



CFturbo 10

User manual for CFturbo 10 software



© CFturbo GmbH

CFturbo 10

Introduction

*This manual describes the usage of the software CFturbo 10
and corresponds to the online help with regards to content.*

© CFturbo GmbH, 2020

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	8
Part II What's new in CFturbo 2020 R1	11
Part III General	13
1 Installation	14
2 Licensing	18
Node-locked license setup	21
Floating license setup	24
License server setup.....	25
Client setup.....	29
Show license information	30
Troubleshooting	31
3 Batch mode	32
Example	36
Exit Codes	38
4 Project structure	39
Add component	41
Geometric coupling	42
Automatic calculations	44
5 Windows Explorer integration	46
6 Troubleshooting	48
Error reporting	49
Emergency recovery	52
Known problems	53
Part IV Getting started	55
1 Start	56
2 Opened project	59
3 Component design process	61
4 Activate/ Rename/ Delete components	63
5 Remove design steps	65
6 Handling	66
General handling	67
Graphical dialogs	67
Progression dialog	70
Edit fields with empirical functions	71
Part V Menu	73
1 File	75
Create new design	75
Stage designer.....	77

Open/ Save design	78
Import external geometry	79
Native format.....	81
RTZT format	82
2 PROJECT	84
General	84
Project information.....	85
Global setup.....	86
Performance prediction.....	92
Euler based.....	96
Casey/Robinson.....	101
Undo.....	102
Additional	103
Export.....	103
Basic	111
CAD, CAM.....	114
CFD.....	117
Specifics	121
STL	122
Tetrahedral volume mesh.....	123
AutoCAD (Autodesk).....	124
BladeGen (ANSYS)	131
CATIA (Dassault Systèmes).....	133
Creo Parametric (PTC).....	134
Inventor (Autodesk).....	150
SpaceClaim (ANSYS)	153
Known issues in Spaceclaim.....	156
AutoGrid (NUMECA).....	159
ICEM-CFD (ANSYS)	162
TurboGrid (ANSYS).....	163
SimericsMP/ SimericsMP+ (Simerics).....	165
TCFD (CFD Support)	175
Data export limitations.....	176
Batch mode/ Optimization	176
Reference components.....	181
Model finishing.....	184
3 SETTINGS	185
Licensing	185
Preferences	185
General.....	186
Units.....	190
General	191
Specific speed	192
Other.....	195
Impeller/ Stator.....	196
Approximation functions	198
Fluids	201
CoolProp library.....	206
Gas mixtures.....	208
Profiles	210
4 HELP	215
Check for Updates	216
5 Component	217

6 3D View	218
7 Report View	219

Part VI Views 220

1 Meridian	222
2 3D Model	225
Model display (top)	227
Model tree (left)	234
Problems when generating the 3D model	238
3 Report	240

Part VII Impeller 242

1 Main dimensions	244
Radial/Mixed-flow Pump / Ventilator	246
Setup.....	247
Parameters	249
Dimensions.....	256
Axial Pump / Ventilator	263
Setup.....	265
Parameters Pump.....	267
Inducer.....	272
Parameters Ventilator.....	274
Dimensions.....	280
Centrifugal Compressor	286
Setup.....	288
Parameters	290
Dimensions.....	297
Radial-inflow Turbine	304
Setup.....	306
Parameters	308
Dimensions.....	313
Axial Turbine	321
Setup.....	323
Parameters	325
Dimensions.....	329
Material density	335
Multi stage	336
Shaft/Hub	337
2 Meridional contour	338
Primary flow path	341
Hub, Shroud.....	343
Bezier	345
Converting Polyline / Bezier.....	349
Circular Arc + Straight line	351
Contour	353
Leading, Trailing edge.....	354
Meridional flow calculation	356
Hub/Shroud solids	360
Bezier curve	365
Line segment curve.....	367
Secondary flow path	367

