape can slightly deviate from the main blade.

- **Splitter linked to Main Blade**
- **5 spans**
- **Mean lines only**

The splitter is guided by all 5 mean lines of the main blade. The resulting splitter shape is following the main blade and can have a curve leading edge but it's no more a ruled surface.

- **NOT Splitter linked to Main Blade**
- **5 spans**

Main and splitter are using their own rulings. There is no coupling between splitter and main blade. The splitter shape can highly deviate from the main blade.
Radial element blades are used especially with highly loaded fast speed turbines in order to avoid bending stresses within the blades due to centrifugal forces. The blades are composed of radial blade fibres if straight lines can be put into the mean surfaces in a way that they go through the axis of rotation at $z = \text{constant}$.

**Radial element blades require the following geometrical boundary conditions for radial & mixed-flow impellers:**

- Blade angle at input (turbines) or output resp. (all other types): $\beta \approx 90^\circ$
- Inclination angle from hub and shroud to the horizontal: $\varepsilon' < 90^\circ$
- Vertical trailing (turbines) or leading edge resp. (all other types) with $z \approx \text{const}$.
- Small wrap angle: $\varphi \approx 360^\circ/\text{number of blades}$
Compound blade shapes are mainly used for radial diffusers (stator), but are available for centrifugal impellers, too.

**Log. Spiral + Straight 2D (axial)**

The inlet section of the blades without overlapping is configured as a logarithmic spiral. The overlapping part is straight. The transition point between these areas can be moved along the logarithmic spiral curve (see stator mean line).

**Circular + Free-form 2D (axial)**

The inlet section of the blades without overlapping is configured as a circular arc with the boundary conditions radius $r_{LE}$, blade angle $\beta_{LE}$ and throat width $a_{LE}$.

The overlapping part is designed by a Bezier curve with optionally 2 (straight), 3 or 4 Bezier points (selectable by context menu). The transition point between these areas can be moved along the circular arc curve (see stator mean line).

Calculation of throat width $a_{LE}$ can be done using the conservation of angular momentum (const. swirl) or a specific deceleration ratio alternatively:

**a) Constant swirl**

Throat width corresponds to the dimensioning in accordance with the conservation of angular momentum ($c_{u}r_{In}$), whereas the deceleration is increased by using the factor $f_{aLE}$ (1.1...1.3).

\[
a_{LE} = f_{aLE}r_{LE} \left\{ \exp \left( \frac{Q}{2b_{LE}(c_{u}r)_{In}} \right) - 1 \right\}
\]

**b) Deceleration**

Alternatively one can use the deceleration ratio $c_{LE}/c_{In}$ (0.7...0.85) for throat width calculation.

\[
a_{LE} = \frac{Q}{2b_{LE}c_{In}} \left( \frac{c_{In}}{c_{LE}} \right)
\]

Trailing edge angle $\beta_{TE}$ is a result of mean line design for these special blade shapes and therefore cannot be specified explicitly ("var.").
7.3.1.2  Spans

On page Spans number of span and their distribution are defined.

By default the meridional lines are equally spaced between hub and shroud (linear distribution curve). Using the context menu, the curve type can be set to Bezier curve. If this option is chosen two control points can be manipulated to generate a certain span distribution. The span positions are illustrated as meridional lines in the Meridian diagram behind the distribution curve.

7.3.1.3  Blade angles

On this page the blade angles are calculated.
Later designed mean lines depend on the number and the meridional position of profile sections as well as the blade angles. Blade angles $\beta_{B1}$ and $\beta_{B2}$ are calculated from the velocity triangles, whereby the blade blockage of the flow channel and the slip velocity is considered.

The degree of freedom when designing the blades depends on the selected blade shape. Referring to the blade angles this means, that they are marked as (auto) and are result of the Mean line calculation.

**Distribution from hub to shroud**

The blade angles can be calculated on all spans. On panel Distribution from hub to shroud you can define how the blade angles of the inner sections are defined.

**Blade angles $\beta_B$**

- Calculation of blade angles using values from Blade setup by pressing button Calculate $\beta_B$
- Manual adaptation of calculated blade angles if required

Calculation or input of blade angles can be executed for each span (blade profile).

When using 2D blade shapes a low number of profiles may be sufficient in dependence of the leading edge shape, e.g. for a straight leading edge. For that reason the initial design for fans is made by 2 blade profiles.

Blade angles are computed under consideration of the equations listed below. They remain unchanged by default if they are determined once. If main dimensions or meridional contours are modified or, on the other hand, values of blade thickness or slip velocity are renewed, a recalculation of blade angles should be executed by pressing the button **Calculate βB**. This recalculation is made automatically if the checkbox **Automatic** is selected.

Efficiency values that are relevant for the blade angle calculation are displayed for information as hint.

- Details of calculation of **Inlet triangle**
- Details of calculation of **Outlet triangle**

(auto)

For special blade shapes some restrictions are existing and only the blade angles of the master mean line at hub can be calculated or adapted manually. The angles of all other sections are calculated automatically later during the mean line design because they depend on the mean line shape. This fact is indicated by the caption "(auto)" in the table. This means that there is a coupling condition based on the selected blade shape that results in an automatic calculation of the blade angles. The blade angles can be displayed in the mean line dialog in the "**Informational values**" panel.

**Circular blades**

For circular blades the radius of the blade R is displayed beside the blade angle table for information. This radius depends on the radii r₁, r₂ and blade angles β₁, β₂ at leading and trailing edge. If the calculation of the circular blade is not possible a warning symbol is displayed.

**Possible warnings**
<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Automated blade angles are active. Values may adapt to changing input parameters.</td>
<td>To fix the blade angles you could uncheck the &quot;Automatic&quot; calculation. Then you have to manually start the calculation if required.</td>
</tr>
<tr>
<td>Blade angles are updated automatically when input parameters are modified.</td>
<td></td>
</tr>
<tr>
<td>Swirl gradient violates Euler equation. Check blade angles and velocity triangles.</td>
<td>Recalculate and/or check blade angles $\beta_B$ and flow angles $\beta$ at leading and trailing edge.</td>
</tr>
<tr>
<td>$c_{u2}r_2$ is lower than $c_{u1}r_1$ (turbines: $c_{u2}r_2$ is higher than $c_{u1}r_1$) resulting in energy transmission in the wrong direction (Euler equation of turbomachinery).</td>
<td></td>
</tr>
<tr>
<td>$\Delta\beta_{B1/2} \text{ (leading/trailing edge) } = \ldots \text{ is larger than warning level of } \ldots$</td>
<td>Check the resulting 3D blade shape and avoid high blade angle differences on spans if possible.</td>
</tr>
<tr>
<td>Blade angle difference (highest - lowest value) at all spans exceeds the warning level (see Preferences: Warning level). The resulting blade could be highly twisted.</td>
<td></td>
</tr>
<tr>
<td>$\Delta\beta \text{ (span) } = \ldots \text{ is larger than warning level of } \ldots$</td>
<td>Check the resulting 3D blade shape and avoid high blade angle differences between leading and trailing edge if possible.</td>
</tr>
<tr>
<td>$\Delta\beta =</td>
<td>\beta_{B2} - \beta_{B1}</td>
</tr>
<tr>
<td>Blade angles $\beta_{B1/2}$ cannot be determined. Thermodynamic state could not be calculated. Check main dimensions, meridional shape or global setup.</td>
<td></td>
</tr>
<tr>
<td>[ for compressors and turbines only ]</td>
<td></td>
</tr>
<tr>
<td>The dimensions or meridional contour might be too tight for the specified mass flow and inlet conditions.</td>
<td>Increase the dimensions (width etc.), meridional contour or change the Global setup (e.g. decrease mass flow).</td>
</tr>
</tbody>
</table>
The inlet triangle is defined by inflow parameters and geometrical dimensions on leading edge.

Between inlet area and leading edge the swirl is constant because transmission of energy from rotating impeller to fluid occurs in blade area only. Cross sections 0 and 1 (see Main dimensions) are different only due to blockage of the flow channel by blades ($\tau_1$) in section 1. This results in an increased meridional velocity $c_m$.

\[
\tan \beta_1 = \frac{c_{m1}}{w_{u1}}
\]

\[
c_{m1} = c_{m0} \tau_1
\]

\[
\tau_1 = \frac{t_1}{t_1 - \sigma_1} \quad \text{with} \quad t_1 = \frac{\pi d_i}{z}, \sigma_1 = \frac{s_i}{\sin \beta_{1B}}
\]

\[
c_{m0} = \frac{Q}{(\pi d_i b_i)}
\]

\[
w_{u1} = u_1 - c_{u1}
\]

\[
u_1 = \pi d_i n
\]

\[
c_{u1} = c_{u5} \frac{r_5}{r_1} = u_5 (1 - \delta_{r}) \frac{r_5}{r_1} \quad \text{(const. inflow swirl)}
\]

Selected blade angle $\beta_{1B}$ does only indirectly influence the velocity triangle due to blade blockage. Differences between selected blade angle $\beta_{1B}$ and flow angle $\beta_1$ is referred as the incidence angle: $i = \beta_{1B} - \beta_1$

In general an inflow without any incidence is intended ($i=0$). If $i > 0$ the flow around the leading edge shows high local velocities and low static pressure:

- $i > 0$: $\beta_1 < \beta_{1B}$ → stagnation point on pressure side
- $i < 0$: $\beta_1 > \beta_{1B}$ → stagnation point on suction side

A small incidence angle $i$ can be profitable for best efficiency point. Calculation of $\beta_{1B}$ inside CFturbo gives inflow without incidence.
Typical inlet blade angles are:

<table>
<thead>
<tr>
<th></th>
<th>Inlet Blade Angle</th>
<th>Reason</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pumps, Fans</td>
<td>$\beta_{1B} &lt; 40^\circ$ due to best efficiency</td>
<td></td>
</tr>
<tr>
<td>Pumps</td>
<td>$\beta_{1B}$ as small as possible due to cavitation; with regard to efficiency not smaller then $15...18^\circ$</td>
<td></td>
</tr>
<tr>
<td>Compressors</td>
<td>optimal blade angle $\beta_{1B}$ is about $30^\circ$</td>
<td></td>
</tr>
</tbody>
</table>

If the radius of leading edge varies from hub to shroud the blade angle $\beta_{1B}$ does not remain constant. A higher radius on shroud results in a lower value for $\beta_{1B}$ - the blade is curved on leading edge.

### Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Leading edge blade angle $\beta_{B1} &gt; xx^\circ$</strong></td>
<td>Too high values indicate too small inlet cross section. Increase leading edge dimensions</td>
</tr>
<tr>
<td>Unusual high inlet blade angles.</td>
<td>The warning level can be adjusted under Preferences: Warning level</td>
</tr>
<tr>
<td>Vaned Stator downstream swirl differs significantly from defined value at $cu, cm$ specification</td>
<td>Adjust the precursor stator trailing edge blade angles manually or by using the soft button “Set $\alpha_{TE}$” in the blade properties of the stator.</td>
</tr>
<tr>
<td>Unusual low inlet blade angles.</td>
<td>Too small inlet angles indicate too high inlet cross section. Decrease leading edge dimensions</td>
</tr>
<tr>
<td>The warning level can be adjusted under Preferences: Warning level</td>
<td></td>
</tr>
<tr>
<td><strong>Leading edge blade angle $\beta_{B1} &lt; xx^\circ$</strong></td>
<td>A reasonable thermodynamic state could not be calculated @LE. Consider change of blade angles or thickness, main dimensions or global setup.</td>
</tr>
<tr>
<td>Vaned Stator downstream swirl differs significantly from defined value at $cu, cm$ specification</td>
<td>[ for compressors and turbines only ]</td>
</tr>
</tbody>
</table>

[axial turbines only]
<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>The geometry does not allow for the establishment of a physically valid state. E.g. the mass flow is too high.</td>
<td>Adjust the leading edge blade angles or thickness values or main dimensions or the global setup (e.g. mass flow or inlet conditions).</td>
</tr>
<tr>
<td>The blade angles are not within the valid range.</td>
<td></td>
</tr>
<tr>
<td>Usage of CFturbo is limited to inlet angles between 0° and 180°.</td>
<td>Blade angle calculation is impossible (see below) or adjust unsuitable user input for blade angles.</td>
</tr>
<tr>
<td>βB indeterminate. It’s not possible to determine blade angle βB.</td>
<td></td>
</tr>
<tr>
<td>Blade angle calculation failed.</td>
<td>Check input values and geometry.</td>
</tr>
</tbody>
</table>

[ TURBINE ROTORS ONLY ]

In case of turbines the calculation of the incidence by Aungier can be used.

According to decreased energy transmission the slip coefficient \( \gamma \) is defined:

\[
\gamma = 1 - \frac{c_{u1}}{c_{u2}}
\]

7.3.1.3.2 Outlet triangle

The outlet triangle is determined by geometrical dimensions of flow channel and selected blade angle \( \beta_{2B} \). The blade angle \( \beta_{2B} \) strongly affects the transmission of energy in the impeller therefore it has to be chosen very carefully.
For determination of $\beta_{2B}$ it is important to be aware about the deviation between flow angle and blade angle. The direction of the relative flow $w_2$ at impeller outlet does not follow exactly with the blade contour at angle $\beta_{2B}$. The flow angle $\beta_2$ is always smaller than blade angle $\beta_{2B}$ due to the slip velocity. This difference is called deviation angle $\delta$:

The deviation angle should not exceed $10^\circ$...$14^\circ$, in order to limit increased turbulence losses by asymmetric flow distribution.
A reduced flow angle $\beta_2$ results in smaller circumferential component of absolute speed $c_{\psi 2}$, which is - according to Euler's equation - dominant for the transmission of energy. Blade angle $\beta_{2B}$ is estimated by $c_{\psi 2,\infty}$ for blade congruent flow (see figure). Therefore an estimation of slip is necessary.

Slip can be estimated by empirical models. The following models are available in CfTurbom (not for Radial-inflow Turbines):

- Gülich/Wiesner
- Aungier/Wiesner
- Pfleiderer
- Von Backstroem
- Specific definitions

Blade angle $\beta_{2B}$ must be determined to reach the desired energy transmission - respectively the required head/ pressure difference - under consideration of slip velocity.

The following recommendations for common blade angles $\beta_{2B}$ exist due to optimal efficiency:

<table>
<thead>
<tr>
<th>Pumps</th>
<th>15°...45°, commonly used 20°...27°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fans</td>
<td>not higher than 50°</td>
</tr>
<tr>
<td>Compressors</td>
<td>35°...50°, unshrouded impellers up to 70°...90°</td>
</tr>
<tr>
<td>Turbines</td>
<td>radius dependent, see sine rule</td>
</tr>
</tbody>
</table>

Centrifugal machines - except for radial-inflow gas turbines - with low specific speed $n_q$ usually have similar values for $\beta_{2B}$. The blades for this type of impellers are often designed with a straight trailing edge ($\beta_{2B}=$ const.). For radial-inflow gas turbine rotors and for Francis runners the radii along the trailing edge from hub to shroud are very different, resulting in very different values for $\beta_{2B}$ and twisted blades.

Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Trailing edge blade angle $\beta_{2B} &lt; xx^\circ$</td>
<td></td>
</tr>
<tr>
<td>Problem</td>
<td>Possible solutions</td>
</tr>
<tr>
<td>------------------------------------------------------------------------</td>
<td>-----------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Unusual low outlet blade angles.</td>
<td>Too small outlet angles indicate too high outlet cross section. Decrease trailing edge dimensions (Main dimensions)</td>
</tr>
<tr>
<td>The warning level can be adjusted under Preferences: Warning level</td>
<td></td>
</tr>
<tr>
<td>Deviation $\delta$ (slip) between blade and flow is pretty high.</td>
<td>Possible solutions could be: increase the impeller diameter (Main dimensions), increase the number of blades, increase meridional blade length (Meridional contour), select a different slip model</td>
</tr>
<tr>
<td>(pumps, fans, compressors only)</td>
<td></td>
</tr>
<tr>
<td>Unusual high deviation (slip) between blade and flow direction at outlet. This indicates too high blade loading.</td>
<td></td>
</tr>
<tr>
<td>The warning level can be adjusted under Preferences: Warning level</td>
<td></td>
</tr>
<tr>
<td>Trailing edge blade angle $\beta_{B2} &gt; xx^\circ$.</td>
<td>Increase trailing edge dimensions (Main dimensions) and/or the slip coefficient $\gamma$.</td>
</tr>
<tr>
<td>Unusual high blade angles at trailing edge. This can be caused by overloading the impeller.</td>
<td></td>
</tr>
<tr>
<td>The warning level can be adjusted under Preferences: Warning level</td>
<td></td>
</tr>
<tr>
<td>A reasonable thermodynamic state could not be calculated @TE. Consider change of blade angles or thickness, main dimensions or global setup. (+ for compressors and gas turbines only)</td>
<td>Adjust the trailing edge blade angles or thickness values or main dimensions or the global setup (e.g. mass flow or inlet conditions).</td>
</tr>
<tr>
<td>The geometry does not allow for the establishment of a physically valid state. E.g. the mass flow is too high.</td>
<td></td>
</tr>
<tr>
<td>Blade angles are not within the valid range.</td>
<td></td>
</tr>
<tr>
<td>Usage of CFturbo is limited to blade angles between $0^\circ$ and $180^\circ$.</td>
<td>Blade angle calculation is impossible (see below) or adjust unsuitable user input for blade angles.</td>
</tr>
<tr>
<td>No possibility to determine Blade angles $\beta B$.</td>
<td></td>
</tr>
<tr>
<td>Blade angle calculation failed.</td>
<td>Try to increase the impeller diameter $d_2$ or outlet width $b_2$ and/or the slip coefficient $\gamma$.</td>
</tr>
<tr>
<td>Problem</td>
<td>Possible solutions</td>
</tr>
<tr>
<td>---------</td>
<td>--------------------</td>
</tr>
<tr>
<td>Deviation $\delta$ (slip) between blade and flow is too high.</td>
<td>Possible solutions could be: increase the impeller diameter (Main dimensions), increase the number of blades, increase meridional blade length (Meridional contour), select a different slip model</td>
</tr>
</tbody>
</table>

The slip calculation results in an extremely high slip angle, which is unrealistic. The blades could be overloaded or the wrong slip model is used. The error level can be adjusted under Preferences: Warning level.

Possible solutions could be:
- increase the impeller diameter (Main dimensions),
- increase the number of blades,
- increase meridional blade length (Meridional contour),
- select a different slip model.

7.3.1.3.2.1 Slip coefficient by GÜLICH/WIESNER

Outflow (slip) coefficient $\gamma$ is defined for the decreased energy transmission:

$$\gamma = 1 - \frac{c_{u_{2\infty}} - c_{u_{2}}}{u_2}$$

The $c_u$-difference is called slip velocity.

The smaller the outflow coefficient, the higher the deviation of flow compared to the direction given by blade.

Wiesner developed an empirical equation for the estimation of outflow coefficient:

$$\gamma = 1 - \frac{\sqrt{\sin \beta_{2b}}}{z^{0.7}}$$

Gülich modified this formula by two additional correction factors:

$$\gamma = f_t \left( 1 - \frac{\sqrt{\sin \beta_{2b}}}{z^{0.7}} \right) k_w$$

with the correction factors:
Circumferential component of blade congruent flow can be calculated as follows:

\[ c_{u,2o} = c_{u,2} + (1 - \gamma)u_2 \]

Contrary to Wiesner's original suggestion an average inlet diameter \( d_{im} \) is not used for the calculation of \( k_w \) in CFturbo but the diameter at hub and shroud respectively. Doing so a slip coefficient for hub and shroud can be calculated. An average slip coefficient is determined by:

\[ \gamma = 0.5 (\gamma_{Hub} + \gamma_{Shroud}) \]

The switch between radial and mixed-flow calculation of the correction factor \( f_1 \) is done by:

\[ f_1 = \max \left( 0.98, 1.02 + 1.2 \cdot 10^{-3} (t_h - 50) \right) \]

### 7.3.1.3.2.2 Slip coefficient by AUNGIER/ WIESNER

Outflow (slip) coefficient \( \gamma \) is defined for the decreased energy transmission:

\[ \gamma = 1 - \frac{c_{u,2o} - c_{u,2}}{u_2} \]

The \( c_u \)-difference is called slip velocity.

The smaller the outflow coefficient, the higher the deviation of flow compared to the direction given by blade.

Aungier adjusted Wiesner's original empirical equation for the estimation of outflow coefficient:

\[ \gamma = 1 - \frac{\sqrt{\sin \beta_{2B}}}{z^{0.7}} \]

The limiting radius ratio \( \epsilon_{Lim} \) is given by:

\[ \epsilon_{Lim} = \frac{r_2}{r} > \epsilon_{Lim} \]

The slip factor is corrected for radius ratios \( \epsilon = r/r_2 > \epsilon_{Lim} \) with:
\[
\gamma_{\text{cor}} = \gamma \left( 1 - \left( \frac{\varepsilon - \varepsilon_{\text{lim}}}{1 - \varepsilon_{\text{lim}}} \right)^{\beta_{28:10}} \right)
\]

[Compressors only]

The model is further adjusted in case it is applied to splitter blades. Then the number of blades in the above equation is corrected by the relative splitter blade length with respect to the main blade length.

\[
z_{\text{cor}} = z_{mB} + z_{sB} \frac{L_{sB}}{L_{mB}}
\]

Circumferential component of blade congruent flow can be calculated as follows:

\[
c_{u2\infty} = c_{u2} + (1 - \gamma) u_2
\]

7.3.1.3.2.3 Slip coefficient by PFLEIDERER

Reduced energy transmission is expressed by decreased output coefficient \(p\):

\[
p = \frac{\bar{Y}_c}{\bar{Y}} - 1
\]

This coefficient can be empirically calculated in dependence of experience number \(\psi'\):

\[
p = \psi' \frac{r_2^2}{zS}
\]

\[
S = \int_{r_1}^{r_2} r dx
\]

static moment from leading to trailing edge

experience number

experience number \(a\):

Centrifugal impeller with guided vanes \(a = 0.6\)
with volute \[ a = 0.65 \ldots 0.85 \]

with plain diffusor \[ a = 0.85 \ldots 1.0 \]

Mixed flow/axial impeller \[ a = 1.0 \ldots 1.2 \]

(the numbers are valid for sufficiently high Re; \( \psi' \) strongly grows with small Re)

More descriptive is the decreased output factor \( k_L \):

\[
k_L = \frac{1}{1 + p} = \frac{\tilde{Y}}{Y_\infty} = \frac{\Delta(c_u)}{\Delta(c_u)_\infty}
\]

\((k_L=1: \text{for flow congruent to blade})\)

Circumferential component of the flow, which is congruent to blade, can be calculated as follows:

\[
c_{u2\infty} = \frac{c_{u2}}{k_L} - \frac{r_1^2}{r_2}\left(\frac{1}{k_L} - 1\right)2\pi(1 - \delta_r)
\]

Now the outflow (slip) coefficient \( \gamma \) according to Wiesner can be calculated:

\[
\gamma = 1 - \frac{c_{u2\infty} - c_{u2}}{u_2}
\]

7.3.1.3.2.4 Slip coefficient by VON BACKSTROEM

Outflow (slip) coefficient \( \gamma \) is defined for the decreased energy transmission:

\[
\gamma = 1 - \frac{c_{u2\infty} - c_{u2}}{u_2}
\]

The \( c_u \)-difference is called slip velocity.

The smaller the outflow coefficient, the higher the deviation of flow compared to the direction given by blade.
Von Backstroem developed an empirical equation for the estimation of the outflow coefficient assuming one single relative eddy in the rotor.

\[
\gamma = 1 - \frac{1}{F_0 \cdot \text{sol} \cdot \sqrt{\sin(\beta_{2B})}}.
\]

Here \( \text{sol} \) is the solidity defined by:

\[
\text{sol} = \frac{1 - \varepsilon \cdot \bar{z}}{2\pi \cdot \sin(\beta_{2B})}.
\]

The limiting radius ratio \( \varepsilon_{\text{Lim}} = 0.5 \), the radius ratio \( \varepsilon = \max(r_1 / r_2, \varepsilon_{\text{Lim}}) \). The constant \( F_0 = 5 \).

### 7.3.1.3.2.5 Specific slip coefficient definitions

#### Waste water pumps (GÜLICH)

For waste water pumps the slip mainly depends on the number of blades.

The table contains typical values for the slip coefficient \( \gamma \):

<table>
<thead>
<tr>
<th>number of blades</th>
<th>slip coefficient ( \gamma )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.48 ... 0.6</td>
</tr>
<tr>
<td>2</td>
<td>0.53 ... 0.65</td>
</tr>
<tr>
<td>3</td>
<td>0.67 ... 0.75</td>
</tr>
</tbody>
</table>

#### Inducer pumps (GÜLICH)

For inducer pumps the deviation angle depends on the blade angles on leading and trailing edge and the solidity.

\[
\delta = \left\{ 2^2 \cdot \left( \frac{\beta_{2B}}{3} \right) \cdot \left( \frac{\beta_{1B}}{L} \right) \right\}^{1/3}.
\]

### 7.3.1.3.2.6 Calculation details

Circumferential component of blade congruent flow can be calculated as follows:
\[ c_{u_{2\infty}} = c_{u_2} + (1 - \gamma)u_2. \]

The slip coefficient is a function of blade angle, number of blades and the meridional geometry (see e.g. Güllich/Wiesner etc.):

\[ \gamma = f(\beta_{2B}, z, \ldots). \]

The relation of the blade angle to the velocity components is:

\[ \tan(\beta_{2B}) = \frac{c_{m_2}}{u_2 - c_{u_{2\infty}}}. \]

A swirl has to be produced by the impeller in accordance to the design specific work (here without pre-swirl) with the Euler equation:

\[ Y = c_{u_2} \cdot u_2. \]

These equations build a set that cannot be solved explicitly but numerically. To this an equation representing the difference of the slip coefficient according to the definition and according to the particular model can be used:

\[ 0 = \Delta \gamma = \frac{c_{m_2}}{u_2 \tan(\beta_{2B})} \cdot \frac{Y}{u_2} - f(\beta_{2B}, z, \ldots). \]

One can test this equation with different values of \( \beta_{2B} \) and will get a function of the form \( \Delta \gamma = f(\beta_{2B}) \). This function together with \( \gamma = f(\beta_{2B}) \) according to the definition of slip coefficient is displayed for hub and shroud. Also, points of interest apart from zero are illustrated such as minimum and maximum and inflection. The equation is fulfilled at zero.
7.3.2 Blade mean lines

The blade mean lines are designed on the number of meridional flow surfaces which were determined in Blade properties.