Additional views	369
3 Mean line design	371
Blade properties	371
Blade setup.....	375
Ruled Surface blade.....	386
Radial element blade	389
Blade angles	389
Inlet triangle.....	393
Outlet triangle	395
Slip coefficient by GÜLICH/ WIESNER.....	399
Slip coefficient by AUNGIER/ WIESNER.....	400
Slip coefficient by PFLEIDERER.....	401
Slip coefficient by VON BACKSTROEM.....	402
Specific slip coefficient definitions	403
B2 calculation details	403
Blade mean lines	405
Additional views.....	412
Informational values	414
t, m conformal mapping.....	416
Blade lean angle.....	417
Blade surface values.....	419
Blade-to-blade flow 1D.....	424
Blade-to-blade flow 2D	426
Sine rule.....	429
Blade angles.....	431
Design mode.....	432
Blade profiles	438
Additional views.....	444
Converting Polyline / Bezier.....	446
Blade edges	447
Additional views.....	454
Edge position	455
4 Airfoil/ Hydrofoil design	456
Blade properties	457
Cu-specification.....	460
Radial equilibrium.....	463
Blade profiles.....	463
Kinematics.....	466
Blade element momentum method.....	470
Lieblein method.....	471
Blade profile	473
Blade sweep	475
5 CFD setup	478
Extension	479
Segment	482
Blade projection	484
RSI connection	485
6 Model settings	486
7 Model finishing	487

Part VIII Stator

494

1 Main dimensions	496
--------------------------------	------------

Extent	499
Inlet	501
Outlet	502
2 Meridional contour	502
3 Blade properties	503
Number of blades	505
4 Blade mean lines	507
5 Blade profiles	510
6 Blade edges	510
7 CFD setup	510
8 Model settings	511
9 Model finishing	511

Part IX Volute 512

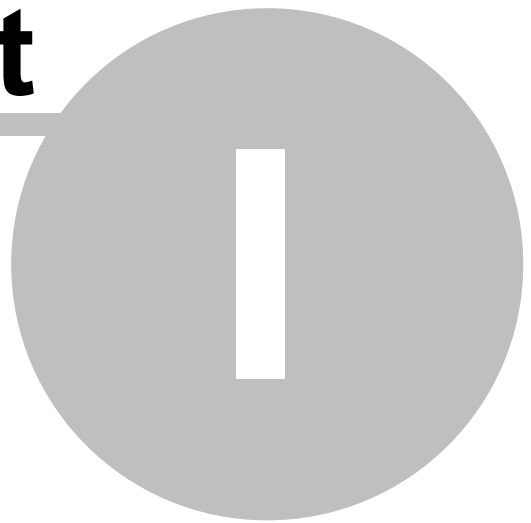
1 Setup + Inlet	513
Setup	514
Inlet details	519
2 Cross Section	520
Bezier cross section	525
Line Segments cross section	526
Radius based cross section	529
Internal cross sections	530
3 Spiral development areas	531
Design rule	534
Cut-water compensation	537
Additional views	538
Double Volute	540
4 Diffuser	543
Additional views	548
5 Cut-water	550
Simple	553
Fillet	556
Sharp	559
Additional views	560
6 CFD setup	562
7 Model settings	563

Part X Appendix 565

1 References	566
2 Symbols	569
3 Contact addresses	572
4 License agreement	572

Index 581

Part



1 CFturbo

CFturbo

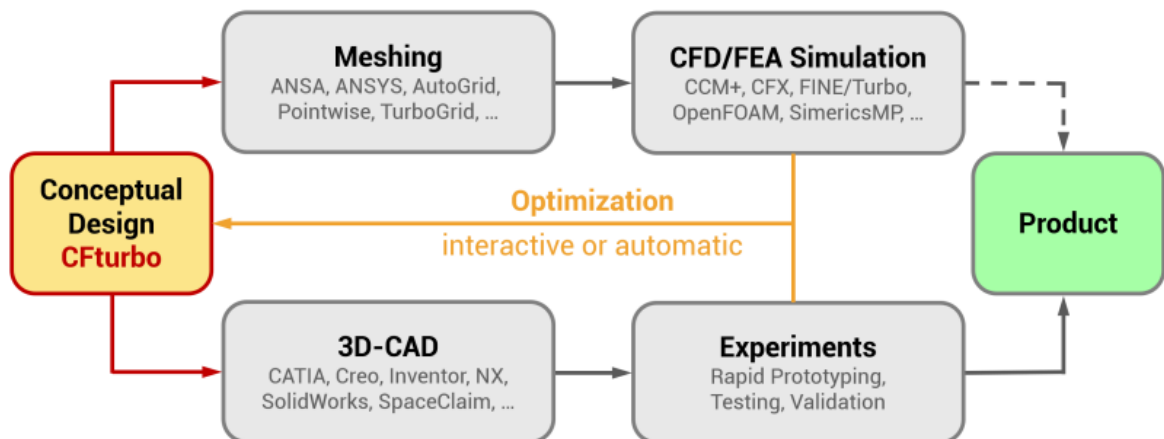
CFturbo is made to interactively design radial, mixed-flow and axial turbomachinery: pumps, ventilators, compressors, turbines. The software is easy to use and does enable quick generation and variation of impeller, stator and volute geometries. Several models can be displayed, compared and modified 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 and Windows 10.

Hardware requirement is a modern office PC.

Integration of geometry data into the CAE environment is easily possible by direct interfaces to various CAD- and CFD-systems.



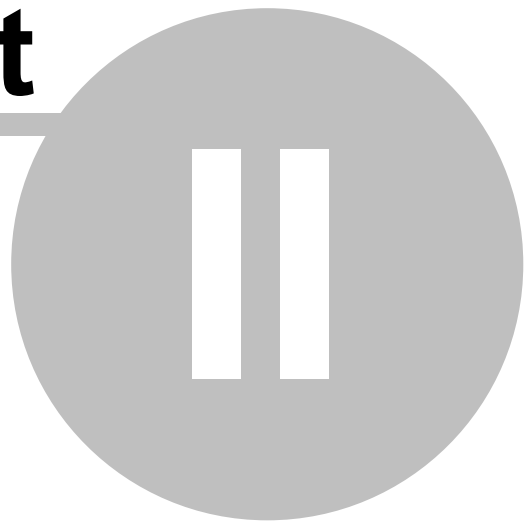
Please read the [License agreement](#)^[572] before using the program.

Information about activating license you can read in chapter [Licensing](#)^[18].

Contact persons you can find under [Contact addresses](#)⁵⁷², actual information on the [CFturbo website](#).

Copyright © [CFturbo GmbH](#)

Part

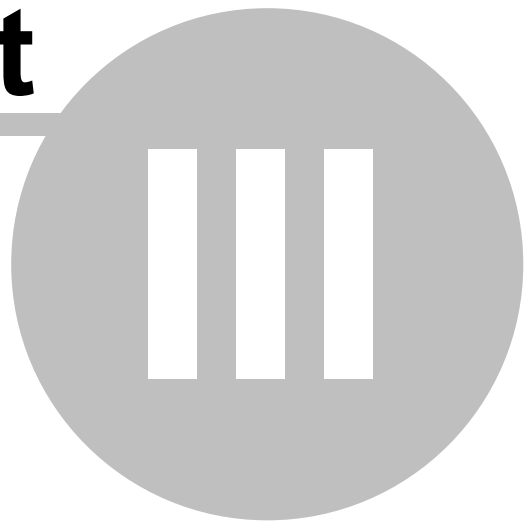


2 What's new in CFturbo 2020 R1

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

<https://cfturbo.com/software/release-notes>

Part



3 General

This chapter contains some general program information about

- [Installation](#) ^[14]
- [Licensing](#) ^[18]
- [Batch mode](#) ^[32]
- [Project structure](#) ^[39]
- [Handling](#) ^[66]
- [Windows Explorer integration](#) ^[46]
- [Troubleshooting](#) ^[48]

3.1 Installation

PLEASE NOTE:

- Administrator rights are required for the installation.
- CFturbo runs on Windows 8.1 and 10.
- 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>

Username

Password

[Forgot your password?](#)

Login

Download is possible for authorized users only.

Registered users can change their password.

Registration is required to get download access.

Here for the first time?

Register here.