Depending on the selected blade shape (see Blade properties) the design of the mean lines is more or less restricted.

The blades of an impeller representing a deceleration cascade for the relative velocity. Therefore the risk of flow separation exists. The user should try to obtain a continuous, smooth change of flow.
direction, as well as the cross section graduation of the flow channel should be as steady as possible.

**Splitter blades**

The splitter blades are displayed and designed on a separate tab (Splitter blade).

The design options depend on the link between main and splitter blades in the Blade properties. If Splitter blade linked to Main blade is activated there, the splitter blade is a shortened main blade. The blade and wrap angles are calculated automatically.

The relative position of the splitter blade between two main blades can be adjusted. In case of linked splitter a single value can be specified only, for unlinked splitter the full flexibility is available.

**Additional views**

Some more blade information is displayed in tables and diagrams in order to check the design and for informational purposes:

- See Additional views

**Design mode**

Select the currently available mode to design the blade mean lines.

- See Design mode

**Coupled linear (Only for Freeform 3D blades)**

For continuous transition between the separate mean lines (blade surface), the matching points of each mean line have to be Coupled linear. If you deactivate this option then you can modify all mean lines independently.

If the linear coupling mode is active you can move and rotate the connecting line. The positions of Bezier points of all mean lines are modified correspondingly, to get uniform profiles. If you select a point of the inner cross sections you can move the entire connecting line.

**Angular positions**

Wrap angle $\Delta \varphi$ (common/ average)
The current average wrap angle of all mean lines is displayed. When design mode = conformal mapping, this value can be modified or reset to default value resulting in the same value for all mean lines, based on empirical functions. The wrap angle of each mean line is given in the table.

Stacking

The stacking position is the relative position at which the mean line is stacked. It can be set only for blades having more than 1 active mean line, e.g. Freeform 3D or Ruled surface 3D. In case of design mode = conformal mapping, the stacking position is zero and cannot be altered currently.

The location of the stacking curve at which the stacking position is applied is determined by the φ-position given in the table. It therefore determines - together with the stacking position - the location of the Leading edge / Trailing edge. For some blade shapes, user defined values can be specified, either directly in the table or using a progression dialog (buttons above the columns).

Shown below are examples with a straight stacking curve located at a stacking angle of 45°. If the stacking angle is different on each span, the stacking curve is not linear. The mean lines are stacked at this curve at the given stacking position:

0% = leading edge 50% = mid 100% = trailing edge
Rake angle

The rake angle $\alpha$ is the angle between the meridional plane and the leading or trailing edge respectively. The following pictures depict a blade with zero rake angle (left) and $\alpha = -20^\circ$ (right).

\[
\tan(\alpha) = \frac{b}{n}
\]
The rake angle can be set directly when the blade shape is either Free-form 3D or Ruled Surface 3D. In case of Design model conformal mapping rake angles at leading and trailing edge can be set both. For different design modes only one rake angle can be set. The other one is determined by the geometric restrictions.

### Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Blade angles and blade extensions could lead to unusual blade shapes.</strong></td>
<td></td>
</tr>
<tr>
<td>The values of the blade angles $\beta_1$, $\beta_2$ and the meridional and tangential blade extension most likely result in an abnormal or strange blade shape. To avoid any subsequent problems such mean line shapes are blocked.</td>
<td>In these cases the blade is highly curved or has a S-shape. To design a reasonable blade the wrap angle has to be not too low and not too high. You can a) modify the blade wrap angle $\varphi$ (checking the blade overlapping) or</td>
</tr>
<tr>
<td><strong>Blade angles and blade extensions could lead to non-feasible blade shapes.</strong></td>
<td></td>
</tr>
<tr>
<td>Problem</td>
<td>Possible solutions</td>
</tr>
<tr>
<td>---------</td>
<td>--------------------</td>
</tr>
<tr>
<td>b) modify the blade angles $\beta_{B1}$ and $\beta_{B2}$ (probably the main dimensions have to be adapted)</td>
<td></td>
</tr>
<tr>
<td>$\Delta\beta_{B1/2}$ (leading/trailing edge) is higher than warning level</td>
<td></td>
</tr>
<tr>
<td>Blade angle difference (highest - lowest value) at all spans exceeds the warning level. The resulting blade could be highly twisted.</td>
<td>Check the resulting blade shape and avoid high blade angle differences on spans if possible.</td>
</tr>
<tr>
<td>$\Delta\beta_{B1/2}$ (leading/trailing edge) is higher than error level</td>
<td></td>
</tr>
<tr>
<td>Blade angle difference (highest - lowest value) at all spans exceeds the error level. Blade design based on these extreme values makes no sense.</td>
<td>Decrease the blade angle differences on spans.</td>
</tr>
<tr>
<td>Blade calculation failed due to boundary conditions and constraints.</td>
<td></td>
</tr>
<tr>
<td>Projection of the design mean line onto the other spans fails for this blade shape.</td>
<td>Decrease wrap angle.</td>
</tr>
<tr>
<td>Very high tangential leading edge sweep angle.</td>
<td></td>
</tr>
<tr>
<td>Leading edge sweep angle (tangential difference between hub and shroud meanline at LE) is very high. This curved shape could result in abnormal or strange blade shape.</td>
<td>Adapt blade properties, e.g. blade angles. In some situations, it might be helpful to increase the number of spans if possible.</td>
</tr>
<tr>
<td>Too high tangential leading edge sweep angle.</td>
<td></td>
</tr>
<tr>
<td>Problem</td>
<td>Possible solutions</td>
</tr>
<tr>
<td>-----------------------------------------------------------------------------------------</td>
<td>----------------------------------------------</td>
</tr>
<tr>
<td>Leading edge sweep angle (tangential difference between hub and shroud meanline at LE) is too high. A reasonable blade cannot be generated.</td>
<td>Adapt <a href="#">blade properties</a>, e.g. <a href="#">blade angles</a>.</td>
</tr>
<tr>
<td><strong>Blade edge exceeds the meridional boundaries.</strong></td>
<td></td>
</tr>
<tr>
<td>The meanlines of inner blade spans are crossing the meridional extents at leading or trailing edge.</td>
<td>Change meridional position of leading/ trailing edge or reduce number of spans to 2.</td>
</tr>
<tr>
<td>This is only possible for ruled surface blades with more than 2 spans.</td>
<td></td>
</tr>
<tr>
<td><strong>Blade angles βB1 and/or βB2 different compared to blade properties.</strong></td>
<td></td>
</tr>
<tr>
<td>Current blade angle values deviate from the specified values in <a href="#">blade properties</a>. This is possible for imported geometry only.</td>
<td>Check imported m,t-curves or β-curves and compare with specified β values at leading and trailing edge. The values resulting from the current meanlines are displayed in &quot;Additional views/ Informational values/ Blade angle βB&quot;.</td>
</tr>
<tr>
<td><strong>Overlapping of adjacent blades might be too low/ high.</strong></td>
<td></td>
</tr>
<tr>
<td>Unusual blade overlapping, which is defined by the overlapping factor F = Wrap angle Δφ/ Pitch angle t (t = 360°/ number of blades)</td>
<td>Modify the blade wrap angle Δφ and/ or the number of blades (see <a href="#">blade angles</a>).</td>
</tr>
<tr>
<td>The min./ max. warning level is defined in Preferences: <a href="#">Warning level</a>.</td>
<td></td>
</tr>
<tr>
<td><strong>Coupling partially deactivated. Blade surface deformation can occur.</strong></td>
<td></td>
</tr>
<tr>
<td>The mean lines are currently not linearly coupled, which can result in deformed blade surfaces.</td>
<td>Activate linear coupling if it is deactivated.</td>
</tr>
<tr>
<td>Either linear coupling has been deactivated or it is impossible because of highly deviating blade angle values.</td>
<td>Homogenize βB2 blade angle values (see <a href="#">blade properties</a>).</td>
</tr>
<tr>
<td>The warning occurs because the intersection of βB2 line and intersection line for one or more</td>
<td></td>
</tr>
<tr>
<td>Problem</td>
<td>Possible solutions</td>
</tr>
<tr>
<td>---------</td>
<td>--------------------</td>
</tr>
<tr>
<td>mean lines cannot be determined. Usually this has one of the following causes:</td>
<td></td>
</tr>
<tr>
<td>a) It is geometrically impossible to determine this intersection (approximate parallel lines).</td>
<td></td>
</tr>
<tr>
<td>b) The intersection is not between the points of hub and shroud mean line.</td>
<td></td>
</tr>
<tr>
<td>c) The point of intersection is too close to the endpoints of the mean line (lower than 5%).</td>
<td></td>
</tr>
</tbody>
</table>

**Curvature very high. Swirl change might not be as high as intended.**

One or more mean lines do not change their direction smoothly enough in the m,t coordinate system resulting in partially high curvature ($\beta_B$ gradient).

The flow does not follow this high curvature and thus the swirl change is much lower as intended. In addition, the blade profile design cannot follow the hook-shaped mean lines.

Design the mean line progression from leading to trailing edge more smoothly by modification of the inner control points and the wrap angle. In particular, the blade angle progression should be checked.

Very high difference between $\beta_{B1}$ and $\beta_{B2}$ can make the solution of the problem more difficult and could require a higher wrap angle.

### 7.3.2.1 Design mode

The mean lines can be designed by alternative methods, which can be selected on panel **Design mode**.

Each method has its specific advantages and disadvantages. The choice depends, among other things, on the user experience. When switching from one method to another, the software tries to maintain the geometry as much as possible.

Currently available design modes:

- t, m Conformal mapping
- $\beta$ Blade angle
- B' Blade loading
- z, T Peripheral projection
7.3.2.1.1 \( t, m \) Conformal mapping

The blade is designed in the conformal \( t, m \) mapping by Bezier curves.

The spatially curved meridional flow surfaces are mapped to a plane by coordinate transformation. This coordinate system has the angle in circumferential direction \( t \) as abscissa and the dimensionless meridional extension \( m \) as the ordinate.

Both quantities are created by the reference of absolute distances in meridional (M) and tangential direction (T) to the local radius \( r \):

\[
\frac{dm}{r} = \frac{dM}{r} \quad \frac{dt}{r} = \frac{dT}{r}
\]

\[
\tan \beta = \frac{dm}{dt}
\]

By default the mean lines are represented by 3\textsuperscript{rd} order Bezier curves (4 control points):

- 2 points define the endpoints at leading edge (left) and trailing edge (right). These points can be moved horizontally only, because their meridional position \( m \) is already fixed.
  CFturbo's primary design is fixing the leading edge point for all cross sections at tangential coordinate \( t=0 \) and meridional coordinate \( m=0 \), while the trailing edge point is determined by the meridional extension \( \Delta m \) (defined in the Meridional contour design step) and the wrap angle \( \Delta \varphi \). The initial wrap angle \( \Delta \varphi \) is based on empirical functions.

- 2 inner points define the inner shape of the meanline. These inner points can be moved along the dashed lines only, which represent the blade angles \( \beta_{B1} \) (leading edge) and \( \beta_{B2} \) (trailing edge) specified in the Blade properties design step.
  If you select any point of the inner (hub) or outer (shroud) cross section, you can rotate the related coupling line between the spans around the opposite end point.
  If you select any point of the inner spans, you can move the complete coupling line in horizontal direction.
The table Angular positions contains the values of the tangential leading edge position $\theta_0$ and the wrap angle $\Delta \theta$. A distribution of these values along the span positions can be specified using the button above the table.

The context menu of the curves contains additional options:

- **Bezier curve**
- **Central Bezier point**
- **Constant $\beta_B$ at leading edge**
- **Constant $\beta_B$ at trailing edge**

- Bezier curve/ Central Bezier point:
  An additional central Bezier point can be activated, which can be moved freely and provides
more flexibility.

- **Bezier curve/ Constant $\beta_1$ ($\beta_2$) near leading (trailing) edge:**
  An additional control point at leading and/ or trailing edge is added, which defines a straight line with constant $\beta_1$/$\beta_2$.
  The exact position can be defined by the point context window as $r$, $z$, or $m$ value.

- **Polyline/ Load curve:**
  After switching the curve mode to polyline, a user-defined polyline can be loaded.

Also, a mean line can be loaded from the profile manager. In this case the dimensionless mean line selected in the profile manager is staggered and scaled with the properties of the reference mean line to be exchanged.
The visibility of the inner mean lines can be toggled via "Inner spans".

7.3.2.1.2 β Blade angle

The blade is designed by means of a blade angle distribution with the help of Bezier curves.

The x-values of the Bezier control points are defined with respect to certain x-axis definitions. These definitions can be selected below the design mode selection and will determine the x-axis definition in the design diagram too.

The endpoints of the curves are completely fixed by their meridional coordinate specified in the Meridional contour design step and the blade angles $\beta_{B1}$ (leading edge) and $\beta_{B2}$ (trailing edge) specified in the Blade properties design step.

The inner Bezier points can be moved without any restrictions.

In the area Angular positions, the stacking position can be defined. Only at this position the tangential coordinate $\theta$ can be specified directly, all other points along the meanline are calculated automatically. Therefore, the blade wrap angle of each span is a result of the given blade angle progression.

The context menu of the curves contains additional options:
• **Bezier curve/ Add/ Remove Bezier point:**
  The number of Bezier points can be modified, which provides high flexibility.

• **Bezier curve/ Constant $\beta_{B1}$ ($\beta_{B2}$) near leading (trailing) edge:**
  An additional control point at leading and/or trailing edge is added, which defines a horizontal line and hence a region with constant $\beta_{B1}$ / $\beta_{B2}$.
  The exact position can be defined by the point context window as $r$, $z$, or $m$ value.

• **Polyline/ Load curve:**
  After switching the curve mode to polyline, a user-defined polyline can be loaded.

  The visibility of the inner mean lines can be toggled via "Inner spans".

7.3.2.1.3 B' Blade loading

The blade is designed by a blade loading distribution $B'(M)$ and can be therefore seen as inverse design. $B'$ is defined as:
with the following components:

- $c_u$: Circumferential abs. velocity
- $r$: Radius
- $M$: Meridional coordinate (not $m'$)
- $\omega$: Angular speed

By integrating $B'$ w.r.t $M/M_{\text{max}}$ one gets the swirl $c_{u2}r^2$ that is to say the Euler work. The $B'$ curve is consisting of 3 parabola. The inner parabola's third coefficient is zero which results in a linear piece of the curve. Parameters of the curve are the absolute values of $B'$ at LE and TE, the M-coordinate of the inner control points and the slope of the inner linear piece of the curve. The dotted line is parallel to the linear piece and can be seen as a see-saw to adjust the slope. There is a corresponding set of meridional velocities to each $B'$ curve coming from the solution of the meridional flow calculation. The result of the definition of the $B'$ curve is a certain $c_u$-distribution that will be used for the calculate the relative $\beta_F$ (impeller) and the absolute flow angle $\alpha_F$ (stator) resp.:

$$\tan(\beta_F) = \frac{c_m}{\omega \cdot r - c_u},$$

$$\tan(\alpha_F) = \frac{c_u}{c_m}.$$  

Using the information of incidence and deviation from the blade properties the absolute flow angle $\beta_B$ (or $\alpha_F$ for stators) is determined.

The value $B'$ at TE is zero to obey the Kutta-condition. Therefore, the associated control point is fixed. After switching from a different design mode to blade loading, control points of the see-saw as well as of the LE are automatically fitted to get a geometry close to the one before the design mode change. Only active spans are converted that way. All blade loading curves of dependent spans are calculated based on the resulting geometry (see blade-to-blade flow 1D). That is why deviations from the Kutta-condition may occur.
In the area Angular positions the stacking position can be defined. Only at this position the tangential coordinate $\theta$ can be specified directly, all other points along the meanline are calculated automatically. Therefore, the blade wrap angle of each span is a result of the given blade loading.

7.3.2.1.4 $z$, $T$ Peripheral projection

[ for Axial impellers & Radial element blades only ]

The blade is designed in its peripheral projection (axial and circumferential coordinates) by Bezier curves.

The leading edge points of all spans are visualized by single dots in order to enable detailed leading edge design.
7.3.2.1.5 Fixed

All mean lines are completely determined by simple blade shape and blade angles (see Blade properties).

Therefore, there is no flexibility in meanline design in this situation.
7.3.2.2 Additional views

The following information can be displayed in the mean line dialog using the "Additional views" button:
The display of the curves can be toggled by the check boxes that are accessible via in the lower corner on the left. In case of splitter those curves of main and splitter blades can be hidden/shown. In case separate curves for suction and pressure side are existing their visibility can be toggled too.

Special display option for splitter blades:
The display of main and splitter curves can be toggled by the check boxes independently.

The visibility of the inner mean lines can be toggled via "Inner spans".

Informational values
The tables contain additional values for information.

Blade passage area
Progression of the blade passage area within a channel built by two neighboring mean surfaces as well as hub and shroud. Additionally for compressible fluids the critical area can be displayed.

Isentropic Mach number
The isentropic Mach number is calculated on the basis of the local total state, on the mass flow according to the design point and on the Blade passage area. Here perfect gas behavior is assumed. The flow may be choked and can be subsonic or supersonic according to the settings in the inlet and outlet of the main dimensions.

[ Only compressible fluids and stators ]

3D-Preview
3D model of the currently designed mean surface as well as surfaces of hub and shroud.

x, y frontal view
The Frontal view represents the designed mean lines in a frontal view, including diameters $d_1$ and $d_2$.

t, m conformal mapping
The spatially curved meridional flow surfaces are mapped to a plane by coordinate transformation. This coordinate system has the angle in circumferential direction t as abscissa and the dimensionless meridional extension m as the ordinate.

See t, m conformal mapping
**z, T unwrapped view**  
Peripheral projection of all mean lines \((z = \text{axial coordinate, } T = \text{circumferential coordinate})\).

[**Axial impellers only**]

**βB blade angle**  
\(β_B\) progression along every mean line.  
Too high local extreme values should be avoided if possible.

Additionally \(β_F\) progression can be displayed (see display options). Those relative flow angles are calculated on the basis of the velocity triangles determined in Blade-to-blade flow 1D.

**βB blade angle (meridian)**  
\(β_B\) progression projected in meridional surface.

**t wrap angle**  
Progression of tangential coordinate \(t\) along every mean line.

**t wrap angle (meridian)**  
Progression of tangential coordinate \(t\) projected in meridional surface.

**λ lean angle**  
Distribution of the lean angle \(λ\). The blade lean angle can be manipulated only indirectly.

With the lean angle the quasi-orthogonal of the blade leans away from the \(z\)-direction. The quasi-orthogonal is a straight line connecting corresponding points on hub and shroud mean line. These lines are setup in the blade properties dialog and are displayed in the meridional cut if just two mean lines were chosen. Otherwise the quasi-orthogonal is not displayed but internally determined by connecting corresponding points on hub and shroud mean line.

➔ See Blade lean angle

**Blade surface values**  
➔ See Blade surface values
Blade-to-blade flow 1D  → See Blade-to-blade flow 1D

Blade-to-blade flow 2D  → See Blade-to-blade flow 2D

7.3.2.2.1 Informational values

The tables contain additional values for information:

**Radial diffuser [ Stator type "Radial diffuser" only ]**

Various values to verify the quality of the diffuser design.
→ see Mean line design for “Radial diffuser” stator type

**Blade passage**

Throat area between neighboring mean surfaces.
This value depends on the number of blades, the wrap angle and the blade shape.

**Circular blade**

Radius, sector angle, center point, leading edge point, trailing edge point of circular arc.

**Lean angle**

Lean angle values at leading (λ₁) and trailing edge (λ₂).
→ see Blade lean angle

**Blade loading [ Pump impeller only ]**

Blade loading estimation with lift coefficient (Guelich):
and with the effective blade loading (Gülich):

\[
\varepsilon_{\text{eff}} = \frac{2\pi \cdot \psi \cdot u_2}{\eta_h \cdot z \cdot \beta_{\text{HL}} / d_2 \left( w_1 + w_2 \right)} \quad \varepsilon_{\text{range}} = \left( \frac{40}{nq} \right)^{0.77} \pm 15\%
\]

**Blade angle**

Table with the blade angles \( \beta_B \) calculated in the Blade properties dialog or computed due to simple blade shapes.

**Blade angle in x-y**

Table with the blade angles of the frontal view \( \beta_{B,xy} \).

In case of strictly radial blades these values are consistent with the blade angles \( \beta_B \).

**Blade angle with sine rule [Turbine rotors only]**

Calculated blade angle using the sine rule.

For every mean line the calculated angles as well as their differences to the actual blade angles are given in a table.

→ see Sine rule

**Blade solidity**

Ratio between blade length (in different definitions) and pitch (e.g. \( \pi d_V / z \)).

**Other information**

Table with:

- mean line length \( l_{ML} \), i.e. the length in a 3D cartesian coordinate system.
- resulting angles of overlapping \( \varphi_B \) of 2 neighboring blades
- incidence angle \( i \) for hub and shroud
7.3.2.2.2 \( t, m \) conformal mapping

The spatially cuned meridional flow surfaces are mapped to a plane by coordinate transformation. This coordinate system has the angle in circumferential direction \( t \) as abscissa and the dimensionless meridional extension \( m \) as the ordinate.

Both quantities are created by the reference of absolute distances in meridional \( (M) \) and tangential direction \( (T) \) to the local radius \( r \):

\[
dm = \frac{dM}{r} \quad dt = \frac{dT}{r}
\]

\[
\tan \beta = \frac{dm}{dt}
\]

- Special display option for splitter blades:
  With "Splitter blade relative to main blade" checked, corresponding mean lines (splitter and main) have the same maximum \( m \)-value. Otherwise all mean lines have the same maximum \( m \)-value as the main blade's hub mean line.

7.3.2.3 Blade lean angle

For each point \( p \) of a mean line on a meridional flow surface a 2D coordinate frame is given by the circumferential direction \( t \) and the meridional direction \( m \). The \textbf{blade angle} \( \beta \) equals the tangent angle of the mean line in this frame. For the point \( p \), this direction corresponds to the intersection of the mean surface with the \( m,t \)-plane, locally. By completing the frame in 3D with an orthogonal direction \( n \) (being perpendicular to \( t \) and \( m \)), other plane intersections with the mean surface can be analyzed.
While **blade angle** $\beta$ only depends on the mean line itself, **blade lean angle** $\lambda$ is measured in $n,t$-plane and gives an information about the slope between mean lines in circumferential direction.
Using the surface normal direction $n_{MS}$ on mean surface the **blade lean angle** $\lambda$ is calculated from the ratio of the portion in $n$ and $t$:

$$\lambda = \arctan \left( -\frac{n_{MS,n}}{n_{MS,t}} \right)$$

With an example of a compressor some means for the manipulation of the blade lean angle are given:

- $\lambda_1 \uparrow$: **blade angle** $\beta_{B1} \downarrow$
- $\lambda_1 \uparrow \downarrow$: move **second Bezier point** at leading edge
- $\lambda_1 \uparrow$: **wrap angle** $\uparrow$
- $\lambda_1 \uparrow$ and enlargement of the curvature: reduction of the meridional extension of the **meridional contour**
7.3.2.2.4 Blade surface values

<table>
<thead>
<tr>
<th>w</th>
<th>Relative velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>cu</td>
<td>Abs. circumferential velocity</td>
</tr>
<tr>
<td>cm</td>
<td>Meridional velocity</td>
</tr>
<tr>
<td>cu + r</td>
<td>Swirl</td>
</tr>
<tr>
<td>Bl'</td>
<td>Derivative of swirl</td>
</tr>
<tr>
<td>w</td>
<td>Relative velocity</td>
</tr>
<tr>
<td>p</td>
<td>Static pressure</td>
</tr>
<tr>
<td>Blp</td>
<td>Pressure based blade loading</td>
</tr>
<tr>
<td>Blv</td>
<td>Velocity based blade loading</td>
</tr>
</tbody>
</table>

![diagram](image.png)
Determination of velocity distribution on impeller blades by Stanitz & Prian

Stream lines must be known a-priori (see Meridional flow calculation). If the meridional flow calculation failed, the blade surface values cannot be calculated and the diagram will not be available. Stream lines rotated around z-axis build stream surfaces. The relative velocities will be calculate in a blade-to-blade section, that is encapsulated by two adjacent stream surfaces. Single values of relative velocities will be determined at \( r = \text{constant} \). Before that an average velocity is calculated on the basis of the continuity equation:

\[
\overline{w} = \frac{\dot{m}}{\overline{P} \cdot A}.
\]

The part mass flow is a function of the entire mass flow, number of blades and number of stream lines. Between two adjacent stream surfaces there is always the same mass flow.