CFturbo

↓ [CFturbo2020.1.0.0_setup_win64.exe](#) (86MB)

CFturbo Setup Version 2020.1.0 (04/2020)

CFturbo client setup (64 Bit)

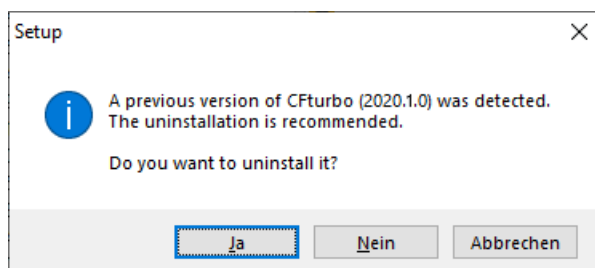
CFturbo Lizenzserver

↓ [CFturboLicenseServer2020.1.0_setup_win64.exe](#) (5MB)

CFturbo License Server Setup Version 2020.1.0 (04/2020)

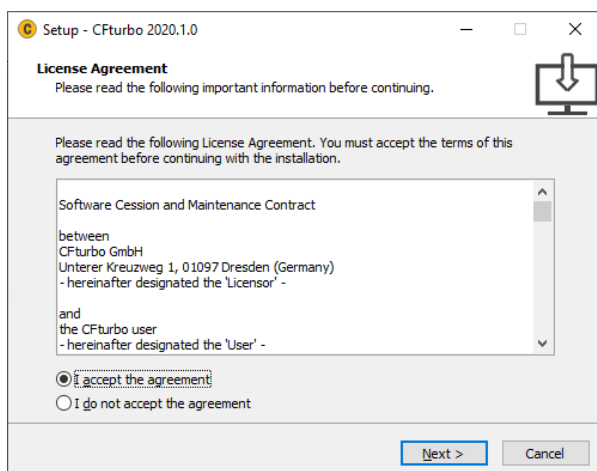
CFturbo License server setup (for customers only)

(2) Installation

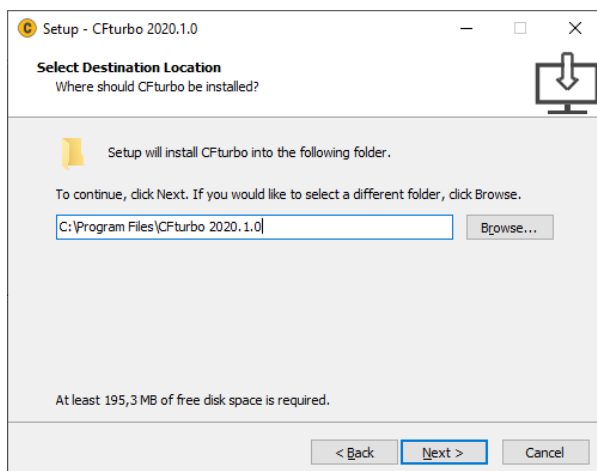


Start the installation program
CFturbo2020.1.*_setup_win64.exe.

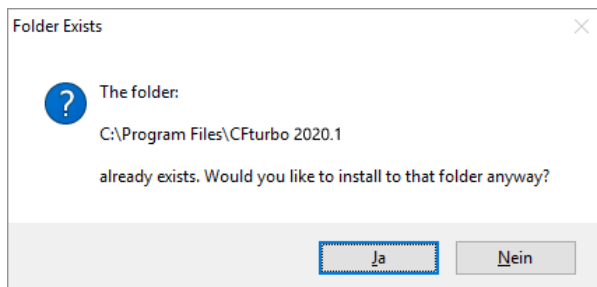
Uninstall existing version optionally.



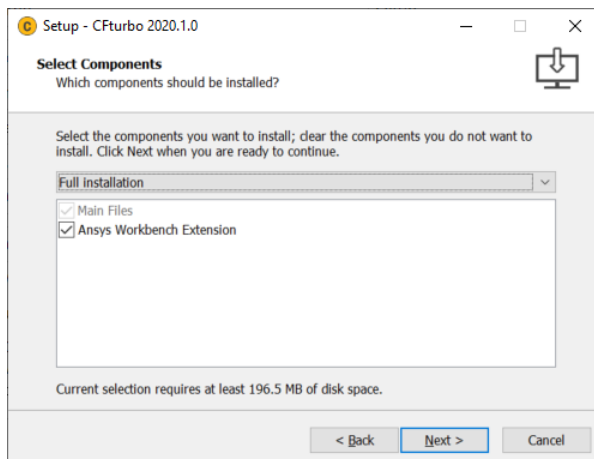
Accept the license agreement.



Accept or select the destination directory.



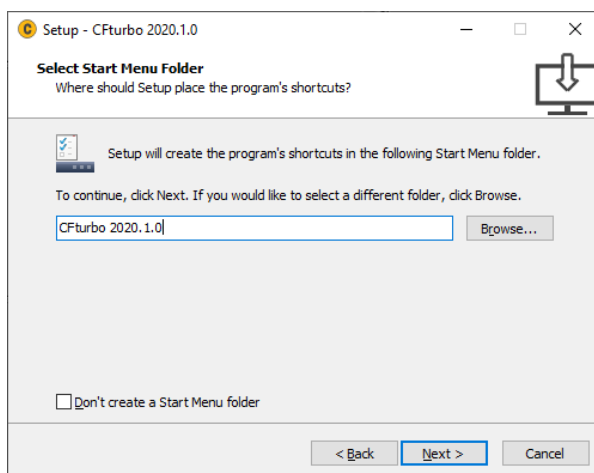
If the directory already exists, the existing installation can be overwritten .



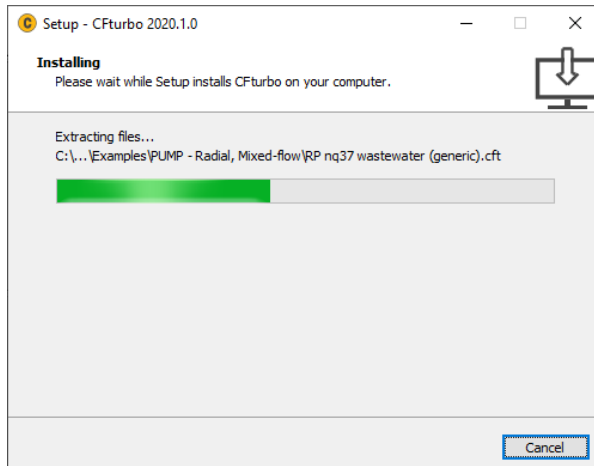
CFturbo Ansys Workbench Extension can be installed optionally.

The extension is installed into all existing Ansys installations from version 19.0 to 2020 R1. Of course, the extension can be installed later manually.

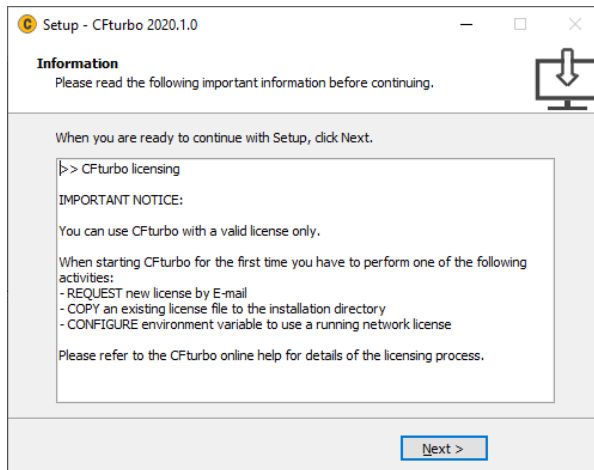
More information about the CFturbo Ansys Workbench Extension is available here: <https://cfturbo.com/software/interfaces-workflows/extension-for-ansys-workbench>



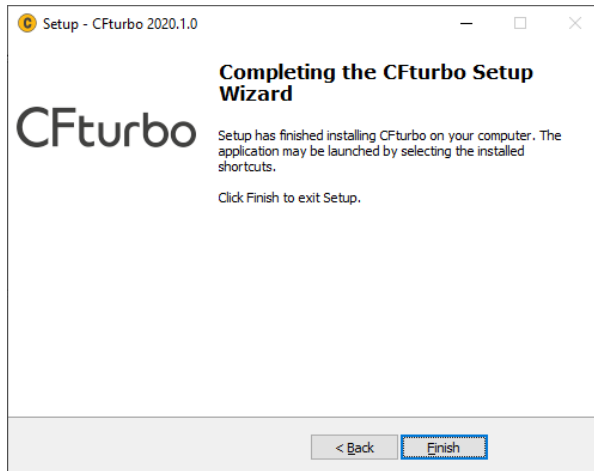
Accept or modify the start menu name.