The cross section is determined by stream line distance \( \Delta h \), the radius \( r \), the tangential distance between pressure and suction side of two neighboring blades \( \Delta t \) and by a mean relative flow angle:

\[
A = r \cdot \Delta t \cdot \Delta h \cdot \sin(\overline{\beta})
\]

In the picture above a section according to the following equation is displayed:
With the assumption of zero circulation of the absolute flow within a stream surface (green surface) the relative velocity at the suction side can be calculated by:

\[
w_{ss} = \frac{\sin(\beta_{ps}) \sin(\beta_{ss})}{\sin(\beta_{ps}) \sin(\beta_{ss})} \left( \frac{2\bar{w}}{\sin(\beta_{ps})} \right) + u \cdot \left( \cot(\beta_{ps}) - \cot(\beta_{ss}) \right) + \frac{\partial (c_u \cdot r \cdot \Delta t)}{\partial m},
\]

here \( u \) is the local circumferential velocity, \( c_u \) is the circumferential component of the absolute velocity, \( \beta_{ss} \) and \( \beta_{ps} \) are the blade angles at suction and pressure side respectively. Due to the fact that mean relative velocity is an averaged value of \( w_{ss} \) and \( w_{ps} \), the relative velocity at the pressure side can be calculated with:

\[w_{ps} = 2 \cdot \bar{w} - w_{ss} \cdot \]

**Annotation**

The continuity equation has to be solved iteratively for the relative velocity since the density of a compressible medium is determined by the relative velocity. The density can be calculated from isentropic relation:

\[\rho = \rho_0 \left( \frac{P}{P_0} \right)^{\frac{k-1}{k}} \]

The average relative flow angle is approximated by the average value of the blade angle at suction- and pressure side. At a certain radius the assumption applies that due to the slip (decreased power)
the flow cannot be considered as blade congruent anymore. The mean relative flow angle will be
corrected by the slip at loci with a radius bigger than this Stanitz-Radius.

The whole procedure is based on the assumption that the flow is considered as frictionless and that
shocks as well as heat transport across boundaries do not occur. There might by geometric
constellations where the cross section (blue surface in the images above) is too small for the mass
flow specified in the global setup\footnote{**107**}. If this happens the equation can't be solved for the average
density and relative velocity and no data is displayed for the respective span.

**Blade loading**

Static pressures at suction and pressure side can be determined by the velocities. To this end a
relation between the enthalpy difference between suction and pressure side and the meridional
derivative of the swirl is used:

\[
\hat{h}_{ps} - \hat{h}_{ss} = \frac{2\pi}{n} \cdot \frac{c_m \cdot \partial (\cdot \bar{c}_u)}{\partial m}
\]

The blade loading can be expressed in terms of the pressure difference between suction and
pressure side and divided by the total inlet pressure:

\[
BLp = \frac{p_{ps} - p_{ss}}{p_{in,total}} \cdot \frac{p_{sv}}{p_{in,total}} \left( \frac{2\pi}{n} \cdot \frac{c_m \cdot \partial (\cdot \bar{c}_u)}{\partial m} - \bar{c}_v \left( t_{ps} - T_{ss} \right) \right)
\]

For incompressible fluids the second term within the brackets is zero.

Another formulation of the blade loading makes use of the velocity difference between suction and
pressure side and divided by the average velocity:

\[
BLv = \frac{w_{ss} - w_{ps}}{\bar{w}}
\]

**Other quantities**

Beyond the afore mentioned variables the average circumferential component of the absolute velocity
\( \bar{c}_u \) as well as the average swirl \( B \) can also be displayed. Those quantities are determined by:

\[
\bar{c}_u = \bar{u} - \bar{w} \cdot \cos(\beta)
\]

Also the Ackeret\footnote{**108**} criteria are displayed together with the relative velocities. In accordance to the
below defined Ackeret criteria the maximum relative velocity of the respective span shall not be
bigger than \( 1.8 \cdot \bar{w}_2 \), whereas the minimum relative velocity shall not be smaller than \( 0.3 \cdot \bar{w}_1 \) (\( \bar{w}_1 \) and \( \bar{w}_2 \) are the average relative velocities at LE and TE resp.). Those limits (max. and min. velocities) are not
displayed for splitter blades.
Ackeret = $\frac{w_2}{w_1}$

Ackeret_{max} = 1.8 = $\frac{w_{maxSS}}{w_2}$

Ackeret_{mn} = 0.3 = $\frac{w_{minFS}}{w_1}$

[Compressors and Turbine rotors only]

Mach Number

Mach Number can be displayed both relative as well as absolute.

$Ma_w = \frac{w}{a}$,

$Ma_c = \frac{c}{a}$.

Here $a$ is the sonic speed defined by:

$a = \sqrt{\kappa \cdot R \cdot Z \cdot T}$.

Cross section area and critical area

The specified mass flow can only be realized for a certain size of the cross section at the given total inlet state. A critical cross section is determined by the following set of equations under assumption of perfect gas behavior:

$p_{cr} = \pi_{cr} \cdot P_{in}$,

$T_{cr} = T_1 \cdot \pi_{cr}^{R/cp}$,
Here $\pi_{cr}$ is the pressure ratio at which the flow is at sonic speed in the smallest cross section:

$$\pi_{cr} = \left( \frac{2}{\kappa + 1} \right)^{\frac{\kappa}{\kappa - 1}}.$$

For Air $\pi_{cr} = 0.528$. At the given inlet total state it is not possible to transport the mass flow through a cross section smaller than $A_{cr}$. Both the actual ($A$) and the critical cross section ($A_{cr}$) can be displayed. The actual cross section is the cross section according to the blue surface in the picture above.

If the combination of mass flow, total inlet condition and geometry (cross section) yields a state that is physically not possible a solution cannot be determined and a hint is displayed saying: "No solution due to shocks or transsonic behavior at span: $x$". $x$ will hold all spans for which the hint is true.

### 7.3.2.2.5 Blade-to-blade flow 1D
Swirl cu·r and its derivative

cu-values are calculated based on the assumption that the flow follows the direction given by the blade, i.e. by the blade angles. For centrifugal and mixed-flow impeller this assumptions is applied until the Stanitz-radius (see Stanitz & Prian). At radii bigger than the Stanitz-radius the slip is taken into account. For axial impellers the assumption of blade congruent flow is applied up an equivalent meridional position. If the flow enters the blade passage with an incidence (see blade properties) the relative flow is assumed to be blade congruent latest as \( m/m_{\text{max}} = 30\% \), depending on the incidence angle. The meridional velocity component \( c_m \) is taken from the meridional flow calculation.

In case the meridional flow calculation failed the blade-to-blade flow 1D results cannot be calculated and the diagram will not be available.
7.3.2.2.6 Blade-to-blade flow 2D
Stream function $\psi$

Stream lines must be known a-priori (see Meridional flow calculation). If the meridional flow calculation failed, blade-to-blade flow 2D cannot be calculated and the diagram will not be available. The stream lines rotated around the z-axis build stream surfaces. The relative stream function and relative velocities will be calculate in a blade-to-blade section, that is encapsulated by two stream surfaces and represent a fluid strip. Since hub and shroud are considered as stream lines, there are always two fluid strips less than stream lines. All calculations of the relative stream function and its derivatives are done within a fluid strip that has a stream surface in between. Results of these calculations are given for fluid strips that correspond to inner stream lines or surfaces resp. In the picture below those stream lines have indexes from 2 .. 6.

In contrast to the Stanitz&Prian approach here a two-dimensional relative flow is calculated. This equation in m-t co-ordinates reads as:

$$\frac{\partial^2 \psi}{\partial t^2} - \frac{1}{\rho} \frac{\partial \rho}{\partial t} \frac{\partial \psi}{\partial t} + \frac{\partial^2 \psi}{\partial m^2} - \frac{1}{\rho \Delta n} \frac{\partial (\rho \Delta n) \Delta n}{\partial m} = \rho \Delta n \cdot 2 \omega \cdot r \frac{\partial r}{\partial m}.$$

This equation can be derived from the the assumption of zero absolute rotation of the flow in the fluid strip between two adjacent blades and from the equation of continuity in two dimensions respectively:

Here $w$ is the relative velocity, $\omega$ is the rotational speed and $\rho$ is the fluid density. In the equation above $\Delta n$ is the normal height of the fluid strip. Another assumption is that there is no variation of the
density with respect to the tangential co-ordinate \( t \). The information about the meridional distribution is coming from the Stanitz&Prian\cite{20} approach.

**Boundary conditions**

The boundary conditions are defined as follows. At the suction side a stream function value of zero is set whereas at the pressure side it is set to the mass flow that is conveyed through the fluid strip. For the 5 fluid strips this is 2/5 times the design point\cite{21} mass flow.

\[
\psi_{ss} = 0,
\]
\[
\psi_{ps} = \frac{2}{\text{No. fluid strips}} \cdot \dot{m}.
\]

At inlet and outlet all stream function values are linearly interpolated with respect to the tangential co-ordinate \( t \). Then all stream function values are defined at the boundaries.

**Calculation grid and solution scheme**

The equation is solved using a finite-difference-method (FDM) on a computational grid, which is generated by interpolating mean lines between pressure and suction side. For more information about the finite-difference-method refer to e.g. Anderson et al\cite{22}.

**Results**

The tangential and the meridional relative velocity component resp. can be calculated by:

\[
w_u = \frac{1}{p \Delta n \cdot r} \frac{\partial \psi}{\partial t},
\]
\[
w_m = \frac{1}{p \Delta n \cdot r} \frac{\partial \psi}{\partial \dot{m}}.
\]

The static pressure \( p_i \) can be determined by using the constancy of the rothalpy. For incompressible fluids that reads:

\[

\text{For compressible fluids the same principle is applied for the specific enthalpy and temperature resp. with the assumption of perfect gas behavior\cite{23}. Since the density is already known the static pressure can be calculated using the equation of state } p = f(T, \rho).
\[ T_1 + \frac{1}{2c_p} (\nu_1^2 - u_1^2) \rightarrow T_1 + \frac{1}{2c_p} (\nu_1^2 - u_1^2). \]

The total pressure is derived from the Bernoulli equation for incompressible fluids and by assuming an isentropic state change from \((p,T,c>0) \rightarrow (p_t,T_t,c=0)\) for compressible fluids.

**[ Compressors and Turbine rotors only]**

Mach Number can be displayed both relative as well as absolute.

\[
Ma_w = \frac{w}{a}, \quad Ma_c = \frac{c}{a},
\]

Here \(a\) is the sonic speed defined by:

\[
a = \sqrt{\kappa \cdot R \cdot Z \cdot T}.
\]

The specified mass flow can only be realized for a certain size of the cross section at the given total inlet state. If the combination of mass flow, total inlet condition and geometry (cross section) yields a state that is physically not possible a solution cannot be determined and a hint is displayed saying: "No solution due to shocks, transsonic behavior or numerical reasons at span: \(x\).\) \(x\) will hold the actual span number for which the hint is true.

7.3.2.2.7 Sine rule

**[ Turbine rotors only]**

With the help of the sine rule blade angles at the outlet can be evaluated. In accordance to this rule blade angles at the outlet should have almost the same size as the angle that is built by a hypotenuse being the pitch \(t\), and a cathetus (opposite leg) being the smallest distance between two neighboring mean lines \(eq\) at a flow surface. If this is the case the outflow can be regarded as almost tangential to the trailing edge.

This is shown in a picture for a single mean line.
Blade angles and relative angles

Three different angle distributions can be displayed for each span:

- Blade angles $\beta_B$
- Relative flow angle $\beta_F$
- Ineffective relative flow angle

With the specified incidence and deviation angles (see blade properties) and the attachment and detachment location (the latter is the Stanitz-Radius), relative flow angles can be determined based on the blade angles. An ineffective blade has got a relative flow angle distribution that does not change the pre-swirl.
7.3.3 Blade profiles

To create blade profiles (main and splitter) the blade thickness distribution for the hub and the shroud profile is used. By default the thickness is defined at leading edge, at trailing edge and at the control points of the blade. For the initial CFTurbo-design, typical values in dependence on the impeller diameter $d_2$ are used (see Approximation functions).

2 impeller types have special thickness distribution:

- **Waste water pumps** have very high thickness at leading edge to avoid solid attachments. Starting from 20% of the blade length the thickness is constant up to the trailing edge.

- **Inducer pumps** have very low thickness at leading edge to improve suction performance. The very small leading edge thickness is increasing up to 40%...80% of pitch ($t = \pi d / n_B$) to achieve constant blade thickness. The thickness distribution is asymmetric and sharpen at the suction side only.

The representation of the thickness distribution is made along the relative blade length (0 = leading edge, 1 = trailing edge).
Each thickness curve has a popup menu (right click on curve) to handle its properties.

“Polyline to Bezier” converts a loaded polyline into a Bezier curve, where the number of desired Bezier points can be specified.

“Load profile from profile manager” can be selected to use a pre-defined thickness distribution.

“Convert to Bezier” is available for polyline thickness definition only and converts it to a Bezier curve very similar to “Polyline to Bezier”.

The position of the control points can be changed by moving them with the mouse or by entering specific values by right-clicking.

For the value input, the relative or absolute position along the blade can be used alternatively.

The following general properties of the profile design can be specified on the right side:

**Design Mode**

**Linear**

Linear interpolation between control points

**Freeform**

Bezier curves are used for the thickness distribution

**LE / TE rounded:** Leading and trailing edge can be rounded optionally. If a polyline was loaded this option is determined automatically and cannot be modified.
Linked to Main

Only for splitter blades: splitter profile is linked to main profile

Basic settings

Thickness definition

Thickness definition specifies the way of adding blade thickness values to both sides of the blade mean line to create the pressure and suction sides of the blade. Three types of method are supported:

**Perpendicular to mean surface**
After creating the mean surface from all mean lines, the thickness values are added along surface normal. Naturally, this method depends on all mean lines.

**Perpendicular to mean line (recommended)**
Thickness values added orthogonal to mean line inside rotational surface defined by span. Compared to method above, this definition only depends on the mean line/span itself. Therefore it provides higher stability in trimming with hub/shroud especially for highly curved blade geometry.

**Tangential**
This method is operating point-wise by adding the thickness values in tangential direction and is therefore the most independent method.
None
No coupling between suction side and pressure side

Symmetric
Symmetric thickness distribution: control points on suction and pressure side are coupled

Constant distance
Shifting the thickness distribution to pressure/suction side whereas the distribution itself remains constant

Flexible length position
Shifting control points in horizontal direction

Global point count
Global number of control points

Hub to Shround/Tip (spanwise)

Identical profiles
All profiles have the same thickness distribution

Thickness exponent
Adjusts the morphing of hub/shroud-thickness for inner profiles. Default is linear.

Asymmetry at edges (hub)
Provides adjustments of asymmetry at leading / trailing edge relative to first inner control points respectively.

Possible warnings
<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure and suction side (...) are intersecting or swapped.</td>
<td></td>
</tr>
<tr>
<td>The blade sides are intersecting or they are on the opposite position.</td>
<td>Check the imported profile data if a) pressure and suction side are not intersecting b) pressure side is always above suction side</td>
</tr>
<tr>
<td>Normal this can occur only when loading profiles from file.</td>
<td></td>
</tr>
<tr>
<td><strong>Profile of Main/ Splitter blade exceeds its valid range.</strong></td>
<td></td>
</tr>
<tr>
<td>Profile is defined for a relative blade length smaller 0% or greater 100%.</td>
<td>Check the imported profile data or correct the Beziér control points to lie between 0% and 100%.</td>
</tr>
<tr>
<td><strong>Loaded profiles do not correspond to settings of design mode</strong></td>
<td></td>
</tr>
<tr>
<td>Profile properties defined by context menu in the design dialog do not match Design mode settings.</td>
<td>May occur if thickness distribution is loaded from profile manager: a) Check and adjust state of check box LE rounded and TE rounded b) Apply profiles to both hub and shroud resp., or choose identical profiles</td>
</tr>
<tr>
<td><strong>Blade thickness values don’t match target thickness on LE/TE.</strong></td>
<td></td>
</tr>
<tr>
<td>Current profile thickness on leading- / trailing edge deviate from the specifications of the Blade properties dialog.</td>
<td>Check the imported profile data if the values for leading and trailing edge match those of the Blade properties dialog.</td>
</tr>
<tr>
<td><strong>Pressure/Suction side at Hub/Shroud:</strong></td>
<td></td>
</tr>
<tr>
<td>max. thickness seems too high to get smooth surface.</td>
<td>Either blade thickness at the specified profile side or meanline curvature at the specified span position has to be reduced.</td>
</tr>
<tr>
<td>The combination of of high blade thickness and high meanline curvature results in degenerated blade profiles and prevents creating smooth blade surface.</td>
<td></td>
</tr>
<tr>
<td>Problem</td>
<td>Possible solutions</td>
</tr>
<tr>
<td>---------</td>
<td>-------------------</td>
</tr>
<tr>
<td>Internal blade thickness is lower than specified in Blade properties dialog.</td>
<td>Adjust the inner control points</td>
</tr>
</tbody>
</table>

After changing the blade thickness on leading or trailing edge in the Blade properties dialog, the thickness of the blade at the inner control points is unaffected. It could happen that the thickness on leading and trailing edge is higher than in the middle of the blade.

### 7.3.3.1 Additional views

The following information can be displayed in the blade profile dialog using the "Additional views" button:

- **Informational values**
- **Blade passage area**
- **3D-Preview**
- **x,y frontal view**
- **Blade to blade**
- **Blade profile**
- **Meridional thickness**
- **Blade surface values**

**Informational values**
The Info panel represents information of the designed blade profile:

Throat area
Smallest cross section between 2 neighboring blades

Actual thickness
Actual orthogonal blade thickness values of hub and shroud profiles at leading edge, at trailing edge, after 1/3 and after 2/3 of the blade length
If the cells are colored red, then the thickness on leading/trailing edge is differing from the Target thickness.

Target thickness
Orthogonal blade thickness values for hub and shroud profiles at leading edge and at trailing edge as defined in the Blade properties dialog.
Please note that the blade thickness on leading and trailing edge should be modified in the Blade properties dialog only. In this case the blade angle calculation should be updated due to the blade blockage.

3D-Preview

3D model of the currently designed blades as well as surfaces of hub and shroud and mean surfaces.

Frontal view
The Frontal view represents the designed profiles in a frontal view, including diameters $d_H$ and $d_2$.

Furthermore, the smallest cross section between 2 neighboring profiles is displayed.

**Blade passage area**

Area that is approximately perpendicularly flown through and formed by hub, shroud and two neighboring blades.

**Blade profile**

Undistorted profiles in relative or absolute co-ordinates. In display options ✓ the span to be displayed can be selected.

**Blade to blade**

Two neighboring blades in m-t-co-ordinates. In display options ✓ the span to be displayed can be selected.

**Profile distance**

Distance of two neighboring blades in m-t-co-ordinates. For axial machines with a coaxial meridian this gives a good impression of the de facto distance distribution.

**Meridional thickness**
Thickness of blade in z-r-co-ordinates. In display options the definition of thickness can be switched.

**Blade surface values**

See **blade surface values**.

### 7.3.4 Blade edges

**IMPELLER | Blade edge**

The previously designed blade has a blunt leading and trailing edge (connection line between endpoints of suction and pressure side).

The blade edges are designed by specifying its thickness distribution. The representation of the blade thickness $s$ is made on 15% of the straight blade length $l$ on leading and trailing edge.

If the complete thickness distribution including leading or trailing edge was already designed in the **Blade profile** dialog, then the **Edge position** (transition from blade edge to blade suction/pressure side) has to be defined only.

There are two different options to design the edge shape from hub to shroud/ tip (except for the “Simple” design mode):
**Linear**
Blade edges at hub and shroud/tip can be designed independently, while the intermediate spans are interpolated linearly.

**Uniform**
Only the edges at hub can be designed. All spans use identical parameters.

In panel **Design mode** the blade edge shape can be selected:

(1) **Simple**

The blade edge has a blunt end. A straight line is calculated from the endpoint of suction side perpendicular to the mean line. Trim on inlet/outlet effects trimming the blade on the corresponding inlet or outlet surface.

(2) **Linear**

The blade thickness is changing linear, with an elliptic rounding at the end.
The edge is defined by the overall length $L$, the radius of the end rounding $R$ and the axis ratio $R/R_2$ of the end rounding. Furthermore an asymmetry $A$ can be specified.

**(3) Ellipse**

The blade edge is rounded elliptically. The *axis ratio* $R/R_2$ can be defined. One axis runs on the mean line, the other perpendicular.
(4) Bezier

For this purpose 4th order Bezier curves are used.

Points 0 and 4 representing the transition between the blade sides and the rounded blade edge. You can move these points only along the corresponding blade side. Bezier points 1 and 3 can only be moved on straight lines which correspond to the gradient of the curve in points 0 or 4, respectively in order to guarantee smooth transition from the contour to the blade edge. Bezier point 2 is not restricted to move - it has the most influence to the shape of the blade edge. Its horizontal position is calculated automatically in such way that the leading edge starts at position $l=0$ and the trailing edge ends at position $l=\text{blade length}$. The blade edges are designed at the first or last 10% of the blade length.

Axis-Symmetric results in symmetric geometry, i.e. points 0/4 and 1/3 have the same horizontal position and point 2 is on the middle line.

Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Blades exceed meridional boundaries due to specified blade edge geometry. Check meridional leading and trailing edge position.</td>
<td>The warning indicates that some parts of the blade leading edge are outside the meridional dimensions of the component. Dependent upon the location of these areas one has to modify leading or trailing edge. If the leading edge (or the trailing edge of turbines) exceeds the meridional boundaries you can adjust it in the Meridional contour dialog only.</td>
</tr>
<tr>
<td>Problem</td>
<td>Possible solutions</td>
</tr>
<tr>
<td>------------------------------------------------------------------------</td>
<td>------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>The orthogonal application of thickness on the mean lines can result in some blade position outside the meridional boundaries. Therefore, the model finishing option 'solid trimming' will not be available.</td>
<td>Exceeding trailing edge (or leading edge of turbines) can be corrected by <em>trim on in/outlet</em>.</td>
</tr>
</tbody>
</table>

**It is impossible to trim blade at leading/trailing edge.**

The resulting blade has to be trimmed on the meridional leading and trailing edge. In special situations this trim operation is not possible for geometric reasons.

**Meridional contour** [red], **Mean line** [blue].

The angle between the mean line and the meridional leading/ trailing edge should be high.

**Blade profile** [green]: Reduce blade thickness.

**Error when extrapolating Blade to reach Hub/Shroud surface.**

Check meridional geometry, blade angles and thickness.
<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>The orthogonal blade thickness is added to the blade mean line to create the blade sides. Then one blade side will be trimmed on hub/ shroud, the other one will be extrapolated to hub/ shroud surface. For the below illustrated configurations of meridional contour and blade geometry the extrapolation fails.</td>
<td><strong>Meridional contour</strong>: Account for blade thickness during leading edge positioning or align leading edge towards the direction of the shroud normal (see images below).&lt;br&gt;The trimming/ extrapolation of blade and hub/ shroud will be successful depending on blade angles and blade thickness. A solution can be the modification of the leading edge by repositioning and changing its angle relative to the shroud.&lt;br&gt;<strong>Blade properties</strong>: Increase the number of spans.&lt;br&gt;<strong>Blade profile</strong>: Reduce blade thickness or change thickness definition to &quot;Perpendicular to mean line&quot;.&lt;br&gt;<strong>Mean line</strong>: Check mean line shape and keep lean angle on a low level.</td>
</tr>
</tbody>
</table>

![Mean surface](image1.png)

**Missing intersection**

![Mean surface](image2.png)
<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Distance between blade and hub is higher than the critical value.</strong></td>
<td></td>
</tr>
<tr>
<td>The ratio (distance from blade to hub) / (average diameter) is higher</td>
<td>Try to make lean-angle smaller and/or</td>
</tr>
<tr>
<td>than the critical value.</td>
<td>decrease thickness.</td>
</tr>
<tr>
<td>This could result in trimming issues later on.</td>
<td></td>
</tr>
<tr>
<td>**Impossible blade edge design: overlapping leading and trailing edge</td>
<td></td>
</tr>
<tr>
<td>blade.</td>
<td></td>
</tr>
<tr>
<td>Overlapping of Leading and trailing edge, impossible to design pressure</td>
<td>Reduce the edge positions of one or both</td>
</tr>
<tr>
<td>and suction side.</td>
<td>edges.</td>
</tr>
</tbody>
</table>

![Diagram showing blade and hub distances and edge overlap](image-url)
7.3.4.1 Additional views

The following information can be displayed in the blade edges dialog using the "Additional views" button:

<table>
<thead>
<tr>
<th>Informational values</th>
<th>3D-Preview</th>
<th>Frontal view</th>
</tr>
</thead>
<tbody>
<tr>
<td>Throat area</td>
<td>3D model of the currently designed blades as well as surfaces of hub and shroud and mean surfaces.</td>
<td></td>
</tr>
<tr>
<td>Smallest cross section between 2 neighboring blades</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Informational values**

The Info panel represents information of the designed blade profile:

- Throat area
  - Smallest cross section between 2 neighboring blades
Blade passage area

Area that is approximately perpendicularly flown through and formed by hub, shroud and two neighboring blades.

Blade surface values

See blade surface values.

7.3.4.2 Edge position

If the complete thickness distribution including leading or trailing edge was already designed in the Blade profile dialog, then the Edge position (transition from blade edge to blade suction/pressure side) has to be defined only.
In panel *Edge split* the transition from the blade edge to the suction/pressure side can be defined.

Position in % of the straight blade length.

The leading edge should be within the range of 0% to 15%, the trailing edge between 85% and 100%.
7.4  Airfoil/ Hydrofoil design

The design of the blade's geometry is made in three steps in this design mode:

1. **Blade properties**
2. **Blade profiles**
3. **Blade sweeping**

7.4.1  Blade properties

Definition of blade properties is made in three steps:

1. **Spans**
2. **Cu-specification**
3. **Blade profile selection**
4. **Kinematics**

**Absolute and relative flow**
Impeller

### Absolute velocity
\[ \dot{c} \]

### Relative velocity
\[ \ddot{\omega} \]

### Rotational speed
\[ \dot{u} = \omega \cdot \dot{r} \]
\[ \dot{c} = \dot{u} + \ddot{\omega} \]

Fundamental kinematic equation of Turbomachinery

#### Velocity triangles

<table>
<thead>
<tr>
<th>Radial impeller</th>
<th>Axial impeller</th>
</tr>
</thead>
<tbody>
<tr>
<td>( u_1 ) = ( u_2 ) = ( u )</td>
<td>( u_1 = \beta_1 )</td>
</tr>
<tr>
<td>( c_1 = c_{m1} )</td>
<td>( c_{u1} = 0 )</td>
</tr>
</tbody>
</table>

**Specification of number of blades**
In the right panel some information are displayed which result from calculated or determined values:

(1) Velocity triangles

The velocity triangles of inflow and outflow are displayed.