The installation takes about 20 seconds normally.



Short licensing information (see [licensing](#)^[18]).



Setup is completed.

(3) Licensing

After successful installation of the software CFturbo the [licensing](#)^[18] must be performed.

3.2 Licensing

? SETTINGS | General | Licensing

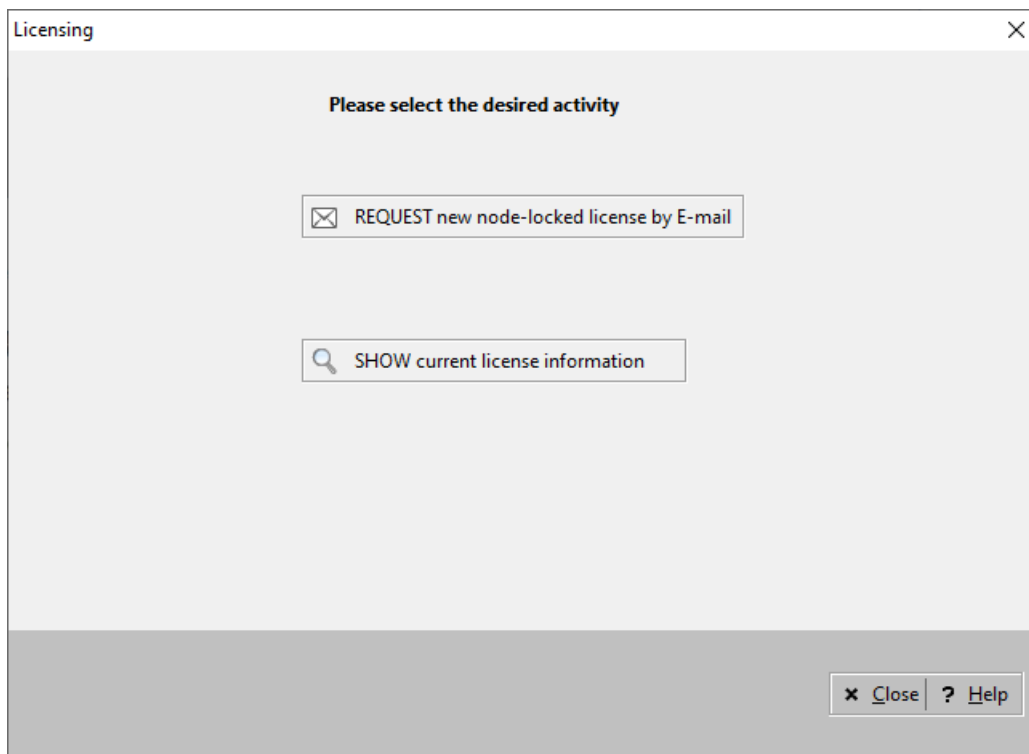


CFturbo can be used without a valid license in viewer mode. This mode allows to open project files independent of the included components for reading access. No changes can be done in viewer mode.

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 stators without blades.

Menu item **Licensing** enables license handling.



[REQUEST](#)^[21] new license by e-mail

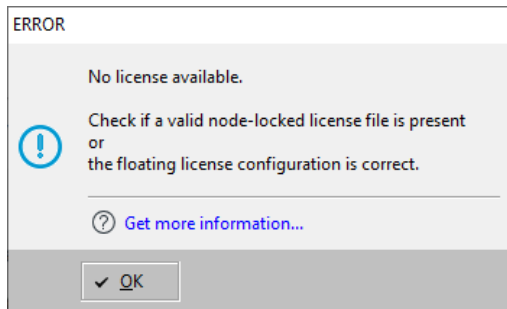
[SHOW](#)^[30] current license information

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](#)^[186].

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 going to be modified:

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

or

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

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!

1. Node-locked License

Step	
1.	Start CFturbo - you see the " License " dialog ^[18] (or open menu SETTINGS General Licensing).
2.	Request ^[21] node-locked license and send license request to sales@cfturbo.com
3.	Save license file (<filename>.lic) received from CFturbo sales team to CFturbo installation directory (C:\Program Files\CFturbo *)
4.	Show ^[30] license information to check modules and dates

2. Floating License

(NOT available for trial license)

To use CFturbo with a floating license, the CFturbo license server software must be setup (includes requesting and installing a floating license). For details see [Floating license setup](#)^[24].