Continuous lines represent flow velocities on hub (blue) and shroud (green).

Velocities directly before and behind blade area are displayed by dashed lines to show the influence of blockage in the flow domain.

Furthermore the blade angles are displayed by thick lines in order to see the incidence angle on the leading edge and the flow deviation caused by slip velocity on trailing edge.
(2) Values

Numerical values of velocity components and flow angles are displayed in a table. The track bar on top of table can be used to get the values at any span. A short description is at mouse cursor too:

- \( z \) Axial co-ordinate
- \( d \) Diameter
- \( \alpha_F \) Angle of absolute flow to circumferential direction
- \( \beta_F \) Angle of relative flow to circumferential direction
- \( u \) Circumferential velocity
- \( c_m \) Meridional velocity \( (c_m = w_m) \)
- \( c_u \) Circumferential component of absolute velocity
- \( c_r \) Radial component of absolute velocity
- \( c_{ax} \) Axial component of absolute velocity
- \( c \) Absolute velocity
- \( w_u \) Circumferential component of relative velocity: \( w_u + c_u \)
- \( w \) Relative velocity
- \( \tau \) Obstruction by blades (see below)
- \( i \) Incidence angle: \( i = \beta_{B1} - \beta_1 \)
- \( \delta \) Deviation angle: \( \delta = \beta_{B2} - \beta_2 \)
- \( w_2/w_1 \) Deceleration ratio of relative velocity
- \( \Delta \alpha_F \) Absolute deflection angle
- \( \Delta \beta_F \) Relative deflection angle
- \( \Delta \beta_B \) Blade deflection angle
- \( \gamma \) Slip coefficient
- \( \Delta(cu-r) \) Swirl difference
- \( M \) Torque
- \( \Delta p_t \) Pressure difference (total-total)
(3) Curves

Here blade angles as well as relative flow angles are displayed versus span.

Progressions of geometric parameters (angles):

- $\beta_{1/2}$ Angle of relative flow to circumferential direction
- $\beta_{B1/2}$ Blade angles at leading and trailing edge

(4) Criteria

Progressions of aerodynamic and airfoil parameters:

- $Re$ Reynolds-number
- $l/t$ solidity
- $DH$ DeHaller criterion
- $ST$ Strsheletzky criterion
- $DF01$ diffusion number

Possible warnings
<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Automated blade angles are active.</strong>&lt;br&gt;Values may adapt to changing input parameters.</td>
<td><strong>Impeller</strong>&lt;br&gt;Stagger angles and chord length are updated automatically when input parameters are modified. To fix stagger angles and chord length uncheck “Automatic” calculation. Then calculation must started manually if required.</td>
</tr>
<tr>
<td><strong>Automated blade angles are NOT active.</strong>&lt;br&gt;Values are fixed but may not reflect input parameters.</td>
<td>Stagger angles and chord length are not updated automatically if any input parameters are modified. To be sure that all parameter modifications are considered you could switch to an automatic calculation by checking the “Automatic” option.</td>
</tr>
<tr>
<td><strong>Swirl gradient violates Euler equation.</strong>&lt;br&gt;Check blade angles and velocity triangles.</td>
<td>$c_{u2} r_2$ is lower than $c_{u1} r_1$ (turbines: $c_{u2} r_2$ is higher than $c_{u1} r_1$) resulting in energy transmission in the wrong direction (Euler equation of turbomachinery). Recalculate and/or check stagger angles $\gamma$ and chord length $l$, check cu-cm-specification or chosen profiles.</td>
</tr>
<tr>
<td>$\Delta \beta B_{1/2}$ (leading/trailing edge) = ... is larger than warning level of ...</td>
<td>Blade angle difference (highest - lowest value) at all spans exceeds the warning level (see Preferences: Warning level). The resulting blade could be highly twisted. Check the resulting 3D blade shape and avoid high blade angle differences on spans if possible.</td>
</tr>
<tr>
<td>$\Delta \beta B$ (span) = ... is larger than warning level of ...</td>
<td>$\Delta \beta B =</td>
</tr>
<tr>
<td>Blade angles $\beta B_{1/2}$ cannot be determined. Thermodynamic state could not be calculated. Check main dimensions, meridional shape or global setup.</td>
<td>[for compressors and turbines only]</td>
</tr>
</tbody>
</table>
Problem | Possible solutions
---|---
The dimensions or meridional contour might be too tight for the specified mass flow and inlet conditions. | Increase the dimensions (width etc.), meridional contour or change the Global setup (e.g. decrease mass flow).

7.4.1.1 Cu-specification

? Impeller | Blade properties

[ Axial machines only ]

On tabsheet cu, cm the velocity triangles at every span can be defined in accordance to the radial equilibrium.
It can be chosen from 3 different modes concerning the manipulation of $c_{u_2}(r)$:

- **Variable load**
- **Free vortex**
- **Variable load rel. to free vortex**

The $c_{u_2}(r)$-specification is controlled by a second order Bezier curve.

$c_{u_2}(r)$ is defined to get the same swirl at every span:

$$c_{u_2}(r_\text{tip}) = c_{u_2,\text{iso}}(r)$$

The slope is the derivative according to:

$$\text{slope} = \frac{d\left(\frac{c_{u_2}}{c_{u_2,\text{iso}}}\right)}{dr_{\text{tip}}}$$

With a slope of zero a free vortex distribution is set.

**Please note:** There is not always a solution of the differential equation of the radial equilibrium. Therefore some Bezier point constellations are not possible.

At the second tab of the diagram the distribution of the corresponding degree of reaction is displayed: $R = \Delta h_{\text{stat}}/\Delta h_{\text{tot}}$
7.4.1.1.1 Radial equilibrium

Basis of this is the balance of pressure and centrifugal forces under the following assumptions:

- the flow is rotationally symmetric
- friction is neglected
- the streamlines are axis-parallel and have no inclination

The radial balance equation is given here for a section behind an impeller [pump, compressor, fan] and before a rotor [axial turbine] respectively:

The definition of total pressure in section 2 differentiated with respect to \( r \) plus above equation yield:
With the blade work according to Euler the equation becomes:

\[ \eta_{\text{imp}} \cdot 2\pi n \frac{d(c_{u_2})}{dr} = \frac{c_{u_2}}{r} \frac{d(c_{u_2})}{dr} + c_{m_2} \frac{dc_{m_2}}{dr} \]

With the following boundary conditions and a given \( c_{u_2}(r) \)-specification the solution of the differential equation gives a \( c_{m_2}(r) \)-distribution and therefore the complete velocity triangles at every span.

\[ m = \int_{r_{\text{Hub}}}^{r_{\text{Strv}}} \rho \cdot c_{m_2}(r) \cdot 2\pi r \cdot dr \]
\[ P = \int_{r_{\text{Hub}}}^{r_{\text{Strv}}} u(r) \cdot c_{u_2}(r) \cdot \rho \cdot c_{m_2}(r) \cdot 2\pi r \cdot dr \]

From the velocity triangles the degree of reaction can be determined by the following equation:

\[ R = \frac{\Delta h}{\Delta h_t} = 1 - \frac{\Delta (\epsilon^2)}{2\Delta (u_2 \cdot c_{u_2})} \]

7.4.1.2 Blade profiles

On tabsheet **Profile selection** the axial blade profile properties are specified. To this end the profiles have to be selected from the **Profile manager**.

Profile specification on 1 span position is necessary at least to use the same profile on all spans.

Alternatively on any other span position deviating profiles can be selected resulting in interpolation between different profiles.

Profile selection per span can be activated by selecting the check-box at the beginning of each line.
In general, two alternative methods for airfoil design are available:

1. **Blade element momentum method**

Here either NACA 4 digit or point based profiles can be used. Also an angle of attack $\alpha$ has to be specified, see [blade element momentum method](#).

2. **Lieblein method**

Here only profiles of the NACA 65 series can be used. A solidity $l/t$ has to be specified that has to be between 0.4 and 2.0 on all spans. It is used for the calculation of the skeleton length and stagger angle, see [Lieblein method](#).
3. Simple method

Here either NACA 4 digit or point based profiles can be used. A solidity l/t has to be specified, see simple method. [Link]
Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Profile is not reasonable.</td>
<td>- Geometric description or polar data not reasonable.</td>
</tr>
<tr>
<td></td>
<td>- NACA 4 digit or Point-based: Polar data must have more than one pair of ((c_l, \alpha)) and ((c, \alpha)), see Profile manager.</td>
</tr>
<tr>
<td></td>
<td>- Point-based: the mid of upper and lower side must start at ((0, 0)) and end at ((1, 0)) resp.</td>
</tr>
<tr>
<td></td>
<td>- NACA 65 series: Thickness value count must be bigger than one.</td>
</tr>
<tr>
<td></td>
<td>- Circular: must have more than 2 data points ((x, y)).</td>
</tr>
</tbody>
</table>
7.4.1.3 Kinematics

[ Axial machines only ]

The following parameters together with the chosen profile describe the blade at each span:

- $\gamma$ stagger angle
- $l$ chord length
- $\phi$ camber angle (Lieblein method)

Three methods are available for the determination of the scaling (solidity) and staggering of the profiles:

- **Blade element momentum method** [only pumps and fans]
  for low pressure applications (high specific speed $n_q$)
- **Lieblein method** [pumps, fans, compressors]
  for high pressure applications (low specific speed \( n_q \))

- **Simple method** [pumps, fans, compressors, turbines]
  for any applications

On the tabsheet **Profile properties** the **stagger angles** and **solidity** are calculated.

Sweep correction \( \mu \)

Sweep correction is used when the profile properties are calculated. A sweep correction \( < 1 \) means that stagger angle and chord length are overestimated to compensate for losses introduced by blade **sweeping**.
Efficiency values that are relevant for the profile properties calculation are displayed for information as hint.

**Limitations**

The design methods are valid only within certain scopes:

The deceleration should no be smaller than the DeHaller criterion:

\[
\frac{W_2}{W_1}_{\text{hub}} \geq 0.6 \ldots 0.75
\]

In a pipe flow having a swirl a dead water zone is built at small radii. Strecheletzky and Marcinowski stated that the diameter of such a dead water zone should be smaller than the hub diameter of an impeller. From this they derived the following criteria for single stage machines:

\[
\frac{c_{m2}}{c_{u2}}_{\text{hub}} \geq 0.8
\]

and for multi stage machines:

\[
\frac{c_{m2}}{c_{u2}}_{\text{hub}} \geq 1
\]

From boundary layer analysis the diffusion number applied for profiles with a maximum thickness of 10\% was derived:

\[
DF_{0.1} = \left(1 - \frac{W_2}{W_1}\right) + \frac{1}{2} \cdot \frac{t}{W_1} \cdot \Delta W
\]

Special NACA-measurements yield a scope to be fulfilled of \(DF_{0.1} \leq 0.6\).

**Possible warnings**

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Automated blade angles/ stagger angles/ camber angles/ chord lengths are active. Values may adapt to changing input parameters.</td>
<td></td>
</tr>
<tr>
<td>Problem</td>
<td>Possible solutions</td>
</tr>
<tr>
<td>-----------------------------------------------------------------------</td>
<td>-------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>The values are updated automatically when input parameters are modified.</td>
<td>To fix the values uncheck &quot;Automatic&quot; calculation. Then one must manually start the calculation if required.</td>
</tr>
<tr>
<td><strong>Automated blade angles/ stagger angles/ camber angles/ chord lengths are NOT active.</strong>&lt;br&gt;<strong>Values are fixed, but may not reflect input parameters.</strong></td>
<td></td>
</tr>
<tr>
<td>The values are not updated automatically when input parameters are modified.</td>
<td>To be sure that all parameter modifications are considered one shall switch to an automatic calculation by checking the &quot;Automatic&quot; option.</td>
</tr>
<tr>
<td><strong>The blade sweep yields a sweep correction factor of x and is different from the value currently set in blade properties (y).</strong></td>
<td>Set $\mu$ according to value given in <a href="#">blade sweep</a>.</td>
</tr>
<tr>
<td><strong>Radial equilibrium calculation failed.</strong></td>
<td></td>
</tr>
<tr>
<td>There is not for all constellations a solution of the <a href="#">radial equilibrium</a>.</td>
<td>Try to change Bezier-points of <a href="#">cu-curve</a> in case of variable load or switch to free vortex.</td>
</tr>
<tr>
<td><strong>Automatic mode not possible in case of failed radial equilibrium calculation.</strong></td>
<td></td>
</tr>
<tr>
<td>In case of failed <a href="#">radial equilibrium</a> calculation the velocity triangles are not correctly determined and cannot be used for setting the blade angles or stagger angles + chord length.</td>
<td>Uncheck &quot;Automatic&quot; and set blade angles or stagger angles + chord length manually.</td>
</tr>
</tbody>
</table>

7.4.1.3.1 Blade element momentum method

This method makes use of the behavior of a single airfoil in an infinite room, i.e. the airfoil is not influenced by other airfoils. This is true if the solidity $l/t$ is smaller than one.

The design described here is based on the relation between aerodynamic or hydrodynamic profile data and design parameter cast into the Euler equation.

The circumferential force $F_u$ based on the profile properties reads as:
\[ F_u = \sin(\beta_x + \delta) \cdot F \]
\[ \approx \sin(\beta_x + \delta) \cdot c_L \cdot \rho \cdot \frac{w_x^2}{2} \cdot 1 \cdot b \quad \text{with} \quad F \approx F_L, \]

whereas if it is derived from the force balance it reads as:
\[ F_u = \dot{m} \cdot (c_{u2} - c_{u1}) \]
\[ = \rho \cdot c_m \cdot t \cdot b \cdot \frac{Y_{\text{Imp}}}{u}. \]

By equalizing both force descriptions one gets the following equation, which co-relates the profile properties lift coefficient \( c_L \) and solidity \( l/t \) with the design point data \((Y, n, m)\):
\[ \sin(\beta_x + \delta) \cdot c_L \cdot \frac{w_x^2}{2} \cdot 1 = c_m \cdot \frac{Y_{\text{Imp}}}{u}. \]

The meaning of the variables is given in the following table:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( Y_{\text{Imp}} )</td>
<td>specific work of the impeller</td>
</tr>
<tr>
<td>( l/t )</td>
<td>solidity (chord length/pitch)</td>
</tr>
<tr>
<td>( b )</td>
<td>width of the profile</td>
</tr>
<tr>
<td>( c_u )</td>
<td>absolute circumferential velocity component</td>
</tr>
<tr>
<td>( c_m )</td>
<td>absolute meridional velocity component</td>
</tr>
<tr>
<td>( \beta_x )</td>
<td>average rel. flow angle</td>
</tr>
<tr>
<td>( w_x )</td>
<td>average rel. velocity</td>
</tr>
<tr>
<td>( c_L )</td>
<td>lift coefficient</td>
</tr>
<tr>
<td>( \alpha )</td>
<td>angle of attack</td>
</tr>
<tr>
<td>( \delta )</td>
<td>angle between resulting force and lift force</td>
</tr>
</tbody>
</table>
7.4.1.3.2 Lieblein method

This method shall be used when the pressure difference to be generated is comparably high and demands for a high solidity, i.e. a high number of blades. In this case the aerodynamic behavior of each individual blade cannot be determined by investigations on an single blade but is dependent on the whole blade cascade.

Lieblein carried out systematic wind tunnel investigations on the swirl change properties of the profiles of the NACA 65 series. The meaning of the used entities is given in the following table

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>γ</td>
<td>stagger angle</td>
</tr>
<tr>
<td>l/t</td>
<td>solidity (chord length/pitch)</td>
</tr>
<tr>
<td>β</td>
<td>Angle of relative flow</td>
</tr>
<tr>
<td>β_B</td>
<td>Blade angle (of the equivalent circular skeleton line)</td>
</tr>
<tr>
<td>φ</td>
<td>camber angle</td>
</tr>
<tr>
<td>u</td>
<td>circumferential velocity</td>
</tr>
<tr>
<td>w</td>
<td>Relative velocity</td>
</tr>
<tr>
<td>i</td>
<td>Incidence angle: ( i = \beta_{B1} - \beta_1 )</td>
</tr>
<tr>
<td>δ</td>
<td>Deviation angle: ( \delta = \beta_{B1} - \beta_2 )</td>
</tr>
</tbody>
</table>

Three limitations apply for this approach:

- The maximum relative thickness must be \( d/l < 0.1 \).
- The Reynolds-Number must be \( Re_i > 2 \cdot 10^5 \).
- The solidity \( l/t \) must be on all spans: \( 0.4 \leq l/t \leq 2.0 \).
Lieblein derived design diagrams for the following parameter

- Incidence $i$
- Deviation $\delta$

The basic approach is as follows: with the specified solidity the skeleton length is calculated. With the relative flow angle $\beta_1$ (from cu-specification) and the solidity $l/t$ the incidence is determined using Lieblein's design diagrams. The same is done with respect to the deviation. Now the the blade angles at leading and trailing edge are known. Note: The blade angles are applied to the equivalent circular skeleton line with the radius:

$$r_{eq} = \frac{1}{2 \cdot \sin \frac{\beta_{2c} - \beta_{mi}}{2}}.$$

From the blade angles the stagger angle can be determined by:

The dimensionless skeleton line used for the generation of the NACA 65 series profile is described by the following equation:
\[ \frac{y_{ml}}{l} = -\frac{c_n}{4\pi} \left[ \ln \left( 1 - \frac{x}{l} \right) + \ln \left( 1 - \frac{x}{l} - \frac{x}{l} \ln \left( \frac{x}{l} \right) \right) \right], \]

with \( c_n \) the theoretical lift coefficient:

\[ c_n = \frac{2\pi}{\ln(2)} \tan\left( \frac{\varphi}{4} \right). \]

Since this skeleton line is perpendicular at the very beginning (LE) and end (TE) to the chord line, blade angles at these locations derived be tangent angles are not reasonable. This is the reason for applying the equivalent circular skeleton line for the determination of \( \beta_{B1} \) and \( \beta_{B2} \).

### 7.4.1.3.3 Simple method

The simple method is based on a solidity to be specified and on relative flow angles at leading and trailing edge (\( \beta_1 \) and \( \beta_2 \)). These angles will be estimated in the automatic mode or on request (by pressing calculate) based on zero incidence and deviation as well as on the Euler equation (\( \beta_2 \)). The geometric profile parameters stagger angle \( \gamma \) and chord length \( l \) will be determined by:

\[ \gamma = 0.5 (\beta_1 + \beta_2), \]

\[ l = l/t \cdot \pi \cdot d/z. \]

For the meaning of the used entities see table in: Lieblein method.

### 7.4.2 Blade profile

**IMPELLER | Blade profiles**

To create 3D blade profiles the specified or calculated values from the Blade properties are used:

- Profile shape based on profile selection
- Chord length (scaling) and Stagger angle (rotation) of each profile at the respective span position based on profile properties

The resulting 2D profiles are displayed top left in the dialog whereas the thickness distribution at each span location can be found below.
The following information can be displayed using the "Additional views" button:

- **Informational values**: resulting blade angles at leading ($\beta_{B1}$) and trailing edge ($\beta_{B2}$).

- **3D-Preview**: 3D blade shape after the 2D blade profiles were projected into its span surface as well as surfaces of hub and shroud and mean surfaces.

**Profile**

The previously selected blade profile names are displayed for information as hint.

For NACA 4 Digit, NACA 65 series and Point-based profiles the trailing edge thickness can be adapted for manufacturing reasons. The additional thickness is added linearly over the length of the profile.

Two modi are available. The thickness value is applied for those spans at which NACA profiles are specified (not interpolated), see blade properties [14h].

<table>
<thead>
<tr>
<th>Relative</th>
<th>Absolute</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thickness is chord length times relative thickness</td>
<td>Thickness is equal the absolute thickness</td>
</tr>
</tbody>
</table>

**Radial 2D blade shape**
Radial 2D blades can be designed by using a constant stagger angle of a selected master span profile.

**Please note:** By applying the radial 2D blade shape the aerodynamic properties of the resulting blade will be different from those stated in the Blade properties.

**Edge split**

The edge split position defines the transition from blade suction/pressure side to the leading edge. It's used for the 3D model generation as well as for the data export.

### 7.4.3 Blade sweep

In this design step the blade sweep can be optionally specified. Blade sweep is normally only useful for acoustic reasons and comes at the cost of slightly reduced efficiency.

In default configuration this design step does not generate any sweep by aligning the centroid points of all profiles exactly in radial direction. You can return to an unswept configuration at any time by using the Reset sweep curve option.

The left area of the dialog is comprised of four diagrams that display the current blade sweep definition, represented in several projections. Depending on the Sweep mode (see below) selected, only two of these diagrams are active at a time, whereas the other two diagrams are merely informative.

The design curves (orange) in active diagrams exhibit control points which are movable along design guide lines (gray) which subdivide the radial space between Hub (blue) and Shroud (green).

The user designed sweep projections are combined into the 3D sweep curve which is then applied to the blade geometry by stacking the blade profiles along it. The informative sweep projections are updated accordingly.

Independently of Sweep mode the blade positioning in the meridional contour can be controlled in the axial projection diagram (top left). Blade positioning can be controlled via a special control point at the base of the sweep curve, which can be moved along the Hub contour and that moves the blade geometry along with it. Design configurations where the Blade exceeds the meridional boundary have to be corrected by adjusting the blade position in order to finish this design step successfully.
The following information can be displayed using the "Additional views" button:

- **Informational values**: Any sweeping will yield a certain deterioration of the aerodynamic performance compared with the unswept state. The sweep correction $\mu$ is a quantitative expression of this and is based on an empirical co-relation. To compensate for the sweeping losses the sweep correction will be applied when profile properties are newly calculated.

- **3D-Preview**: The final result of the sweep design process, the swept 3D blade shape as well as surfaces of hub and shroud and mean surfaces.

- **Acoustic benefit**: ...compared to the unswept blade design

$$L_{W|_{k=0}} - L_{W|_{k=0}} = 10 \cdot \log(\cos(\theta)^{1/3}) dB$$

**Sweep mode**

The Sweep mode controls which of the 2D Sweep projections define the blade sweep and are modifiable by the user.

For defining a blade sweep two alternative options are available:

- **Axial z + Tangential t**
  Sweep projected in meridional and axis-normal view.
  This view also shows the blade outline.
**Profile stack mode**

The profile stack mode controls how 2D-Profiles are stacked relative to profile geometry onto the 3D-sweep curve. This Design choice will subsequently also be reflected in the display of profiles in the previous Blade profile dialog.

The blade sweep for each sweep mode can be defined on one of the following blade profile positions:

- ▼ leading edge
- □ centroids (default)
- ▼ trailing edge

**Possible warnings**
<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>The air/hydrofoil blade is not located within the predefined blade area. Either extend the meridional blade area or reposition the blade.</td>
<td>Increase the meridional extension or/and the predefined blade area (meridional contour) or adjust chord length and stagger angle (blade properties) or change the blade position.</td>
</tr>
<tr>
<td>The axially projected chord length is higher than the meridional length or the predefined blade area.</td>
<td></td>
</tr>
</tbody>
</table>

7.5 CFD setup

**Extension, RSI connection**
Description can be found in the topic [Virtual geometry](#).

**Blade projection**

Blade projection can be used for meshing purposes.

In case of an unshrouded impeller the outer blade profile is projected onto the casing.

If an [Extension](#) exists, the blade trailing edge is projected onto the Rotor-Stator-Interface (RSI).
7.5.1 Virtual geometry

The designed geometry can be extended by **virtual** elements for a simplified flow simulation (CFD).

- **Extension**
- **RSI connection**

**Virtual vs. real geometry**

Real and virtual geometry can coexist in the project, but can be visualized in the 3D-View and exported separately.

The **real geometry** represents the real design including the secondary flow path geometry for the impeller. If the model does not contain any secondary flow path, the real impeller geometry corresponds to the isolated impeller.
Virtual geometry represents the combination of designed geometry (without secondary flow path) and virtual elements (CFD extension or RSI connection) activated on impeller side.

The virtual geometry has 2 objectives:

1) Simplification of the flow domain by neglecting the secondary flow path to accelerate CFD simulation.
   Of course, deviations from the real flow behavior of the machine occur, but a qualitative comparison of different geometries is usually still possible.

2) Closing the necessary gap between the rotating impeller and adjacent static components.
Please note: In the 3D model, the extension is part of the impeller solid “Flow domain (virtual)”. The rotor-stator-interface (RSI) is the border to the neighboring component. On the other side, the RSI connection is part of the neighboring component solid “Flow domain (virtual)”.

3D model

In the 3D model, both geometries can be displayed independently using the proper tree nodes or selecting the corresponding model state:
The designed geometry can be extended in meridional direction at the outlet. This extension defines the Rotor-Stator-Interface (RSI). It is only used for CFD purposes and its position is not taken into account by neighboring components. Typically, the RSI is located in the middle of the rotating and the non-rotating component.

Using the extension is recommended, because otherwise the trailing edges of the blades would just lie on the rotor-stator interface, which can cause both meshing problems and numerical simulation errors. Meshing problems could occur especially for small values of the blade angle $\beta_{B2}$. 
The designed extension is displayed in the diagram in olive color.

The drop down menu **Direction** sets the direction of the extension. If it is set to **tangential**, hub and shroud will be tangentially extended. If it's set to **connected**, the extension will match the next component inlet.

Below you can specify the **Length** of the extension and whether the length should be measured radial or absolute (i.e. in the direction specified above). Furthermore, you can set **Extension outlet at r = constant** or **Extension outlet at z = constant**, which means that the outlet of the extension is forced to be horizontal (parallel to the z-axis) or vertical (parallel to the r-axis).

**Limitations**

The extension is not available for **stators**. For CFD simulation of vaned stators, some space after the trailing edge might be needed. This can be achieved by unchecking the **Trailing edge fixed on outlet** checkbox in the meridional contour dialog.

**Possible warnings**

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Length of extension is smaller than the model tolerance.</strong></td>
<td><strong>This is problematic for &quot;Flow domain&quot; creation during model finishing.</strong></td>
</tr>
<tr>
<td>The length of the extension is smaller or equal to the <strong>distance tolerance</strong>. This might cause geometrical defects when sewing faces during <strong>Model finishing</strong>.</td>
<td>If geometrical problems occur, change the distance tolerance or the length of the extension.</td>
</tr>
</tbody>
</table>
### Problem

<table>
<thead>
<tr>
<th>Extension outlet has nearly constant radius. Selecting &quot;Extension outlet at r = constant&quot; is recommended.</th>
</tr>
</thead>
<tbody>
<tr>
<td>The endpoints of the hub and shroud extension have a slightly different radius. This can result in almost flat cone surfaces for the adjacent RSI connection, which may be problematic to import into other CAD/CFD systems.</td>
</tr>
<tr>
<td>Set the endpoints of the hub and shroud extension to the same radius by checking the &quot;Extension outlet at r = constant&quot; checkbox.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Extension outlet is nearly vertical. Selecting &quot;Extension outlet at z = constant&quot; is recommended.</th>
</tr>
</thead>
<tbody>
<tr>
<td>The endpoints of the hub and shroud extension have a slightly different z-coordinate. This can result in almost flat cone surfaces for the adjacent RSI connection, which may be problematic to import into other CAD/CFD systems.</td>
</tr>
<tr>
<td>Set the endpoints of the hub and shroud extension to the same z-coordinate by checking the &quot;Extension outlet at z = constant&quot; checkbox.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Extension is overlapping neighboring component.</th>
</tr>
</thead>
<tbody>
<tr>
<td>The interface defined by hub and shroud extension overlaps neighboring component.</td>
</tr>
<tr>
<td>Reduce extension length or increase gap between impeller and static neighboring component.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Extension is not well defined.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Under certain circumstances it is possible that geometric interface and CFD-interface (extension) are misaligned.</td>
</tr>
</tbody>
</table>
| Check configuration of extension regarding:
  - intersection
  - matching
  - inversion (negative extension)

See diagram for its visual representation. |

---

### 7.5.1.2 RSI connection

The RSI connection is intended to close the gap between the impeller and the neighboring static component. It provides a simplified, closed volume model for flow simulation neglecting impeller secondary flow path or other casing parts. The RSI connection is added to static components located downstream or upstream to an impeller.
The designed RSI connection is displayed in the diagram in orange color.

Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>RSI connection at Inlet/Outlet cannot be generated.</td>
<td>Disable RSI connection or perform corrections at neighbouring component interface if necessary.</td>
</tr>
</tbody>
</table>

RSI connection can be generated only if a gap between components exists.

7.6 Model settings

On dialog **Model settings** you can specify how many data points are to be used for the 3D model and for the point based export formats.

The number of points can be set for both cases separately for all geometry parts.

- **Meridian**: hub/shroud
- **Blade**: mean line, pressure/suction side, leading/trailing edge
3D Model

Distance tolerance

The distance tolerance defines the maximum allowed distance between sewed surfaces, e.g. the faces of a solid.

If it is too small, the solids cannot be created.

If it is too big, small faces are ignored when creating a solid.

Point Export

Presetting

Select from 3 global presets.

When a new impeller is created the model settings of the last opened impeller are carried over.
7.7 Model finishing

The dialog offers different possibilities to design the connection between blade, hub and shroud.

**No model finishing**
### Extend blade only

Extends blades through hub and shroud for later trimming in a CAD-system.

Additionally, extends blades through leading/trailing edge, if it is fixed as well as simple and trimmed on inlet/outlet.

### Solid trimming

Trims blades on hub, shroud and possibly leading and trailing edge.

The solid trimming affects only the solids (and solid faces) of Flow domain, Material domain and Blade.

Trimming is a time-consuming operation (up to 30 seconds).

Because only solids are trimmed, point-based exports cannot take advantage of this operation.

**Details:**

*Solid trimming* is based on a segment. If no segment is defined, it is created temporarily, not visible to the user.

**Internal workflow:**

- The blades are extended (see **Extend blade only**)
- A single blade is trimmed with Flow domain
- From Flow domain, a segment is cut. In this way the trimmed Segment.Real geometry is created.
- CFD setup option: If there is an Extension or RSI connection, Segment.Real geometry is fused with Segment.Extension and Segment.RSI connection. In this way, Segment.Flow domain is created.
- Segment.Flow domain is copied multiple times. The copies are rotated and sewed in order to create a new Flow domain.
- If hub and shroud materials have been defined, a Material domain is created by fusing the blades and the solids of hub and shroud.

---

**Meridian.Flow domain**

**Meridian.Material domain**

© CFturbo GmbH
- CFD setup option: If Blade projection was chosen, the corresponding projection surfaces are exactly trimmed.

<table>
<thead>
<tr>
<th>Option: Blade root fillet</th>
</tr>
</thead>
</table>

The edges between blade and hub/shroud are rounded.

The fillet affects only the **solids** (and solid faces) of Flow domain and Material domain.

The **fillet radius** should not be larger than the recommended value.

**Limitations**

Fillet creation is not possible if the fillets of two neighboring blades would meet or if the fillet would protrude beyond the impeller inlet or outlet.

Fillet creation is not possible if the blade has very sharp edges, i.e. if the fillet radius is much larger than the edge radius.

**Update modes**

- **<OK>** The configuration is saved in the project, but the 3D-model is not updating.
- **<OK + Run>** The configuration is saved in the project and the 3D-model is updating using the model finishing options.

The model finishing can be executed globally for all vaned components by **PROJECT/ Model finishing**.

**Possible warnings**
<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solid trimming is not available. Meridional contour is not suitable.</td>
<td></td>
</tr>
<tr>
<td>Generation of segment may fail due to meridional contour.</td>
<td></td>
</tr>
<tr>
<td>The angle between rotation axis and hub contour is too small (lower than 0.1 degree).</td>
<td>Set a value for hub diameter higher than zero or round the hub contour to avoid a geometry with needle form.</td>
</tr>
<tr>
<td>Extend/solid trimming may fail due to tangential difference between hub and shroud at leading/trailing edge, or low number of spans.</td>
<td></td>
</tr>
<tr>
<td>Very low number of spans</td>
<td>Increase the number of spans up to at least 4</td>
</tr>
<tr>
<td>Fillets not supported.</td>
<td></td>
</tr>
<tr>
<td>Fillets are not supported if solid trimming is not possible.</td>
<td></td>
</tr>
<tr>
<td>Problem</td>
<td>Possible solutions</td>
</tr>
<tr>
<td>------------------------------------------------------------------------</td>
<td>-------------------------------------------------------------</td>
</tr>
<tr>
<td>Fillets creation on shroud deactivated.</td>
<td>-</td>
</tr>
<tr>
<td>Fillets on shroud are not supported for unshrouded designs.</td>
<td></td>
</tr>
<tr>
<td>Curvature of Leading edge/Trailing edge/Blade edges at Hub/Shroud could be too large for fillet creation.</td>
<td></td>
</tr>
<tr>
<td>Sharp Blade edges could prohibit fillet creation between blade and hub/shroud and could cause long computation time respectively.</td>
<td>Less sharp design of blade edges or reduction/deactivation of fillet radius.</td>
</tr>
<tr>
<td>3D-Error: Finishing failed!</td>
<td></td>
</tr>
<tr>
<td>Inlet (nearly) tangential to hub or shroud</td>
<td>Change Meridional contour. Avoid tangentiality</td>
</tr>
<tr>
<td>3D-Error: Finishing failed! (Could not extend blade)</td>
<td></td>
</tr>
<tr>
<td>Extending the blade to a scaled Hub/ Shroud surface failed.</td>
<td>See solution of Error while extrapolating Blade to reach Hub/ Shroud surface</td>
</tr>
<tr>
<td>3D-Error: Finishing failed! (Could not trim solids of Blades and Flow domain)</td>
<td>Try using a different number of data points.</td>
</tr>
<tr>
<td>3D-Error: Finishing failed! (Could not create fillet)</td>
<td></td>
</tr>
<tr>
<td>Fillet creation at blade root failed (see limitations)</td>
<td>Decrease fillet radius or Deactivate fillet</td>
</tr>
<tr>
<td>3D-Error: Finishing failed! (Creating trimmed segment)</td>
<td></td>
</tr>
<tr>
<td>Cutting a segment from Flow domain failed. This can occur due to heavy bendings of hub or shroud.</td>
<td>Reduce bending at high curvature regions of hub- or shroud-contours.</td>
</tr>
<tr>
<td>3D-Error: Finishing failed! (Fusing solids)</td>
<td></td>
</tr>
<tr>
<td>Problem</td>
<td>Possible solutions</td>
</tr>
<tr>
<td>------------------------------------------------------------------------</td>
<td>-------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Fusing of real geometry with CFD setup components (Extension or RSI connection) failed.</td>
<td>Increase the number of spans or Remove Extension / RSI connection from CFD setup</td>
</tr>
<tr>
<td><strong>3D-Error:</strong> Finishing failed! (Creating new Flow domain from segment)</td>
<td></td>
</tr>
<tr>
<td>Creating a new Flow domain from CFD setup. Segment failed.</td>
<td>Decrease blade fillet radius.</td>
</tr>
<tr>
<td><strong>3D-Error:</strong> Finishing failed! (Creating Material domain from Flow domain)</td>
<td></td>
</tr>
<tr>
<td>Creating a Material domain failed.</td>
<td>Deactivate hub and shroud materials</td>
</tr>
<tr>
<td><strong>3D-Error:</strong> Blade projection to RSI failed!</td>
<td></td>
</tr>
<tr>
<td>Projection of blade to RSI (Extension) failed.</td>
<td>Change CFD setup: Modify Extension or remove Blade projection</td>
</tr>
<tr>
<td><strong>3D-Error:</strong> Blade tip projection to casing failed!</td>
<td></td>
</tr>
<tr>
<td>Projection of blade to casing (shroud) failed.</td>
<td>Change CFD setup: Remove Blade projection or RSI connection</td>
</tr>
</tbody>
</table>
Part VIII
8 Stator

This chapter describes in detail the design process for stator type components featured in CFturbo.

The content reflects the design steps in the sequence they are encountered during the design process.

Design steps

- Main dimensions
- Meridional contour
- Blade properties
- Blade mean lines
- Blade profiles
- Blade edges
- Model finishing
- Model settings
- CFD setup

Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
</table>
| Neighboring blades are intersecting each other. | Numerous details of the design influence the blade shape. Some examples for possible solutions:  
- Modify main dimensions |

(see message)
8.1 Main dimensions

The Main Dimensions menu item is used to define main dimensions of the stator.

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>· Reduce number of blades</td>
</tr>
<tr>
<td></td>
<td>· Reduce blade wrap angle</td>
</tr>
<tr>
<td></td>
<td>· Reduce blade thickness</td>
</tr>
</tbody>
</table>

The diagram shows the main dimensions of the stator, including radial and axial coordinates.
**General**

Initial definition of the stator type. Currently the following types are available:

- Free form
- Radial diffuser
- Bowl diffuser
- Axial diffuser
- 90° bend left
- 90° bend right

Using the button "Set default" the default properties for each stator type can be set.

The solid density of the stator is an informational value that is not relevant for the hydraulic or aerodynamic design but is used for the calculation of moments of inertia. Density values can be directly entered or selected from a list by pressing the settings button right beside the input area.

**With blades**

Here you can define if the stator should be vaned or vaneless.

For vaned stators you have to define the number of blades and the existence of splitter blades.

Via Unshrouded you can decide to design a shrouded or unshrouded stator. For unshrouded stator you have to define the tip clearance.

**Extent/ Inlet/ Outlet**

Extent, inlet and outlet are coupled geometry definitions. Two of the three categories must be explicitly selected and specified, with the remaining one resulting automatically.

- **Extent**
- **Inlet**
- **Outlet**

**Information**
Right in the dialog some additional information are displayed.

- The **Meridian preview** is based on the until now designed main dimensions and visualizes the general proportions.

- **Information values** lists important coefficients, which result from determined main dimensions. The specific values depend on the selected tab sheet on the left side: Extent, Inlet or Outlet.
  If the font color is blue then a hint for the recommended range of this value is available when the mouse cursor is on the table row.
  If the font color is red then the current value is outside the recommended range.

### Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solution</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Hub/ Shroud/ Midline length is zero (invalid geometry).</strong>&lt;br&gt;The extent of the stator is 0 at hub, shroud or midline.</td>
<td>Specify a reasonable length value or remove the stator completely.</td>
</tr>
<tr>
<td><strong>Thermodynamic state could not be calculated for given main dimensions.</strong>&lt;br&gt;[ for compressors and turbines only ]&lt;br&gt;The dimensions might be too tight for the specified mass flow and inlet conditions.</td>
<td>Increase the dimensions (width etc.) or change the Global setup (e.g. decrease mass flow).&lt;br&gt;If the stator is vaned, its blade angles can be changed too.</td>
</tr>
</tbody>
</table>
8.1.1 Extent

Stator extent has to be considered in relation to its inlet and outlet. These 3 areas are coupled, i.e. one is inherently defined by the two others.

Extent from inlet to outlet can be defined by 2 alternative possibilities in principle:

1. Center line

   - preselection of extension direction: Radial, Axial, Tangential (to outlet of previous component), Free form
   - Definition of axial extension $\Delta z$ and radial extension $\Delta r$
   - Definition of axial extension $\Delta z$ and radial extension $\Delta r$
     or length $L$ and angle of center line to horizontal direction $\epsilon$
   - Definition of end cross section (Inlet or Outlet) by width $b$ and angle to horizontal direction $\gamma$

2. Hub, Shroud

separately for hub and shroud:

   - preselection of extension direction: Radial, Axial, Const. area (with respect to opposite side), Tangential (to outlet of previous component), Free form
Definition of axial extension $\Delta z$ and radial extension $\Delta r$ or length $L$ and angle of hub/shroud to horizontal direction $\varepsilon$

The angles $\varepsilon$ and $\gamma$ are defined by 0° horizontal right and rising in counter clockwise direction (mathematical positive). A menu with some default angles is supporting angle input:

- Right: 0°
- Left: 180°
- Up: 90°
- Down: 270°
- Perpendicular

Perpendicular: $\varepsilon$ perpendicular to inlet or outlet cross section
Parallel: $\gamma$ parallel to inlet or outlet cross section
Depending on the type of geometric coupling, the extent is defining the inlet or the outlet of the component. If the stator has the primary side at outlet, the extent will modify the outlet. Otherwise if the stator has the primary side at inlet, the inlet will be defined by the extent.

If the neighboring components are primary both at inlet and outlet then the extent of the stator cannot be specified because it's clearly defined by its neighbor.

### Information

<table>
<thead>
<tr>
<th>Design point</th>
<th>Design point information, see Global setup</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ratio outlet to inlet</td>
<td></td>
</tr>
<tr>
<td>Diameter ratio</td>
<td>$d_{\text{Out}}/d_{\text{In}}$</td>
</tr>
<tr>
<td>Width ratio</td>
<td>$b_{\text{Out}}/b_{\text{In}}$</td>
</tr>
<tr>
<td>Area ratio</td>
<td>$A_{\text{Out}}/A_{\text{In}}$</td>
</tr>
<tr>
<td>Inlet area</td>
<td>$A_{\text{In}}$</td>
</tr>
<tr>
<td>Outlet area</td>
<td>$A_{\text{Out}}$</td>
</tr>
</tbody>
</table>

#### 8.1.2 Inlet

Here the inlet of the stator can be defined.

If the outlet can be modified, it will be updated by addition of extent to inlet. Otherwise the extent will be adapted.
8.1.3 Outlet

Here the outlet of the stator can be defined.

If the inlet can be modified, then it will be updated by subtraction of extent from outlet. Otherwise the extent will be adapted.

[ Compressors and Turbine only ]

Radio buttons are available that allow to define whether the flow in the stator outlet is supposed to be subsonic or supersonic. This will influence the values displayed in the information panel as well as the isentropic Mach number distribution.

8.2 Meridional contour

In principle, the same features are available as for the meridional design of impellers.
The endpoints of hub and shroud curves are fixed by main dimensions and cannot be modified here.

For “Radial diffuser” type of stators (see main dimensions) the following geometrical dimensions are defined:

![Diagram of blade properties]

### 8.3 Blade properties

In principle, the same features are available as for the blade properties of impellers.

To support the selection of a suitable blade count a separate dialog can be used, which can be started by pressing the button right beside the edit field.

The outlet angles αTE are input values for most of the blade types according to the desired change of flow direction. Slip models are not available for stators. Some angle oversizing should be considered if necessary.
For blade shape "Straight 2D (axial)", a diffuser opening angle \( \theta \) is calculated and displayed for information in accordance to Bohl\(^\text{[1]}\) with:

\[
\tan(\theta) = \frac{a_4 - a_5}{l},
\]

where \( a_4 \) and \( a_5 \) are LE and TE blade passage distance (expressed by equivalent diameters) and \( l \) is the mean line length.

**Warnings**

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solution</th>
</tr>
</thead>
<tbody>
<tr>
<td>A reasonable thermodynamic state could not be calculated.</td>
<td>[for compressors and turbines only]</td>
</tr>
<tr>
<td>The dimensions might be too tight for the specified mass flow and inlet conditions. The chosen blade angles might also yield this state.</td>
<td>Increase the dimensions (width etc.), change blade angle or change the Global setup(^\text{[2]}) (e.g. decrease mass flow).</td>
</tr>
<tr>
<td>Diffuser opening angle bigger than 10(^\circ). Flow separation possible.</td>
<td>[for Straight 2D (axial) only]</td>
</tr>
<tr>
<td>The diffuser opening angle is to big and might give room for flow separation.</td>
<td>Change ( \alpha_{\text{BLE}}, s_1, s_2, \text{LE or TE radius (in meridian)} )</td>
</tr>
</tbody>
</table>

**8.3.1 Number of blades**

Number of blades, stator outlet diameter and minimum blade distance are significant for the actual diffuser part of the stator and therefore have high influence on the flow losses. These 3 parameters have to be adjusted carefully.
The number of blades of impeller and stator has to be coordinated carefully in order to minimize pressure pulsation and therefore mechanical load and noise emission.

The number of impeller blades is defined and fixed by one of the impellers in the project, otherwise it's an input value.

The number of stator blades can be modified and should be one of the recommended ones.

According to the number of blades \( z \) different pressure fields are generated in the impeller and the stator, which are moving relative to each other and are characterized by the periodicity \( p \):

\[
\begin{align*}
\text{impeller periodicity} & \quad p_I = n_I \cdot z_I \\
\text{stator periodicity} & \quad p_{II} = n_{II} \cdot z_{II}
\end{align*}
\]

(\( n \) = integer multiplier)

The interference of both pressure fields cannot be calculated exactly. But most important for the resulting pressure field is the difference of both periodicities:

\[
m = |p_I - p_{II}| = |n_I \cdot z_I - n_{II} \cdot z_{II}|
\]

The following recommendations should be kept:

- \( m = 0 \) (impeller and stator blade count have shared integer multipliers) should be avoided in each case, because high pressure pulsation can be generated here.
- \( m = 1 \) should not be allowed in first and second order (\( n_I = 1; \ n_I = 2 \)) due to unacceptable shaft vibration, if possible also in third order (\( n_I = 3 \)).
- \( m = 2 \) as well represents a periodic impeller load, but is acceptable in most cases.
- Vibration modes with \( m > 2 \) normally don’t generate resonance and are allowed therefore.

For each modification of the stator blade count \( z_{II} \) the \( m \)-values for each combination (\( n_I = 1..3 \)) and (\( n_{II} = 1..3 \)) are calculated and displayed in the table. Values \( m=0 \) are marked in red color, \( m=1 \) in orange and \( m=2 \) in yellow.

The recommended stator blade count according to the current number of impeller blades are represented below the input field.
8.4 Blade mean lines

In principle, the same features are available as for the mean lines of impellers.

For special radial diffuser blade shapes "Log. Spiral + Straight 2D" and "Circular + Free-form 2D" the mean line design is made in the frontal view. The mean lines are the inner vane sides (concave sides).

Initially the blade thickness is ignored for the mean line design (red/magenta in the sketch). The opposite side if the flow channel is generated by rotation and adding the blade thickness. The blade thickness is assumed as linear between sLE and sTE (see blade properties), if the thickness distribution was not defined yet. Otherwise the thickness distribution defined in the blade profile design is used. In the later blade profile design the thickness is added to one side of the mean line only.
Difuser area has to be designed carefully in order to minimize losses. The quality of the diffuser design can be verified according to the following criteria (see panel **Radial diffuser** in **Informational values** area). Values outside the recommended range are displayed in red color.

<table>
<thead>
<tr>
<th>Name</th>
<th>Description</th>
<th>Definition/ recommended range</th>
</tr>
</thead>
<tbody>
<tr>
<td>Throat distance a3 (LE)</td>
<td>Throat width at inlet (leading edge)</td>
<td></td>
</tr>
<tr>
<td>a3 Optimum *</td>
<td>Optimal value: average of calculation by const. swirl and deceleration ratio</td>
<td>see blade properties</td>
</tr>
<tr>
<td>a3 Actual</td>
<td>Actual value: shortest distance from vane leading edge to neighboring vane</td>
<td></td>
</tr>
<tr>
<td>Outlet distance a4</td>
<td>Shortest distance from vane trailing edge to neighboring vane</td>
<td></td>
</tr>
<tr>
<td>Diffuser opening angle θ</td>
<td>Allowable diffusion angle</td>
<td></td>
</tr>
<tr>
<td>Stator</td>
<td>577</td>
<td></td>
</tr>
<tr>
<td>--------</td>
<td>-----</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>θ Maximum allowable</th>
<th>Max. allowable value to avoid flow separation depending on equivalent inlet radius and length</th>
</tr>
</thead>
</table>
|                     | $\theta_{\text{max}} = 16.5^\circ \sqrt{\frac{R_{1,\text{eq}}}{L}}$  
|                     | $R_{3,\text{eq}} = \sqrt{\frac{a_3 b_3}{\pi}}$                                        |

<table>
<thead>
<tr>
<th>θ Actual</th>
<th>Actual value calculated by equivalent inlet radius, length, inlet and outlet area</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$\theta = \frac{R_{3,\text{eq}}}{L} \left( \sqrt{\frac{A_4}{A_3^2}} - 1 \right)$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Area ratio AR=A4/A3</th>
<th>Area or deceleration ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$A_R = \frac{A_4}{A_3}$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>AR Optimum *</th>
<th>Optimal value</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$A_{R,\text{opt}} = 1.05 + 0.184 \frac{L_{3-4}}{R_{3,\text{eq}}}$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>AR Actual</th>
<th>Actual value</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$A_R &lt; 3$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Pressure recovery coeff. cp</th>
<th>Pressure recovery of the diffuser identified by a dimensionless coefficient</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$c_p = \frac{P_4 - P_3}{\frac{\rho}{2} c_3^2}$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>cp Ideal (loss-free) *</th>
<th>Pressure recovery in an ideal (loss-free) diffuser</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$c_{p,\text{id}} = 1 - \frac{1}{A_R^2}$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>cp Optimum *</th>
<th>Pressure recovery for optimal area ratio $A_R$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$c_{p,\text{opt}} = 0.36 \left( \frac{L_{3-4}}{R_{3,\text{eq}}} \right)^{0.26}$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>cp Actual *</th>
<th>Pressure recovery in real diffuser (with energy losses)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Based on test results; plotted in diagrams; target: $c_{p,\text{act}} = c_{p,\text{opt}}$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Diffuser efficiency $\eta_D$ *</th>
<th>Diffuser efficiency</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$\eta_D = \frac{c_p}{c_{p,\text{id}}} = \frac{c_p}{1 - \frac{1}{A_R^2}}$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Inlet velocity ratio c3q/c2</th>
<th>Inlet deceleration ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$c_{3q}/c_2 = 0.7...0.85$ for low specific speed</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Non-dimensional length L34/a3</th>
<th>Ratio of length to throat width</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$L_{3-4}/a_3 = 2.5...6$</td>
</tr>
</tbody>
</table>
### 8.5 Blade profiles

**? STATOR | Blade profile**

In principle, the same features are available as for the blade profiles of impellers.

For the special radial diffuser blade shapes "Log. Spiral + Straight 2D" and "Circular + Free-form 2D" the blade thickness is added to one side of the mean line only (see Mean line).

For radial diffusers the same informational values as in the mean line design are displayed in the Info area. The reason is the influence of the blade thickness to these numbers.

### 8.6 Blade edges

**? STATOR | Blade edges**

In principle, the same features are available as for the blade edges of impellers.

### 8.7 CFD setup

**? STATOR | CFD setup**

Blade projection can be enabled here.
If a RSI connection was enabled for the impeller, these geometrical parts are added to the virtual geometry of the stator (see virtual geometry for more information).

### 8.8 Model settings

**? STATOR | Model settings**

In principle, the same features are available as for the model settings of impellers.

### 8.9 Model finishing

**? STATOR | Model finishing**

In principle, the same features are available as for the model finishing of impellers.
Part IX
This chapter describes in detail the design process for volute type components featured in CFturbo.

The content reflects the design steps in the sequence they are encountered during the design process.

### Design steps

- **Setup + Inlet**
- **Cross section**
- **Spiral development areas**
- **Diffuser**
- **Cut-water**
- **Model settings**
- **CFD setup**

### 9.1 Setup + Inlet

The first design step of the volute is to define the inlet side. It consists of 2 steps:

1. **Setup**
2. **Inlet details**
9.1.1 Setup

On page **Setup** you can define some general properties used for the spiral design.

Depending on the project type different input parameters are required (see below).

**for pumps, fans, compressors**
for turbines
Volute type

- Single volute (default)
  This simple type is commonly used and has a single cut-water.

- Double volute
  A second cut-water (splitter) is designed in order to reduce the radial forces.

Design flow rate

- Volumetric efficiency $\eta_v$ (default: 1.0)
  to consider any internal volumetric losses (recirculation)

- Flow factor $F_\Omega$ (default: 1.0)
  for over dimensioning, particularly for better efficiency at overload operation
Spiral inlet (outlet for turbines)

- Inlet diameter $d_{\text{in}}$ ($d_4$)
- Inlet width $b_{\text{in}}$ ($b_4$)
- Abs. flow angle $\alpha_4$ (turbines)
- Automatic update from interface to apply changes from neighbor components

Please note:

For stand-alone volutes you have to define the inlet interface first, see Inlet Details, instead of specifying $d_{\text{in}}$ and $b_{\text{in}}$ values.

[for pumps, fans, compressors]

d_{\text{in}}$ and $b_{\text{in}}$ are suitable to the previous component outlet. If the previous component is an impeller $d_4$ and $b_4$ are determined using the ratios $d_4/d_2$ and $b_4/b_2$, which are calculated from functions dependent on the specific speed $n_q$ (see Approximation function).

Clicking on the Set Default button at top recalculates the standard values.

A short distance between the impeller and the cut-water is desirable for reasons of flow. For acoustic and vibration reasons, however, a certain minimum distance is necessary. The inlet width $b_{\text{in}}$ should be chosen such that the width/height ratio at the end cross-section of the volute is close to 1. The ratio $b_4/b_2$ can be varied within a relatively wide range without significant negative effect on the efficiency. For centrifugal impellers with open impeller sides, values up to $b_4/b_2=2$ are possible. At higher specific speeds (wider impellers), however, high width ratios have a negative effect on flow (intensive secondary flows, turbulence losses). In this case, $b_4/b_2$ should be between 1.05 and 1.2.

Values $d_{\text{in}}$ and $b_{\text{in}}$ are coupled to the corresponding interface values.

[for turbines]

d_{\text{out}}$ and $b_{\text{out}}$ has to be set by the user.

Information
Various calculated values are shown, for information purposes, on the right side (**Values**):

<table>
<thead>
<tr>
<th>Calculated internal flow rate</th>
<th>( Q_i )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet/Outlet diameter ratio</td>
<td>( \frac{d_{in}}{d_2} )</td>
</tr>
<tr>
<td>Inlet/Outlet width ratio</td>
<td>( \frac{b_{in}}{b_2} )</td>
</tr>
<tr>
<td>Inlet/Outlet meridional velocity</td>
<td>( c_m )</td>
</tr>
<tr>
<td>Inlet/Outlet circumferential velocity</td>
<td>( c_u )</td>
</tr>
<tr>
<td>Inlet/Outlet velocity</td>
<td>( c )</td>
</tr>
<tr>
<td>Inlet/Outlet flow angle</td>
<td>( \alpha )</td>
</tr>
</tbody>
</table>

**[for compressors, turbines]**

<table>
<thead>
<tr>
<th>Static pressure</th>
<th>( p )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature</td>
<td>( T )</td>
</tr>
<tr>
<td>Density</td>
<td>( \rho )</td>
</tr>
<tr>
<td>Mach Number</td>
<td>( Ma )</td>
</tr>
</tbody>
</table>

**Possible warnings**

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Swirl-free inflow is implausible for volute designs.</td>
<td>Adapt pre-swirl in the <strong>Global setup</strong>. With swirl-free inflow a volute calculation will be impossible.</td>
</tr>
</tbody>
</table>

**Thermodynamic state could not be calculated.**
For the chosen configuration of global setup, precursor component and spiral dimensions, a reasonable thermodynamic state cannot be calculated. Then an automatic velocity based contour design will not be possible since the necessary values of $c_{u4}$ and $Q_i$ are not available.

e.g. reduce the mass flow or increase the cross section

### 9.1.2 Inlet details

On page Inlet details the details of the inlet interface can be specified.

Details: see Interface Definition

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>For the chosen configuration of global setup, precursor component and</td>
<td>e.g. reduce the mass flow or increase the</td>
</tr>
<tr>
<td>spiral dimensions, a reasonable thermodynamic state cannot be</td>
<td>cross section</td>
</tr>
<tr>
<td>calculated. Then an automatic velocity based contour design will not</td>
<td></td>
</tr>
<tr>
<td>be possible since the necessary values of $c_{u4}$ and $Q_i$ are not</td>
<td></td>
</tr>
<tr>
<td>available.</td>
<td></td>
</tr>
</tbody>
</table>
Stand-alone volutes

For stand-alone volutes you have to define the inlet interface first (z and r at hub and shroud side), instead of specifying $d_{in}$ and $b_{in}$ values at page Setup.

By using the button you can transfer this interface definition to the geometry. On the right side on page Meridian you should see the desired inlet geometry now.

Diameter and width ratio

If the upstream component is an impeller then additional edit fields for the diameter ratio $d_4/d_2$ and width ratio $b_4/b_2$ are available. Here you can define the inlet diameter and the inlet width using empirical functions.

Information

Page Values of the right panel contains some information of the design point (Global setup) and flow properties on the outlet of upstream component.

9.2 Cross Section

The shape of the volute cross-sections can be selected here. The general cross section shape is illustrated whereas the radial extension is assumed.

In general, very small cross-sections width should be avoided. The achievable cross-section shape strongly depends on manufacturing and the available space.
The table contains the cross section definitions (at least 1 cross section). Each cross section is defined by:

- the circumferential position: angle \( \varphi \)
- (de)activation by selecting the checkbox on the left side (at least 1 cross section has to be active)
- cross section type on the left side
- optional cross section type on the right side or symmetric

The section definition is running in the range \( 0^\circ < \varphi \leq 360^\circ \). The section at \( \varphi=0^\circ \) is flat always - therefore a section definition at this position makes no sense.

Between 2 neighboring cross section definitions a smooth transition is realized. If only a single section is defined then this definition is used for all circumferential positions.

The following cross section types are available:
<table>
<thead>
<tr>
<th>Cross-Section Shape</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Rectangular</strong></td>
<td>Most simple cross-section shape; cannot be achieved in cast parts; only sensible for low specific speeds, since otherwise the cross-section becomes too large.</td>
</tr>
<tr>
<td><strong>Trapezoid</strong></td>
<td>Cannot be achieved in cast parts; the angle δ can be specified; results in a flatter cross-section than a rectangular cross-section, with less intense secondary flow.</td>
</tr>
</tbody>
</table>
| **Round - symmetric** | Simple elliptic geometry with a beneficial stress distribution; does not develop on rotation surfaces.  
  - **Ratio of ellipse radii**: axis ratio is preserved for developing sections. (ratio of one gives a circular section). |
| **Round - asymmetric, external** | More favorable secondary flow structure than with a symmetrical cross-section; often with mixed-flow impellers.  
  - **Ratio of ellipse radii**: axis ratio is preserved for developing sections. (ratio of one gives a circular section).  
  - **Strictly external**: cross sections don't fall below inlet radius |
- **Open to right**: asymmetric development to right (pos. z-direction)
- **Square on top**: square shape on right top of cross section

### Round - asymmetric, internal

Limitation of radial extension; ratio of ellipse radii possible;
additional bend necessary
see [Internal cross sections](#)

### Bezier - Rectangle type

analogous with **Rectangle**; with chamfers (cast radii)
see [Bezier cross section](#)

### Bezier - Trapezoid type

analogous with **Trapezoid**; with chamfers (cast radii)
see [Bezier cross section](#)
Section properties

Here you can specify some properties of the currently selected cross section in the table Sections. Details can be found in the table above.

Limitations

For double volutes the cone angle (opening) of all cross sections has to be constant. Therefore, round types and Line segments are not available.

If any of these impossible cross section types are already part of the project then they are converted automatically when selecting the double volute type (see Setup & Inlet). The following message will be displayed:

- "Volute section type(s) were modified due to double volute requirements."
  if any cross section type was modified automatically

- "Cone angle(s) were modified due to double volute requirements."
  if the cone angle of any cross section was adapted automatically
9.2.1 Bezier cross section

The shape of a Bezier cross-section is described by a Bezier curve.

One half of the shape of the cross-section is described using a 4th degree Bezier polynomial. Points 0 and 4 are the end points and cannot be changed. Point 1 can be moved along a straight line which corresponds to the cone angle of the cross-section (0° for a rectangle type, δ for a trapezoid type). Point 3 can only be moved in the horizontal direction in order to guarantee a smooth transition between the two symmetrical halves. The intersection of the two lines which points 1 and 3 are on is designated by the letter S and plays an important role in the positioning of Bezier points 1 and 3. Point 2 can be moved freely and therefore he has the major influence on the shape of the cross-section. In the first design, point 2 is identical with point S.

Two basic shapes of the cross-section can be selected, rectangular or trapezoid. Only the end cross-section of the volute is designed, all other cross-sections result from this. Under the heading Inner point position, you can select whether positioning of the inner points 1 and 3 should be relative (0..1; 0=point 0/4; 1=point S) or absolute (distance from point S). The numeric values of the positions can be changed by right-clicking on points 1 or 3. If the option Show all points under the heading Options is selected, the different positioning methods become apparent.

The minimum curvature radius of the designed contour is shown in the box to the bottom right.
9.2.2 Line Segments cross section

The shape of a Line segments cross-section is described by a series of line segments.

One half of the shape of the cross-section is initially based on line segments arranged in a trapezoid shape. Points 1 and 4 are the fix start- and endpoint.

All corner points are connected by line segments. The coordinates of each point and the related corner radius can be adjusted in the context dialog.
Coordinates and radius of vertex

Using the context menu of a line segment, points can be added at the cursor position or be removed:
When moving points the following constraints can be enforced by pressing a key on keyboard:

- **CTRL**: Point moves on a circle around the previous point. The radius stays constant while pressed.
- **CTRL + SHIFT**: Point moves on a circle around the next point. The radius stays constant while pressed.
- **ALT**: Point moves on a line between its last position and previous point.
- **ALT + SHIFT**: Point moves on a line between its last position and next point.

### 9.2.3 Radius based cross section

The shape of a radius based cross section is described by straight lines and circular arcs.

The geometry is described by the following parameters:
base height \( h \) height of the radial base part

base radius \( R_B \) rounding between base part and cone part (radius can be limited due to length of base part and cone part)

opening angle \( \delta \) angle of the cone part

corner radius \( R_C \) rounding between cone part and main circular arc on top (radius can be limited due to length of cone part and circular arc on top)

main radius \( R \) radius of main circular arc on top

9.2.4 Internal cross sections

Internal volutes are limited in its radial and axial extensions (see gray lines in the picture).

The additional bend can be described by the following parameters:

<table>
<thead>
<tr>
<th>Neck width</th>
<th>side distance from volute inlet to actual volute cross sections</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inner bend shape</td>
<td>shape of the inner bend wall</td>
</tr>
<tr>
<td>Ratio</td>
<td>semiaxis ratio for quarter bend</td>
</tr>
<tr>
<td>---------------------</td>
<td>---------------------------------</td>
</tr>
<tr>
<td>Outer bend shape</td>
<td>shape of the outer bend wall</td>
</tr>
<tr>
<td>Bend area ratio</td>
<td>ratio of outlet to inlet section of the bend</td>
</tr>
</tbody>
</table>

### 9.3 Spiral development areas

The spiral development areas can be designed and calculated in this window.

**Design mode**
The **wrap angle** can be defined - default value is 360°. Slightly reduced wrap angle can be helpful for cut-water design.

You can choose between two kinds of **Design rules** for volute calculation. **Velocity-based** rules are Pfleiderer and Stepanoff. **Geometry-based** rules are defined by section height, cross section area or ratio of area to center of mass radius.

→ **Details Design Rule**

### Cut-water compensation

In panel **Cut-water compensation** you can specify parameters for the cut-water design.

→ **Details Cut-water compensation**

### Circular arc approximation

For spirals with rectangular or trapezoidal cross sections, an **approximation by circular arcs** is provided.

→ **Details Circular arc approximation**

### Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spiral contour could not be calculated exactly.</td>
<td>Check geometry and &quot;Volute/ Inlet definition&quot;.</td>
</tr>
<tr>
<td>Spiral sections cannot be calculated due to unusual inflow direction or volute cross section definition.</td>
<td>Too narrow cross section shape can result in unreasonable high height-width-ratio. Try to select another cross section shape.</td>
</tr>
<tr>
<td><strong>Volute end cross section not sensible.</strong></td>
<td>Check geometry and &quot;Volute/ Inlet definition&quot;.</td>
</tr>
<tr>
<td>The properties of the end cross section are not reasonable, e.g. the ratio H5/B5 is too low or too high.</td>
<td>Check the properties of the end cross section.</td>
</tr>
<tr>
<td></td>
<td>See also the hints to the error &quot;It's not possible to calculate spiral contour exactly.&quot;</td>
</tr>
<tr>
<td>Problem</td>
<td>Possible solutions</td>
</tr>
<tr>
<td>------------------------------------------------------------------------</td>
<td>------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td><strong>Flow angle at volute inlet is too large.</strong></td>
<td>Check flow properties of design rule.</td>
</tr>
</tbody>
</table>
| Spiral sections cannot be calculated due to invalid reference parameters with a resulting flow angle of more than 70°. | For velocity-based design rules the flow angle at inner radius results from $c_m$ and $c_u$. While $c_m$ is calculated from volume flow $Q_i$  
\[ c_m = \frac{Q_i}{(d \cdot b \cdot \pi)} \]  
circumferential velocity is set directly. Increase $c_u$ or reduce $Q_i$ to decrease flow angle.  
\[ \tan(\alpha) = \frac{c_m}{c_u} \] |
| **Spiral contour calculation failed due to invalid inflow conditions.** | Check "Volute/ Inlet definition".                                                   |
| Spiral sections cannot be calculated due to invalid inflow direction, i.e. due to invalid flow angle. The flow angle at the volute inlet must be positive and should be small ($\angle \leq 45^\circ$, $90^\circ$ is completely invalid). It can be checked in "Volute/ Inlet definition", page "Volute" right at "Values": Flow angle $\alpha$. | The inlet flow angle is defined by the swirl of the previous component. If no previous component exists, the inflow angle is defined by "Global setup/ Inflow". Check if the swirl is inside the mentioned range. Contra rotating impeller at volute inlet are currently not supported. |
| **Angle of last cross section definition is higher than spiral wrap angle.** |                                                                                     |
| One or more cross sections are defined at positions $\varphi >$ spiral wrap angle $\varphi$ | Adapt circumferential position of the cross section definition ("Volute/ Cross section") or spiral wrap angle ("Volute/ Spiral areas"). |
| **Automated contour update is active.**                                | Cross section sizes are not fixed and adapt to external flow properties.            |
| Spiral cross section extents are updated automatically if anything on the inlet side or any spiral properties are modified. | To fix the spiral cross section extents you could uncheck the "Automatic" calculation right top. Then you have to manually start the calculation if required. |
9.3.1 Design rule

The flow rate through a cross-section, \( A \), of the circumferential angle, \( \phi \), is generally calculated as:

\[
Q_\phi = \int\int c_u dA = \int c_u b(r) dr
\]

Using \( Q_\phi = \phi / (2\pi) \int Q_i + Q_0 \) results in an equation to calculate the circumferential angle, \( \phi \), dependent on the outer radius \( r_a \):

\[
\phi = \frac{2\pi}{Q_i} \int_{r_i}^{r_a} c_u b(r) dr
\]

\( b(r) \) is a geometrical function which is defined according to the shape of the cross-section. The velocity \( c_u \) is chosen in accordance with the design instructions. Under Design rule, two alternatives can be selected.

**Velocity-based rules**
For all velocity-based rules the area for each cross section is calculated using a linearly increasing volume flow $Q_{\varphi}$ starting at $Q_{0}$ for $\varphi=0$ (blind volume flow) and an assumed velocity distribution $c$ over $r$. While $Q_{\varphi}$ depends on a total reference volume flow $Q_{i}$, the velocity distribution is defined by a reference $c_{u}$ at $r_{i}$ and one of the following velocity rules. Note that both reference values can be chosen manually by deactivating **Use flow properties from inlet**.

### Pfleiderer

Experience has shown that the losses can be greatly minimized if the volute housing is dimensioned such that the fluid flows in accordance with the principal of conservation of angular momentum. The cross-section areas are therefore designed in accordance with the principal of conservation of angular momentum, i.e. angular momentum exiting the impeller is constant. In addition, an exponent of angular momentum, $x$, can be chosen so that the principle $c_{u}r^{x} = \text{const.}$ is obeyed. When $x=1$, the angular momentum is constant. For the extreme of $x=0$, the circular component of the absolute velocity $c_{u}$ remains constant at the impeller outlet.

$$\varphi = \frac{2\pi c_{u,i} r_{i}^{x}}{Q_{i}} \int_{r_{i}}^{r_{i}^{(\varphi)}} \frac{b(r)}{r^{2}} dr \quad \Rightarrow \quad Q_{\varphi} = \int_{r_{i}}^{r_{i}^{(\varphi)}} c_{u,i} \left( \frac{r_{i}}{r} \right)^{x} b(r) dr \quad \Rightarrow \quad c_{u}(r) = c_{u,i} \left( \frac{r_{i}}{r} \right)^{x}$$

The integral can be explicitly solved for simple cross-section shapes (rectangles, trapezoids, circles). For other, arbitrary, shapes, it can be solved numerically.

### Stepanoff

Alternatively, it can be beneficial to design the volute with a constant velocity in all cross-sections of the circumference. According to Stepanoff, this constant velocity can be determined empirically:

$$c_{u} = k_{s} \sqrt{2gH}.$$  

The constant $k_{s}$ can be determined dependent on the specific speed $n_{q}$ (see **Approximation function**).
\[ \varphi = \frac{2\pi k_s \sqrt{2gH}}{Q_i} \int_{r_i}^{r_f} b(r) \, dr \quad \Rightarrow \quad Q_\varphi = k_s \sqrt{2gH} \int_{r_i}^{r_f} b(r) \, dr \quad \Rightarrow \quad c_u = k_s \sqrt{2gH} = \text{const.} \]

Note, for manually defined reference velocities \( c_u, k_s \) has no influence because \( c \) is constant over \( r \).

### Geometry-based rules

Contrary to velocity-based rules the geometry progression is defined directly. Height-, Area- or Area/Radius-progression (Set Progression) can be defined, where the final value can be entered.

#### 9.3.2 Cut-water compensation

The cut-water is available for external volutes only. For internal volutes the cut-water is a result of the intersection of spiral and diffuser.

Some initial cut-water parameters can be specified in the **Cut-water compensation** section:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inner radius ( r )</td>
<td>Informative, see Inlet. ( r ) is the inlet radius of the volute and/or outlet radius of radial diffusers</td>
</tr>
<tr>
<td>Thickness ( e )</td>
<td>Thickness of the cut-water at the start of the volute (for compensation)</td>
</tr>
<tr>
<td>Compensation ( \varphi_c )</td>
<td>Angle, above which cut-water correction begins (standard: 270°)</td>
</tr>
</tbody>
</table>
The cut-water does disturb the flow, since the cross-section of the flow is narrowed suddenly by the thickness of the cut-water.

To weaken this negative influence, the cut-water can be corrected. This is achieved by assuming that from the angle $\varphi_C$ the inner radius $r$ increases linearly to a value of $r+e$ at the end cross-section of the volute. This results in larger volute cross-sections in this area, so that the narrowing of flow caused by the cut-water becomes less significant.

By clicking on Default, you can return to the standard values for the cut-water.

9.3.3 Circular arc approximation

For spirals with rectangular or trapezoidal cross sections, an approximation by circular arcs is provided. The arcs are optimized with respect to the minimal deviation from the initial contour, which is defined by the design rule.

The number of circular segments can be specified and the max. deviation from the initial contour is displayed for information.

The Spiral contour view contains the circular segments, which can be displayed using the "Circle segments" display option bottom left.
Information about the resulting circular arcs (e.g. midpoints, radii and angles) are shown in the “informational values” view. In addition, their details are given as hint of the arc in the diagram if enabled in display properties.

Note that further calculations are based on the initial contour.

9.3.4 Double Volute

Double Volutes are used to compensate asymmetric casing forces that are inevitable for Single Volutes. Their design can be activated in the initial volute setup.
General procedure for Double Volute design

Double volutes are calculated analogously to Single Volutes. The blockage at splitter leading edge has to be compensated by splitter compensation (see parameters below), exactly like Cut-water compensation. Furthermore, the calculation of the outer contour is considering the geometry of the splitter (position, fillet-radius, thickness).

The inner radius of the splitter $r_{a,II}$ and thus the Inner area (II) at $\varphi$ is given by the outer radius $r_a$ at $\varphi - \varphi_{Spl}$.

The Outer area (I) is calculated based on the Design rule for
* a constant flow rate defined by the splitter start angle (normally 50% of overall flow rate)
* starting from the splitter outside radius $r_{i,I} = r_{a,II} + \Delta r$. 
For double volutes you can define additional properties of the spiral and splitter.

- The **start angle** $\phi_{Spl}$ is the angular position where the splitter starts. It also determines the splitter contour.
- The **angular offset** $\Delta \phi_{Spl}$ can be used to achieve a radial offset without changing the contour.
- The **thickness** $e_{Spl}$ defines the distance between the inner and outer splitter contour.
- The **compensation** $\phi_{Spl,C}$ is used analogous to the cut-water compensation.
- The **fillet radius** defines the radial corner radius between spiral and splitter surface.
**Additional views**

The progression diagrams contain curves for each part of the volute, like the area progression below.

[Graph showing area progression for volute parts]
Beside the default informational values separate values for inner and outer part of the volute are reported.

Furthermore 2 additional ratios are displayed:

- Expansion of outer volute (using end point of blue curve / start point of blue curve)
- Ratio of outer to inner throat (using end point of blue curve / end point of green curve)

### 9.3.5 Additional views

The following information can be displayed in the spiral dialog using the "Additional views" button:

#### 3D-Preview

**3D model** of the currently designed spiral development areas.

#### Informational values

Some informative values relating to the **end cross-section** are displayed:
<table>
<thead>
<tr>
<th>Inner radius</th>
<th>$r'$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outer radius</td>
<td>$r$</td>
</tr>
<tr>
<td>Min. axial</td>
<td>$z$</td>
</tr>
<tr>
<td>Height</td>
<td>$H$</td>
</tr>
<tr>
<td>Width</td>
<td>$B$</td>
</tr>
<tr>
<td>Side ratio</td>
<td>$H/B$</td>
</tr>
<tr>
<td>Equivalent diameter</td>
<td>$D$</td>
</tr>
<tr>
<td>Area</td>
<td>$A$</td>
</tr>
<tr>
<td>Area-radius-ratio</td>
<td>$A/r_c$</td>
</tr>
<tr>
<td>Volume flow</td>
<td>$Q$</td>
</tr>
<tr>
<td>Average velocity</td>
<td>$c$</td>
</tr>
<tr>
<td>Static pressure</td>
<td>$p$</td>
</tr>
<tr>
<td>Total pressure</td>
<td>$p_t$</td>
</tr>
<tr>
<td>Density</td>
<td>$\rho$</td>
</tr>
<tr>
<td>Temperature</td>
<td>$T$</td>
</tr>
<tr>
<td>Mach Number</td>
<td>$Ma$</td>
</tr>
<tr>
<td>Total temperature</td>
<td>$T_t$</td>
</tr>
</tbody>
</table>

**Spiral contour**  
Frontal view of volute (x-y)

**Cross sections**  
Volute cross sections (z-r)

**Radius progression**  
Radius distribution ($\phi$-$r$)

**Area progression**  
Area distribution ($\phi$-$A$)

**Area-radius-ratio**  
Area/Gravity center radius ($r_c$) distribution ($\phi$-$A/r_c$)

**Contour angle progression**  
Angle between the outer spiral contour and the circumferential direction ($\phi$-$\alpha$). Note, that due to the differential characteristic of the contour angle, the continuity of this distribution is decreased by one.
9.4 Diffuser

The geometry of the outlet or inlet diffuser can be designed and calculated in this design step.

Dimensions x,y-plane

In general, 3 basic shapes are available:
Tangential diffuser

Radial diffuser
The tangential diffuser is easier to manufacture, the radial diffuser has the advantage of minimizing tangential forces. The spline diffuser is similar to the radial but with extended flexibility.

**Tangential diffuser**

For the tangential diffuser the eccentricity can be specified:

- The right side is parallel to the center line (perpendicular to the last spiral cross section). The diffuser opens to left side only.
- The diffuser opens to both sides (default).
- The left side is parallel to the center line (perpendicular to the last spiral cross section). The diffuser opens to right side only.
- $\epsilon_{ab}$ The eccentricity can be specified manually.

**Radial diffuser**

In the case of a radial diffuser, the angle $\varepsilon$ between the outlet branch and the line connecting impeller-center and outlet branch center can be selected.

**Spline diffuser**

For the Spline-diffuser the angle $\phi$ between connecting line impeller-center $\leftrightarrow$ outlet branch center and diffuser start section has to be defined. Points 0 and 4 are start and endpoint of the middle line.
on the inlet and outlet cross section, point 2 is fixed by the intersection of appropriate perpendicul\mus of these sections. Position of points 1 and 3 influence the curve shape of the middle line.

By clicking on Default, you can return to the default values for the diffuser geometry.

The extension H of the diffuser can be defined in panel Dimensions x,y-plane.

For all diffuser shapes the extension is defined by the diffuser height $H$, which is the distance from the diffuser outlet to a parallel line through the center point.

The distance $C$ from the H-line to the center point is displayed for information, both in the diagram and numerical in the Information panel.

The starting position of the diffuser is defined by the angle $\phi_0$, whereas $0^\circ$ is horizontal right. The whole volute can be rotated by this value. By using the button Vertical outflow direction the volute can be rotated for vertical direction of the pressure joint.
Additionally, the diffuser can be shifted in radial direction by the radial offset \( \Delta r_c \) to reduce the intersection of spiral and diffuser.

This radial offset can increase the available space for the cut-water in order to locate it closer to the desired radius.

Radial offset is available for strictly external volutes with 360° wrap angle only.

**Dimensions z-direction**

Two options are available for aligning the diffuser in z-direction, bend and offset (rake).

<table>
<thead>
<tr>
<th>Dimensions</th>
</tr>
</thead>
<tbody>
<tr>
<td>( x,y )-plane | ( z )-direction |</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>Bend</th>
<th>Offset</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bend angle</td>
<td>( \alpha )</td>
<td>45.0°</td>
</tr>
<tr>
<td>Bend radius</td>
<td>( R )</td>
<td>198.86 mm</td>
</tr>
<tr>
<td>Straight inlet</td>
<td>( L_1 )</td>
<td>20 %</td>
</tr>
<tr>
<td>Bend length</td>
<td>( L_2 )</td>
<td>80 %</td>
</tr>
<tr>
<td>Straight outlet</td>
<td>( L_3 )</td>
<td>0 %</td>
</tr>
</tbody>
</table>

**Bend**

The diffuser bending in z-direction is described by the parameters shown in the sketch. There exist 2 straight segments 1, 3 and a circular segment 2.

- The lengths \( L_1, L_2 \) and \( L_3 \) are specified as percentage.
- The curvature is defined by the radius \( R \), the direction by the angle \( \alpha \).
- The z-bend is illustrated in the diagram by a green center line.
Offset

There are 4 offset options:

1. The diffuser sections are shifted in negative z-direction in order to avoid higher z-coordinate compared to the last spiral section.

2. The diffuser is developing centric with respect to the z-direction (default).

3. The diffuser sections are shifted in positive z-direction in order to avoid lower z-coordinate compared to the last spiral section.

4. The offset can be specified manually in both directions. Additionally the progression of the diffuser can be chosen: linear or quadratic with respect to the length of the diffuser.

The following image illustrates this feature: left centric design, right negative offset.
End cross-section

The end cross-section of the diffuser can be either round or rectangular. The diameter $D$ can be directly defined or selected from standard tables. In the case of a rectangular end cross-section the height and width can be chosen.

Section progression

The position of end shape specifies the percentage position along the diffuser, where the type of end cross section is reached (default = 100%). To reach certain cross section areas a scaling of those sections is necessary. Instead of just scaling uniformly in both directions ($z$ and $r$) a scaling ratio ($z/r$ growth) can be defined.

The choice of the area progression influences the scaling of the morphed cross sections.

<table>
<thead>
<tr>
<th>Linear blending</th>
<th>The morph between two different cross sections is linear which results in an quadratic area progression. (unscaled)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear area</td>
<td>The size of the morphed cross sections is scaled to achieve a linear area progression.</td>
</tr>
<tr>
<td>Quadratic area</td>
<td>The size of the morphed cross sections is scaled to achieve a quadratic progression from the diffuser inlet to the end shape position. The progression to diffuser outlet is linear again.</td>
</tr>
<tr>
<td>Custom area</td>
<td>The size of the morphed cross sections is scaled with respect to a Beziér curve.</td>
</tr>
</tbody>
</table>
**Splitter of Double Volute**

The position of splitter end defined the relative length of the splitter inside the diffuser.

### 9.4.1 Additional views

The following information can be displayed in the diffuser dialog using the "Additional views" button:

<table>
<thead>
<tr>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>Informational values</td>
</tr>
<tr>
<td>3D-Preview</td>
</tr>
<tr>
<td>Cross sections</td>
</tr>
<tr>
<td>Area progression</td>
</tr>
</tbody>
</table>

#### 3D-Preview

3D model of the currently designed diffuser geometry as well as spiral surfaces.

#### Informational values

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Formula/Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deceleration ratio AR</td>
<td>$A_R = (D_{in} / D_{out})^2$</td>
</tr>
<tr>
<td>Length L</td>
<td>Length of the diffuser</td>
</tr>
<tr>
<td>Angle to middle $\varphi$</td>
<td>Angle between connecting line impeller-center $\leftrightarrow$ outlet branch center and diffuser start section</td>
</tr>
<tr>
<td>Center distance C</td>
<td>Distance from the h-line to the center point</td>
</tr>
<tr>
<td>Cone angle $\varphi$</td>
<td>Cone angle from $D_{in}$ to $D_{out}$ over the length L</td>
</tr>
<tr>
<td>Max. theo. cone angle $\varphi_{max}$</td>
<td>$\varphi_{max} = 16.5^\circ(D_{in}/2/L)^{1/2}$, see Gülich, not for turbines</td>
</tr>
<tr>
<td>Diffuser radius $R$</td>
<td>Radius of middle line (for radial diffuser only)</td>
</tr>
<tr>
<td>Diffuser outlet center $C$</td>
<td>Spatial position of diffuser outlet</td>
</tr>
</tbody>
</table>
Inlet & Outlet information

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Equivalent diameter $D$</td>
<td>Diameter of the equivalent circle at the diffuser inlet / outlet</td>
</tr>
<tr>
<td>Area $A$</td>
<td>Area at diffuser inlet / outlet</td>
</tr>
<tr>
<td>Velocity $c$</td>
<td>Velocity at the diffuser inlet / outlet</td>
</tr>
<tr>
<td>Pressure $p$</td>
<td>Pressure at the diffuser inlet / outlet</td>
</tr>
<tr>
<td>Total pressure $p_t$</td>
<td>Total pressure at the diffuser inlet / outlet</td>
</tr>
</tbody>
</table>

For turbines and compressor additional information is visible:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature $T$</td>
<td>Temperature at the diffuser inlet / outlet</td>
</tr>
<tr>
<td>Total temperature $T_t$</td>
<td>Total temperature at the diffuser inlet / outlet</td>
</tr>
<tr>
<td>Density $\rho$</td>
<td>Density at the diffuser inlet / outlet</td>
</tr>
<tr>
<td>Mach Number $Ma$</td>
<td>Mach Number at the diffuser inlet / outlet</td>
</tr>
</tbody>
</table>

Cross sections

Diffuser cross sections (z-r)

Area progression

Area distribution (l-A)

9.5 Cut-water

? VOLUTE | Cut-water

The geometry of the cut-water can be designed in this dialog box.
Generally, the cut-water can be designed in three modes: Simple, Fillet or Sharp.

**Splitter of Double volute**

The leading/trailing edge axis ratio specifies the ratio between the minor and major axis length of an ellipse, representing the leading and trailing edge of the splitter.

**Limitations**

<table>
<thead>
<tr>
<th>General</th>
<th>The wrap angle must be at least 330°.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Simple</td>
<td>For cornered spiral cross sections the side position is fixed to the corner position and cannot be modified individually.</td>
</tr>
<tr>
<td></td>
<td>Rounding of cut-water edges (Round edges) is possible only if side position is higher than the position of maximum curvature and if no diffuser radial offset is defined.</td>
</tr>
<tr>
<td>Fillet, Sharp</td>
<td>Not available for cornered cross sections, either spiral or diffuser.</td>
</tr>
<tr>
<td></td>
<td>Intersection of spiral and diffuser geometry is necessary to create the cut-water.</td>
</tr>
</tbody>
</table>
Fillet cut-water is usually not possible, if the spiral development is at the beginning very flat and a tangential diffuser with a big end cross-section is chosen.

For asymmetric spiral cross sections, only non-tangential surface transition is available.

### Possible warnings

<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Intersection of Side A1/B1 with Inlet/Outlet</strong></td>
<td></td>
</tr>
<tr>
<td>Cut-water Side A1/B1 face intersects with Inlet/Outlet face.</td>
<td>Any parameter modification of the Fillet Cut-water affects the geometry of the side patches.</td>
</tr>
<tr>
<td>Inlet/Outlet is a virtual face bordering the Volutes’s Flow domain. The intersection affects volumetric meshing of the Flow domain, which may result in very small mesh entities and in a non smooth surface in these regions.</td>
<td>Use automatic Surface transition.</td>
</tr>
<tr>
<td>Generally, the intersection is very small and hardly visible in 3D-model.</td>
<td></td>
</tr>
</tbody>
</table>

**The cut-water cannot be created because the spiral wrap angle is too low (minimum: ...).**

- A minimum spiral wrap angle is necessary for cut-water creation.
  - Increase spiral warp angle at least to the given minimal value.

**Cutwater is self-intersecting.**

- Cut-water faces intersect each other.
  - The problem might have various reasons. Therefore, modify spiral, diffuser or cutwater design.
  - E.g. define a flat radius progression at the start of spiral development areas, or change angular position / radial offset of the cutwater.
<table>
<thead>
<tr>
<th>Problem</th>
<th>Possible solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Multiple intersection curves between spiral and diffuser.</td>
<td>This problem can be solved by manipulating the <strong>spiral start position</strong>. A single intersection curve is required for generation of <strong>sharp</strong> and <strong>fillet</strong> cut-waters.</td>
</tr>
</tbody>
</table>

Spiral and diffuser may have multiple intersection curves. (red)

![Diagram of spiral and diffuser](image1.png)

---

**3D-Error: Could not create bounded surface for Cut-water!**

Parameter **side position** is disadvantageous.

The **side position** should not be too low when edges are rounded.

---

**3D-Error: Creation of Cut-water failed! Possibly, the fillet radius is too large.**

[for asymmetric volutes]

Fillet cannot be created because intersection curve of spiral and diffuser is wavy.

Modify the **Position of end shape** in the Diffuser dialog to avoid wavy intersection curve.

![Diagram of cut-water](image2.png)
### Problem

*for asymmetric volutes*

Fillet cannot be created because intersection curve of spiral and diffuser is tangential to the sharp diffuser edge.

### Possible solutions

- Modify **Spiral start position**

---

<table>
<thead>
<tr>
<th>Creating fillet cutwater requires intersection between diffuser and spiral contour.</th>
</tr>
</thead>
<tbody>
<tr>
<td>There is no intersection of original spiral surface and diffuser surface.</td>
</tr>
<tr>
<td>The fillet cut-water is generated along this intersection curve.</td>
</tr>
<tr>
<td>The shape of spiral and diffuser determine the intersection.</td>
</tr>
<tr>
<td>The diffuser shape is additionally influenced by the &quot;Radial offset&quot; (reduce if possible) and the &quot;Diffuser base form factor&quot; (enlarge if possible).</td>
</tr>
</tbody>
</table>

---

#### 9.5.1 Simple

The simple cut-water is a rounding-off between spiral and diffuser.
The rounding position is defined by the angular position $\varphi_{C,0}$ ($0^\circ=$ start of volute).

**Side position** defines the transition position from the central rounding surface to the side surfaces. For asymmetric spiral cross sections two independent values can be specified for left and right side.

The created edge can be rounded optionally (**Round edges**).
The cut-water height has a similar effect like side position and defines the transition position of the cut-water surface on the spiral outlet.
The cut-water itself is designed by a 4th order Bezier curve. The shape can be modified interactively after zooming in (Zoom Cut-water).

### 9.5.2 Fillet

For fillet cut-water design the spiral and the diffuser are trimmed and rounded at their intersection curve.
Prerequisites:

- The wrap angle must be high enough so that spiral and diffuser intersect.

The corresponding fillet radius can be specified.

Additionally, the Diffuser base form factor defines the roundness of the first diffuser cross section on its base side and is between 0.2 and 1:

- 0 = cornered base side (like spiral section)
- 1 = full rounded base side

The factor affects the shape of the intersection curve and therefore the shape of the cut-water. A high value could help to get an intersection of spiral and diffuser as a pre-condition for fillet cut-water.
The **Spiral start position** indicates the angular position at which the spiral begins and influences the intersection of spiral and diffuser. It has to be at least 1° and must be lower than the intersection position of spiral and diffuser. If **Automatic** is activated the optimal angular position is determined automatically.

The **Surface transition** defines the transition from the side patch surfaces to the central fillet surface:

- Tangential: Tangential transition between both surfaces (Time-consuming)
9.5.3 Sharp

For sharp cut-water design the spiral and the diffuser are trimmed only at their intersection curve. The resulting geometry can be processed in the CAD system.
Prerequisites:

- The wrap angle must be high enough so that spiral and diffuser intersect.

9.5.4 Additional views

The following information can be displayed in the cut-water dialog using the "Additional views" button:

- Informational values
- 3D-Preview
- Cross sections
- Area progression

Informational values

Some informative values relating to both end cross-sections and the cut-water cross-section are displayed:

<table>
<thead>
<tr>
<th>Equivalent diameter</th>
<th>D</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cross-section area</td>
<td>A</td>
</tr>
</tbody>
</table>

Especially for cut-water, some diameters and angle are displayed:
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Expression</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inner / Outer / Average cut-water angle</td>
<td>$\alpha_{in} / \alpha_{out} / \alpha_{avg}$</td>
</tr>
<tr>
<td>Inner / Outer / Average cut-water diameter</td>
<td>$d_{in} / d_{out} / d_{avg}$</td>
</tr>
<tr>
<td>Minimal cut-water diameter</td>
<td>$d_{min}$</td>
</tr>
<tr>
<td>Cutwater angular position</td>
<td>$\varphi_{C,1}$</td>
</tr>
</tbody>
</table>

**Diagram:**
- Spiral
- Diffuser
- Throat
- $\alpha_{in}$
- $d_{in}$
- $d_{out}$
- $d_{avg}$
- $\alpha_{out}$
- $d_{avg}$
- $\varphi_{C,1}$
9.6 CFD setup

For flow simulation (CFD), the diffuser can be extruded in normal direction of its outlet. The Length can be specified either absolutely or as a multiple (Factor) of the diffuser outlet diameter D6.
If a RSI connection was enabled for the impeller, these geometrical parts are added to the virtual geometry of the volute (see virtual geometry for more information).

9.7 Model settings

On dialog Model settings you can specify how many data points are to be used for the 3D model and for the point based export formats.

The number of points can be set for both cases separately for all geometry parts.

- **Spiral**: cross sections, points per cross section
- **Diffuser**: cross sections
- **Cutwater (sides)**: cross sections, points per cross section

*The cutwater cross sections setting does not refer to the center face, because its section count is determined by the number of points of the spiral and by the side position.*
3D Model

Distance tolerance
The distance tolerance defines the maximum allowed distance between sewed surfaces, e.g., the faces of a solid.

If it is too small, the solids cannot be created.

If it is too big, small faces are ignored when creating a solid.

Point Export

Presetting
Select from 3 global presettings.

When a new volute is created, the model settings of the last opened volute are adopted.
Part X
10 Appendix

10.1 References

**GENERAL**

Willi Bohl, Wolfgang Elmendorf
Strömungsmaschinen 1+2
Vogel-Verlag, 2008

Werner Fister
Fluidenergiemaschinen Bd. 1 und 2
Springer-Verlag, 1984 und 1986

Wolfgang Kalide
Energieumwandlung in Kraft- und Arbeitsmaschinen
Hanser-Verlag, 1989

Carl Pfeiderer, Hartwig Petermann
Strömungsmaschinen
Springer-Verlag, 1991

Joachim Raabe
Hydraulische Maschinen und Anlagen
VDI-Verlag, 1989

Arnold Whitfield, Nicholas C. Baines
Design of Radial Turbomachines
Longman Scientific & Technical, 1990

**PUMPS**

Johann F. Gülich
Kreiselpumpen
Springer-Verlag, 1999

Kurt Holzenberger, Klaus Jung
Kreiselpumpen Lexikon
KSB AG, 1989

Val S. Lobanoff, Robert R. Ross
Centrifugal Pumps, Design & Application
Gulf Professional Publishing, 1992

Michael Schwanse
Strömungsmechanische Auslegung und Nachrechnung von radialen und diagonalen
Kreiselpumpenlaufrädern
A. J. Stepanoff  
Centrifugal and Axial Flow Pumps  
John Wiley & Sons, 1957

John Tuzson  
Centrifugal pump design  
John Wiley & Sons, 2000

Walter Wagner  
Kreiselpumpen und Kreiselpumpenanlagen  
Vogel-Verlag, 1994

Gotthard Will  
Kreiselpumpen  
in: Taschenbuch Maschinenbau, Band 5  

FANS

Leonhard Bommes, Jürgen Fricke, Reinhard Grundmann  
Ventilatoren  
Vulkan-Verlag, 2003

Bruno Eck  
Ventilatoren  
Springer-Verlag, 1991

Thomas Carolus  
Ventilatoren  
Teubner-Verlag, 2003

Hans Werner Roth  
Optimierung von Trommelläufer-Ventilatoren  
Dissertation, Universität Karlsruhe, 1980

R.A. Wallis  
Axial flow fans and ducts  
John Wiley&Sons, 1983

COMPRESSORS

Ronald H. Aungier  
Centrifugal Compressors  
ASME Press, 2000

Ronald H. Aungier  
Axial-flow Compressors  
ASME Press, 2003
Klaus H. Lüdtke
Process Centrifugal Compressors
Springer-Verlag, 2004

Bruno Eckert, Erwin Schnell
Axial- und Radialkompressoren
Springer-Verlag, 1980

Davide Japikse
Centrifugal Compressors Design and Performance
Concepts ETI, 1996

N. A. Cumpsty
Compressor aerodynamics
Krieger publishing, 2004

Ernst Lindner
Turboverdichter
in: Taschenbuch Maschinenbau, Band 5

Arthur J. Wennerstrom
Design of highly loaded axial-flow fans and compressors
Concepts ETI, 2000

Members of the staff of Lewis Research Center
Aerodynamic design of axial-flow compressors
NASA SP-36, Washington, D.C. 1965

P. de Haller
Das Verhalten von Tragflügelgittern in Axialverdichtern und im Windkanal
Brennstoff-Wärme-Kraft, Band 5, Heft 10, 1953

Michael Casey and Chris Robinson
A method to estimate the performance map of a centrifugal compressor stage

Jakob Ackeret
Zum Entwurf dicht stehender Schaufelgitter
Schweizer Bauzeitung 120 (1942), S. 103-108

TURBINES

Ronald H. Aungier
Turbine Aerodynamics
ASME Press, 2006

Hany Moustapha, Mark Zelesky, Nicholas C. Baines, Davide Japikse
Axial and Radial Turbines
Concepts NREC, 2003
O. E. Balje
A study on design Criteria and matching of Turbomachines: Part A - Similarity Relations and Design Criteria of Turbines

Further literature

John D. Stanitz, Vasily D. Prian
A rapid approximate method for the determining velocity distribution on impeller blades of centrifugal compressors
NACA Technical note 2421; July 1951

John David Anderson, R. Grundmann, E. Dick
Computational Fluid Dynamics: An Introduction
Springer, 1996

Redlich, O., Kwong, J.N.S.

Aungier, R.H.
A Fast, Accurate Real Gas Equation of State for Fluid Dynamic Analysis Applications,

Soave, G.
Equilibrium constants from a modified Redlich-Kwong equation of state.,
Chemical Engineering Science. 27, No. 6, pp. 1197–1203, 1972

Peng, D.Y., Robinson, D.B.
A New Two-Constant Equation of State,

Von Backström, Th. W.
A unified correlation for slip factor in centrifugal impellers

Mason, E.A., u. S. C. Saxena
Approximate formulae for the thermal conductivity of gas mixtures
Phys. Fluids 1 (1958), 361

Waldemar Steinhilper, Rudolf Röper
Maschinen- und Konstruktionselemente 3
Springer, 1996
## 10.2 Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\alpha)</td>
<td>Angle of absolute flow</td>
</tr>
<tr>
<td>(\beta)</td>
<td>Angle of relative flow</td>
</tr>
<tr>
<td>(\gamma)</td>
<td>slip coefficient, Stagger angle</td>
</tr>
<tr>
<td>(\delta)</td>
<td>Deviation angle flow / blade</td>
</tr>
<tr>
<td>(\delta_r)</td>
<td>Swirl number</td>
</tr>
<tr>
<td>(\epsilon)</td>
<td>Drag ratio</td>
</tr>
<tr>
<td>(\eta)</td>
<td>Efficiency</td>
</tr>
<tr>
<td>(\mu)</td>
<td>Decreased power factor (sweeping)</td>
</tr>
<tr>
<td>(\nu)</td>
<td>Diameter ratio</td>
</tr>
<tr>
<td>(\pi)</td>
<td>Pressure ratio</td>
</tr>
<tr>
<td>(\rho)</td>
<td>Density</td>
</tr>
<tr>
<td>(\sigma)</td>
<td>Thickness in circumf. direction; Speed coefficient</td>
</tr>
<tr>
<td>(\tau)</td>
<td>Obstruction of flow channel by blades</td>
</tr>
<tr>
<td>(\varphi)</td>
<td>Wrap angle; Flow coefficient</td>
</tr>
<tr>
<td>(\psi)</td>
<td>Work coefficient (= pressure and head coefficient)</td>
</tr>
<tr>
<td>(\omega)</td>
<td>Angular velocity</td>
</tr>
<tr>
<td>(\Lambda, \sigma, \delta, \nu)</td>
<td>Sweep angles</td>
</tr>
<tr>
<td>(A)</td>
<td>Cross section area</td>
</tr>
<tr>
<td>(b)</td>
<td>Width</td>
</tr>
<tr>
<td>(c)</td>
<td>Absolute velocity</td>
</tr>
<tr>
<td>(c_m)</td>
<td>Meridional velocity ((c_m=w_m))</td>
</tr>
<tr>
<td>(c_u)</td>
<td>Circumferential component of absolute velocity</td>
</tr>
<tr>
<td>(c_L)</td>
<td>Lift coefficient</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>--------------------------------------------------</td>
</tr>
<tr>
<td>$c_D$</td>
<td>Drag coefficient</td>
</tr>
<tr>
<td>d</td>
<td>Diameter</td>
</tr>
<tr>
<td>f</td>
<td>Camber</td>
</tr>
<tr>
<td>F</td>
<td>Force</td>
</tr>
<tr>
<td>h</td>
<td>Enthalpy</td>
</tr>
<tr>
<td>H</td>
<td>Pump head</td>
</tr>
<tr>
<td>i</td>
<td>Incidence angle</td>
</tr>
<tr>
<td>l</td>
<td>Chord length</td>
</tr>
<tr>
<td>L</td>
<td>Length</td>
</tr>
<tr>
<td>M</td>
<td>Torque; Meridional coordinate</td>
</tr>
<tr>
<td>m</td>
<td>Meridional coordinate (dimensionless)</td>
</tr>
<tr>
<td>$?_q?_s$</td>
<td>Mass flow</td>
</tr>
<tr>
<td>n</td>
<td>Number of revolutions</td>
</tr>
<tr>
<td>$n_q?_s$</td>
<td>Specific speed</td>
</tr>
<tr>
<td>p</td>
<td>Pressure</td>
</tr>
<tr>
<td>P</td>
<td>Power</td>
</tr>
<tr>
<td>Q</td>
<td>Flow rate</td>
</tr>
<tr>
<td>r, R</td>
<td>Radius</td>
</tr>
<tr>
<td>s</td>
<td>Entropy, Orthogonal thickness</td>
</tr>
<tr>
<td>S</td>
<td>Static moment</td>
</tr>
<tr>
<td>t</td>
<td>Pitch; Tangential coordinate (dimensionless)</td>
</tr>
<tr>
<td>T</td>
<td>Temperature, Tangential coordinate</td>
</tr>
<tr>
<td>u</td>
<td>Circumferential velocity (Rotational speed)</td>
</tr>
<tr>
<td>v</td>
<td>Velocity</td>
</tr>
<tr>
<td>w</td>
<td>Relative velocity</td>
</tr>
</tbody>
</table>
# Appendix

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$w_u$</td>
<td>Circumferential component of relative velocity ($w_u + c_u = u$)</td>
</tr>
<tr>
<td>$Y$</td>
<td>Specific energy</td>
</tr>
<tr>
<td>$z$</td>
<td>Geodetic height; Number of blades</td>
</tr>
</tbody>
</table>

## 10.3 Contact addresses

www.cfturbo.com  
info@cfturbo.com

**CFturbo GmbH**

Unterer Kreuzweg 1  
01097 Dresden, Germany  
Phone +49 351 40 79 04 79

**CFturbo, Inc.**

NEWLAB, 19 Morris Avenue, Building 128  
Brooklyn Navy Yard  
Brooklyn, NY 11205, USA  
Phone +1 929 351 2009

## 10.4 License agreement

**Software Cession and Maintenance Contract**

between  

CFturbo GmbH  
Unterer Kreuzweg 1, 01097 Dresden (Germany)
- hereinafter designated the 'Licensor' -

and

the CFturbo user

- hereinafter designated the 'User' -

§ 1 LICENSE AGREEMENT

By virtue of this agreement, the User acquires from the Licensor the non-transferable and non-exclusive right to use the software 'CFturbo' (hereinafter designated the 'Software') for a period of time, in exchange for the licence fee agreed between the Licensor and the User.

1. Licence Object

The User acquires a nodelocked license or a license for one local office network (LAN) at one distinguished location of the company.

The program package consists of a data medium (CD-ROM or DVD) with the Software and a user manual in the form of a PDF file. In the event that the Software was downloaded from the official website of the Licensor, the program package consists of the corresponding installation file including electronic documentation.

2. Duration / commencement of the licence

The User obtains the right to use the Software. The right is obtained after the payment of the full licence fee and implicitly expires at the end of the arranged time period.

4. Right of Use

(1) In accordance with this contract, the Licensor grants the User a right of use to the Software described under 1. as well as a right to use the necessary printed matter and documentation. The printing-out of the manual for the purposes of working with the Software is permitted.

(2) The User may duplicate the Software only insofar as the duplication in question is necessary for the use of the Software. Necessary reasons for duplication notably include the installation of the Software from the original data medium onto the mass storage of the hardware used, as well as the loading of the Software into the RAM memory.
(3) The User is entitled to perform duplication for backup purposes. However, in principle, only a single backup copy may be created and stored. The backup copy must be labelled as being a backup copy of the ceded Software.

(4) If, for reasons of data security or the assurance of a fast reactivation of the computer system after a total failure, the regular backing-up of the entire dataset including the computer programs used is essential, then the User may create the number of backup copies which are compulsorily required. The data media concerned must be labelled accordingly. The backup copies may only be used for purely archival purposes.

(5) The User is obliged to take appropriate measures to prevent the unauthorized access of third parties to the program including its documentation. The supplied original data media, as well as the backup copies, must be stored in a location protected against the unauthorized access of third parties. The employees of the User must be explicitly encouraged to observe these contractual conditions as well as the provisions of copyright law.

(6) Additional duplications, also including the printing-out of the program code on a printer, must not be created by the User. The copying and the handover or transfer of the user manual to third parties is not permitted.

5. Multiple Use and Networks

(1) The User may use the Software on any hardware available to him, provided that this hardware is appropriate for the use according to the Software documentation. In the event of changing the hardware, the Software must be erased from the previously used hardware.

(2) The simultaneous reading in, storage or use on more than one hardware device is not permitted unless the User has acquired multiple-use licences or network licences. Should the User wish to use the Software on multiple hardware configurations at the same time, for example to permit the use of the Software by several employees, he must purchase the corresponding number of licences.

(3) The use of the ceded Software on different computers on a network or another multiple-workstation computer system is permitted, provided that the User has purchased multiple-use licences or network licences. If this is not the case, the User may only use the Software on a network if he prevents simultaneous multiple use by means of access protection mechanisms.

6. Program Modifications

(1) The disassembly of the ceded program code into other code forms (decompilation) as well as other types of reverse-engineering of the different manufacturing stages of the software, including a modification of the program, is not permitted.

(2) The removal of the copy protection or similar protection mechanisms is not permitted. Insofar as the trouble-free use of the program is impaired or hindered by one of the protection mechanisms, the Licensor is obliged to remedy the fault on an appropriate request. The User bears the burden of proof of the impairment or hindrance of trouble-free usability as a result of the protection mechanism.
(3) Copyright notices, serial numbers and other marks used for program identification purposes must in no event be removed or modified. This also applies to the suppression of the screen display of such marks.

7. Resale and Leasing

Resale and leasing of the Software or other cession of the Software to third parties is only permitted with the written agreement of the Licensor.

8. Warranty

(1) The Licensor makes no warranty with respect to the performance of the Software or the obtained data and the like. He grants no guarantees, assurances or other provisions and conditions with respect to the merchantability, freedom from defects of title, integration or usability for specific purposes, unless they are legally prescribed and cannot be restricted.

(2) Defects in the ceded software including the user manuals and other documents must be remedied by the Licensor within an appropriate period of time following the corresponding notification of the defect by the User. The defect is remedied by free-of-charge improvements or a replacement delivery, at the discretion of the Licensor.

(3) For the purposes of testing for and remedying defects, the User permits the Licensor to access the Software via telecommunications. The connections necessary for this are established by the User according to the instructions of the Licensor.

(4) A right of cancellation of the User due to the non-granting of use according to § 543 para. 2 clause 1 no. 1 of the Civil Code is excluded insofar as the improvement or replacement delivery is not to be regarded as having failed. Failure of the improvement or replacement delivery is only to be assumed if the Licensor was given sufficient opportunity to make the improvement or replacement delivery.

(5) Furthermore, the statutory regulations also apply.

9. Liability

(1) The claims of the User for compensation or replacement of futile expenditure conform, without regard to the legal nature of the claim, to the existing clause.

(2) In the Software, it is a question of a design procedure. It is considered to be purely an approximation method. The Licensor is not liable for the functioning of the data obtained in practice, for the manufactured prototypes or components, or for possible consequential damages resulting therefrom.

(3) The Licensor is liable for damage involving injury to life and limb or to health, without limitation, insofar as this damage is the result of a negligent or intentional breach of obligation on the part of the Licensor or one of his legal representatives or vicarious agents.
(4) Otherwise, the Licensor is liable only for gross negligence and deliberate malfeasance.

(5) Liability for consequential damages due to defects is excluded.

(6) The above regulations also apply in favour of the employees of the Licensor.

(7) The liability according to the Product Liability Act (§ 14 ProdHaftG) remains unaffected.

(8) The liability of the Licensor regardless of negligence or fault for defects already existing on entering into the contract according to § 536 a para. 1 of the Civil Code is expressly excluded.

10. Inspection Obligation and Notification Obligation

(1) The User will inspect the delivered Software including its documentation within 8 working days after delivery, in particular with regard to the completeness of the data media and user manuals as well as the functionality of the basic program functions. Defects determined or detectable hereby must be reported to the Licensor within a further 8 working days by means of a registered letter. The defect notification must contain a detailed description of the defects.

(2) Defects which cannot be detected in the context of the described appropriate inspection must be reported within 8 working days of their discovery with observance of the notification requirements specified in paragraph 1.

(3) In the event of the violation of the inspection and notification obligation, the Software is considered to be approved with regard to the defect concerned.

11. Intellectual Property, Copyright

The Software and all the authorized copies of this Software made by the User belong to the Licensor and are the intellectual property of the latter. The Software is legally protected. Insofar as it is not expressed stated in this contract, the User is granted no ownership rights to the Software, and all rights not expressly granted by means of this contract are reserved by the Licensor.

12. Return

(1) At the end of the contractual relationship, the User is obliged to return all of the original data media as well as the complete documentation, materials, and other printed matter ceded to him. The program and its documentation must be delivered to the lessor free of charge.

(2) The appropriate return also includes the complete and final deletion of all installation files and online documentation, as well as any copies that may exist.

(3) The Licensor may dispense with the return and order the deletion of the program and the destruction of the documentation. If the Licensor exercises this elective right, he will explicitly inform the User to this effect.
The User is expressly advised that, after the end of the contractual relationship, he may not continue to use the Software and, in the event of non-compliance, is violating the copyright of the copyright holder.

§ 2 SOFTWARE MAINTENANCE

The Licensor performs the maintenance and upkeep of the Software modules included in this contract under the following conditions. The maintenance of computer hardware is not the subject matter of this contract.

1. Scope of the maintenance obligation

(1) The contractual maintenance measures include:

a) The provision of the respectively newest program versions of the Software modules named under § 1 no. 1 as free-of-charge downloads. The Software is installed by the User.

b) The updating of the Software documentation. Insofar as a significant change to the functional scope or operation of the software occurs, completely new documentation will be provided.

c) On the expiration of the defect liability period resulting from the Software cession contract, the remedying of defects both in the program code and in the documentation.

d) Both the written (also by fax or e-mail) and telephone advising of the customer in the event of problems regarding the use of the Software as well as any program errors that may need to be recorded.

e) The telephone advice service (‘hotline’) is available to customers on working days between 9.00 a.m. and 4.00 p.m. (CET).

f) Defects reported in writing or requests for advice are answered no later than the afternoon of the working day following their receipt. As far as possible, this occurs by telephone for reasons of speed. The customer must therefore add the name and direct-dial telephone number of the responsible employee to every written message. For defect reports or requests for advice sent by e-mail, the answer may also be given by e-mail.

(2) The following services, among others, are not included in the contractual maintenance services of the contractor:

a) Provision of advice outside of the working hours specified under § 2 para. 1 letter e).

b) Maintenance services which become necessary due to the use of the Software on an inappropriate hardware system or with an operating system not approved by the Licensor.

c) Maintenance services which become necessary due to the use of the Software on another hardware system or with another operating system.
d) Maintenance services after interference of the customer with the program code of the Software.

e) Maintenance services with respect to the interoperability of the Software which is the subject matter of the contract with other computer programs which are not the subject matter of the maintenance contract.

f) The remedying of faults and damage caused by incorrect use by the User, the influence of third parties or force majeure events.

g) The remedying of faults and damage caused by environmental conditions at the setup location, by defects in or absence of the power supply, faulty hardware, operating systems or other influences not attributable to the Licensor.

2. Payment

(1) If the User has acquired the Software for a limited period of time, then the payment for the maintenance has already been effected in full with the payment of the licence fee.

(2) In the event of a right of use for an unlimited period of time, the first twelve months of maintenance are included in the licence fee. In the following period, the annual maintenance fee can be found in the enclosed price table. The Licensor is entitled to adjust the maintenance fee on an annual basis in accordance with the general trend of prices. If the increase in the maintenance fee amounts to more than 5%, the customer may cancel the contractual relationship.

3. Duration of the Contract

In the case of a time-limited right of use, maintenance contract ends with the expiration of the right of use of the Software.

In the case of a time-unlimited right of use:

the maintenance contract is extended after the first twelve months by a further twelve months respectively, unless the User opposes this in writing to the Licensor within a period of 3 months prior to the expiration.

or

the User may demand, after the first twelve months, a continuation of the maintenance contract by a further 12 months respectively up to the date of the expiration of the contract. The demand must be made in writing.

4. Cooperation Obligations

(1) In the transcription, containment, determination and reporting of defects, the customer must follow the instructions issued by the Licensor.
(2) The customer must specify its defect reports and questions as accurately as possible. In doing so, he must also make use of competent employees.

(3) During the necessary test runs, the customer is personally present or seconds competent employees for this purpose, who are authorized to pronounce and decide on defects, functional expansions, functional cutbacks and modifications to the program structure. If necessary, other work involving the computer system must be discontinued during the time of the maintenance work.

(4) The customer grants the Licensor access to the Software via telecommunications. The connections necessary for this are established by the customer according to the instructions of the Licensor.

5. Liability

(1) The Licensor is liable only for deliberate malfeasance and gross negligence and also that of his legal representatives and managerial staff. For the fault of miscellaneous vicarious agents, the liability is limited to five times the annual maintenance fee as well as to such damage the arising of which is typically to be expected in the context of software maintenance.

(2) The liability for data loss is limited to the typical data retrieval expenditure which would have come about in the regular preparation of backup copies in accordance with the risks.

§ 3 MISCELLANEOUS AGREEMENTS

1. Conflicts with Other Terms of Business

Insofar as the User also uses General Terms of Business, the contract comes about even without express agreement about the inclusion of General Terms of Business. Insofar as the different General Terms of Business coincide with respect to their content, they are considered to be agreed. The regulations of the anticipated law replace any contradictory individual regulations. This also applies to the case in which the Conditions of Business of the User contain regulations which are not contained in the framework of these Conditions of Business. If the existing Conditions of Business contain regulations not contained in the Conditions of Business of the User, then the existing Conditions of Business apply.

2. Written Form

All agreements which contain a modification, addition or substantiation of these contractual conditions, as well as specific guarantees and stipulations, must be set down in writing. If they are declared by representatives or vicarious agents of the Licensor, they are only binding if the Licensor has granted his written consent to them.
3. Notice and Cognizance Confirmation

The User is aware of the use of the existing General Conditions of Business on the part of the Licensor. He has had the opportunity to take note of their content in a reasonable manner.

4. Election of Jurisdiction

In relation to all of the legal relations arising from this contractual relationship, the parties agree to apply the law of the Federal Republic of Germany, with the exception of the United Nations Convention on Contracts for the International Sale of Goods.

5. Place of Jurisdiction

For all disputes arising in the context of the execution of this contractual relationship, Dresden is agreed to be the place of jurisdiction.

6. Severability Clause

Should one or more of the provisions of this contract be ineffective or void, then the effectiveness of the remaining provisions remains unaffected. The parties undertake to replace the ineffective or void clauses with legally effective ones which are as equivalent as possible to the originally intended economic result. The same applies if the contract should contain a missing provision which requires addition.
Index

- 1 -

1D-streamline  264, 307, 326

- 3 -

3D Model  48, 243, 245, 252, 256
3D view  256
3D-model  194
3D-View  427

- A -

Acoustic benefit  544
Administrator  20
ALT  594
alternative speed  391
angle of flow  327, 329
ANS  169
Ans  132, 163, 166, 169, 173, 177, 189
ANSYS Workbench  191
approximation  92, 605
Approximation functions  215
Area circles  398
Area progression  398
Assumptions  267, 311, 329
asymmetric  588, 598
AUNGIER  462
Auto fit view  88
AutoCAD  132, 135
AutoGrid  169, 174
Automated component design  82
Automatic  65, 620
Automatic design  83
Automatic update  86
Axial extension  401, 405
axial impeller  311, 481
Axial position  581

- B -

Background  245
BACKSTROEM  464
Balje  395
Barske  265
Basic values  309, 327
Batch  35
Batch mode  191
Batch mode template  127
bend  598
Beta progression  467
Bezier  92, 467, 512, 588, 593
Beziers  88
Beziers mode  398, 401, 405, 413
Beziers polynom  488
Blade  520, 542, 544
Blade angle  455, 457
blade angle distribution  478
Blade angles  429, 488
Blade blockage  455, 457
Blade lean angle  488
Blade lines  429
blade loading  479
blade number  329
Blade properties  429
Blade root fillet  557
Blade shape  429
Blade thickness  429, 504
Blade thickness leading edge  215
BladeGen  47, 132
Blades  252
BREP  48
Brumfield  291

- C -

CAD  10, 120
CAE  120
Calculate  275, 318, 335, 429
Calculation  65
camber angle  540
Casing  103, 554
Catia  132, 143
CFD  10, 120, 177, 633
CFT 100
CFTurbo 10
CFTurbo2ICEM 177
Characteristic numbers 267, 311, 329
Check 200, 234
checksum 22
Chord length 535, 538, 540, 542
Circle 409, 588, 612
circular arc 605
cloud 32
Color 252
color 20, 22
Compare 194
Compensation 599, 604
Compressor 10
conformal mapping 467, 475
Constant 504
Contact addresses 643
continuity equation 275, 318, 335
Contour 252, 412
cotra rotating 391
Coordinate system 245, 467
Coordinates 88
Copy 88
copy to clipboard 22
Cordier 372, 385, 394
Coupled 401, 405
Coupled linear 512
Creo Parametric 132
Cross section 401, 405
Cross sections 264, 307, 326, 588
curvatures 398
cut-water 599, 604, 620
cut-water diameter ratio 215
cut-water width ratio 215
- D -
data points 555
Deactivate 65
Deceleration ratio 429
Decreased output 429, 463
default 210, 212
Degree of reaction 395
Density 309
Design point 103, 309, 327
Design report 127
Design rule 599, 602
DesignModeler 132
Deviation angle 429, 457
Deviation flow - blade 429
diagram 211
Diameter coefficient 215, 267, 311
diameter ratio 267, 329
Diameter ratioVolumetric e 311
Diameter.cftdi 275, 393
Dimensions 275, 318, 329, 335, 365, 372, 377, 385
direction of rotation 581
Display options 88
distance tolerance 256, 555
DoE 191
double volute 606
double-click 200
download 234
- E -
Edge 520
Edge position 520
edit 93
Efficiency 267, 311, 368, 381
Hydraulic 267
Internal 267
Mechanical 267
Overall 267
Side friction 267
tip clearance 267
Volumetric 267
Efficiency total-to-static 329
Efficiency total-to-total 329
Ellipse 512
emergency 74
empirical 93
End cross section 599, 612
End shape 612
error 213
Errors 256
Euler's Equation of Turbomachinery 275, 318, 335
Exact 446
Exit diameter 398
Exit width 398
Expiration 200
- F -

file 33
File location 215, 275, 318, 393
find 33
Finishing 198, 557
Flow angle 63, 309, 429
Flow angle inflow 215
Flow angle outflow 215
Flow angles 267, 311
Flow direction 63
Flow rate 309
Fluid 103
found 33
Francis 365, 366, 368, 372
Freeform 488
Frontal view 467, 504, 512
Full impeller 97, 309, 327
Full volute 97
Function 93, 215
Functions.cftfu 215

- G -

General geometry 127
Global setup 103
Graphic 88
Grid 398
GridPro 169

- H -

handling 88
Head 309
Help 233
Hexpress 169
Hub 393, 398, 401, 405
hub diameter 275, 318, 335, 393
Hydraulic efficiency 215

- I -

ICEM 177
ICEM-CFD 169
IGES 48, 127, 243, 252
IGG 169
Impeller 10, 99, 212, 213, 581
Impeller diameter 275, 318, 394, 395
Impeller efficiency 311
Impeller Options 210
Import 45, 48, 91, 252
Incidence angle 429, 455
Inclination angle 401, 405
Inclination angle hub 215
Inclination angle shroud 215
Inclination angle trailing edge 215
Inducer 291
ineffective blade angle 503
Inflow 103
Inflow swirl 309
information 32
initial 212
Initial design 88
Inlet 412, 554
Inlet definition 581
inlet diameter 335
Inlet triangle 455
inner 606
input 35, 88
install 19
installation 15, 19
Intake coefficient 215, 267
Interface 120
Interface definition 63
Interfaces 41
internal 588, 598
Internal efficiency 311
Inventor 132, 160
inverse design 479
Isentropic Mach number 484

- K -

Kaplan 377, 379, 381, 385
Index

- L -

Language 200
Leading edge 398, 401, 405, 413, 455, 512
Length unit for Export 634
License 22, 32, 33, 200
License agreement 643
License key 20
Licensing 10, 20
Line Segments 594
Line width 245
Linear 504
Linked 446
Load from impeller 581
local 22
Log file 70
Logging 70

- M -

machine ID 20, 22
Main dimensions 264, 265, 275, 307, 318, 326, 329
main window 80
Manual 620
Manual dimensioning 265
Material 252
material density 390
Material domain 557
Max. curvature 401, 405
Mean line 467
Mechanical Efficiency 215, 311, 329
meridional velocity 415
Meridian 48
meridinal deceleration 329
Meridional 467
meridional boundaries 504, 512
Meridional contour 398
Meridional deceleration 215, 267, 311
Meridional extension 488
Meridional flow coefficient 291
Meridional preview 372, 385
Meshing 169, 173
Minimal relative velocity 267
Mixed-flow impeller 311
mixed-flow rotor 329, 335
Model settings 634
Model state 252
Model-finishing 557
Model-settings 555
modules 22
Mouse 243
multi stage 391
multi-stage 99

- N -

NACA 542
Named Selections 173
Navigation 82
neck 598
network 22
New design 97
NPSH 267
number of blades 215, 568
Number of revolutions 309
Numeca 174
NX 132

- O -

Obstruction 429
OMNIS 169, 174
Open 100
Optimal 429
Optimization 35
Optimization 191
Options 200
Other 200
outer 606
Outflow coefficient 461, 462, 464
Outlet 412, 554
Outlet triangle 455, 457
Outlet width 275, 318
Outlet width ratio 267, 311
Output 35
Overall efficiency 311

- P -

Parallel to z 401, 405, 413
Parameter 35, 93, 191, 215, 265
<table>
<thead>
<tr>
<th>Parameters</th>
<th>120, 368, 381</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parametric model</td>
<td>86</td>
</tr>
<tr>
<td>Parasolid</td>
<td>48, 127, 243, 245, 252</td>
</tr>
<tr>
<td>permission</td>
<td>33</td>
</tr>
<tr>
<td>permissions</td>
<td>33</td>
</tr>
<tr>
<td>PFLEIDERER</td>
<td>429, 463, 599, 602</td>
</tr>
<tr>
<td>Physical variable</td>
<td>215</td>
</tr>
<tr>
<td>pitch</td>
<td>535, 538, 540, 542</td>
</tr>
<tr>
<td>point based export</td>
<td>555</td>
</tr>
<tr>
<td>Points</td>
<td>215, 252</td>
</tr>
<tr>
<td>Pointwise</td>
<td>169</td>
</tr>
<tr>
<td>Polyline</td>
<td>91, 92</td>
</tr>
<tr>
<td>Position</td>
<td>620</td>
</tr>
<tr>
<td>potential flow</td>
<td>415</td>
</tr>
<tr>
<td>Power loss</td>
<td>267, 311</td>
</tr>
<tr>
<td>Power output</td>
<td>309</td>
</tr>
<tr>
<td>Power partitioning</td>
<td>265</td>
</tr>
<tr>
<td>Preferences</td>
<td>200, 210</td>
</tr>
<tr>
<td>Pressure coefficient</td>
<td>267, 311</td>
</tr>
<tr>
<td>Pressure difference</td>
<td>309</td>
</tr>
<tr>
<td>Pressure side</td>
<td>455</td>
</tr>
<tr>
<td>Primary side</td>
<td>63</td>
</tr>
<tr>
<td>Print</td>
<td>88, 245</td>
</tr>
<tr>
<td>prism_params</td>
<td>177</td>
</tr>
<tr>
<td>Pro/ENGINEER</td>
<td>144</td>
</tr>
<tr>
<td>problem</td>
<td>33</td>
</tr>
<tr>
<td>problems</td>
<td>33, 256, 555</td>
</tr>
<tr>
<td>Profile</td>
<td>504, 542</td>
</tr>
<tr>
<td>progression</td>
<td>91, 211</td>
</tr>
<tr>
<td>Progressions diagrams</td>
<td>398</td>
</tr>
<tr>
<td>Project information</td>
<td>101</td>
</tr>
<tr>
<td>Project structure</td>
<td>41</td>
</tr>
<tr>
<td>Project types</td>
<td>41</td>
</tr>
<tr>
<td>Projection</td>
<td>554</td>
</tr>
<tr>
<td>Pump</td>
<td>10</td>
</tr>
</tbody>
</table>

**Radial** 612

- Radial 2D 542
- Radial blade 449
- Radial blade fibre 449
- Radial blade section 449
- Radial diffusor 581
- Radial element blade 449, 481, 488
- radial equilibrium 528, 530
- Radial impeller 311
- Radial rotor 329, 335
- Radius 409, 594
- Rake 467
- RDP 20
- recovery 74
- Rectangle 588, 593, 612
- Redesign 48
- Redo 82
- Reference 194
- References 637
- Register 20
- Remote 20
- Remove design steps 86
- request 22
- Required driving power 267, 311
- Resolution 245
- Reverse engineering 48
- rights 33
- rotational speed 327
- rotor power 327
- Rotor-Stator-Interface 63
- RSI 555, 633
- RSI connection 555
- RTZT 47
- Ruled surface blade 446
- Runner 365, 377

**S**

- Save 88, 100, 245
- Secondary side 63
- send E-mail 22
- server 33
- session code 22
- Settings 555
- Setup 19, 366, 379
- Shaded 252
- shaft 393
- shaft diameter 275, 318, 393
- Shaft/ hub 335
- Sharp 620
- Shroud 398, 401, 405
- Shroud angle 267
- shroud diameter 335
- SI 204
Index

Side friction efficiency  215
silent  19
Simcenter STAR-CCM+  169
Simerics  169
SimericsMP  169
SimericsMP+  169
Simple  512
simple blade shape  482
Simple mode  398, 409
Single blade  252
single-flow  327
Single-intake  309
single-stage  309, 327
Slip  429, 457
Slip velocity  461, 462, 464
Solid  252, 256, 419
solid density  390
Solids  256
SOLIDWORKS  132
solidity  535, 538, 540, 542
SpaceClaim  132, 163, 166
Specific diameter  394, 395
Specific energy  309
Specific speed  309, 327, 394, 395
specific work  327
Speed coefficient  309
Spline  612
Splitter  265, 446
splitter blades  568
Squirrel cage  265
Stack  544
stage  99
Stagger angle  535, 538, 540, 542
Stagnation point  455
Standard specifications  275
Stanitz-RADIUS  493
Start  78, 120
Start angle  612
start date  22
Static moment  398, 401, 405
Status bar  88
STEP  48, 127, 243, 252
Step by step  97
STEPANOFF  599, 602
Stepanoff constant  215
STL  127, 243, 252
Straight  401, 405, 413
Straight line  409
stream function  415
Stress.cftst  275, 318, 393
STRG  594
Strictly external  588
Suction diameter  275, 318
Suction side  455
Suction specific speed  267, 291
Surfaces  256
Sweep  544
Sweep correction  535, 544
Swirl  455
swirl number  327
Symbols  641
Symmetric  588

- T -

Tangential  405, 412, 467, 612
Test  215
Thickness  504, 604
Through-flow area  633
tin  177
tinXML  177
Tip  554
tip clearance  568
Tip clearance efficiency  215
Tip projection to casing  554
torque  393
torsional stress  393
Trailing edge  401, 405, 413, 512
Transmission of energy  457
Transparency  252
Trapezoid  588, 593
Trimming  557
Turbine  10, 365, 366, 368, 372, 377, 379, 381, 385
TurboGrid  169, 189
Turbomachinery CFD  169
Type number  309, 327

- U -

Undo  74, 82
Uniform  512
uninstall  19
uninstallation 19
Units 204
unshrouded 265, 554, 568
unwinded length 512
Update 233, 234
Update warnings 82
Updates 200
US 204
user 33
User defined 504

- V -

values 88
Velocity components 429
Velocity triangle 335, 429, 455, 457
Velocity triangles 275, 318, 372, 385
Ventilator 10
Version 100, 120
View 245
Visible 252
Vista TF 169
VNC 20
Volumetric efficiency 215, 311, 581
Volute geometry 599

- W -

warning 213, 504, 512
Wastewater 265
Website 10
Width lines 398
Width number 215
WIESNER 429, 461
Wireframe 252
Work coefficient 215
Workbench 173
Wrap angle 215, 488, 599

- Z -

Zoom 88, 245
ZW3D 132