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

Step	
1.	Floating license setup ^[24]
2.	Start CFturbo and open menu SETTINGS General Licensing
3.	Show ^[30] license information to check modules and dates

3.2.1 Node-locked license setup

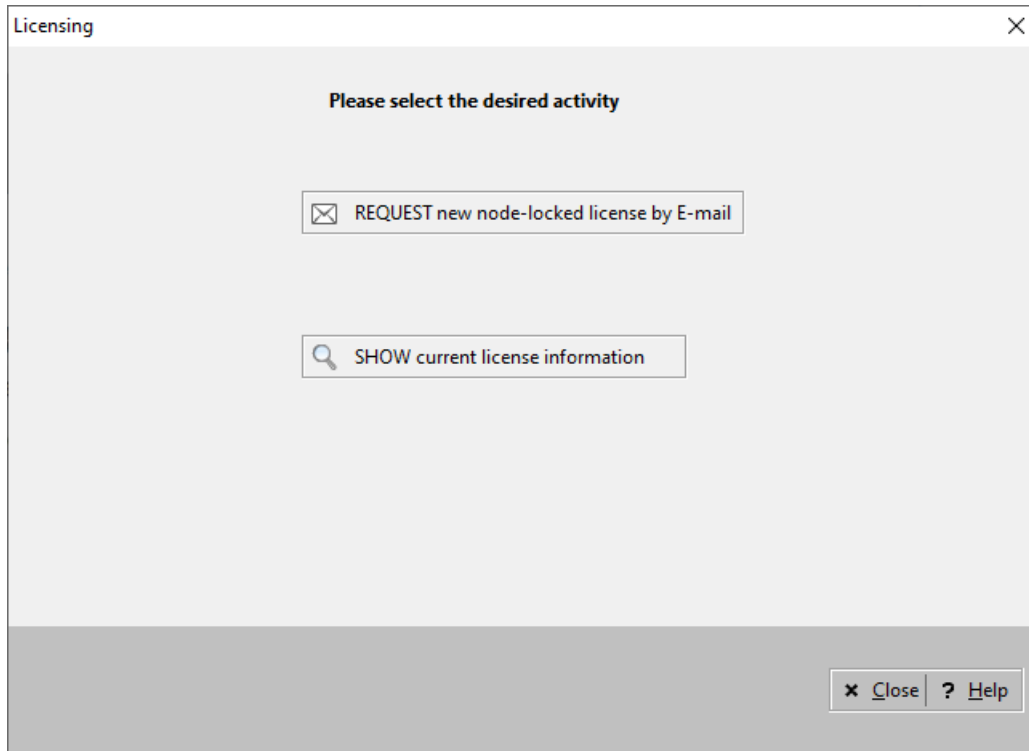
For using CFturbo with a node-locked license 2 steps have to be performed:

- Requesting a license using the CFturbo license dialog
- Storing the received license file in the CFturbo installation directory

Note: If CFturbo is [configured for using a floating license](#)^[24], 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**.

Licensing

Back Request Node-locked License

Modules

- ☒ Radial Pump Impeller
- ☐ Radial Ventilator Impeller
- ☐ Radial Compressor Impeller
- ☐ Radial Turbine Rotor
- ☐ Axial Pump Impeller
- ☐ Axial Ventilator Impeller
- ☐ Axial Turbine Rotor
- ☐ Stator
- ☐ Volute

Company CFturbo GmbH

Start date 11.03.2020

Checksum 977401315

Machine ID CFIDLOC=36cfc53aca79598f225818131bb05615

License request

Create E-Mail... Copy to Clipboard

All fields must be completed for the license request.

Close Help

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](#)^[30].

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 has to be located in the same local area network (LAN) of all CFturbo clients. Usage of the floating licenses in a wide area network (WAN) is not allowed.
- 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](#)^[25]
2. [Requesting a license using the Request Generator](#)^[27]

3. [Storing the received license file in the CFturbo license server installation directory](#)^[25]
4. [Configuring the clients for accessing the floating license](#)^[29]

3.2.2.1 License server setup

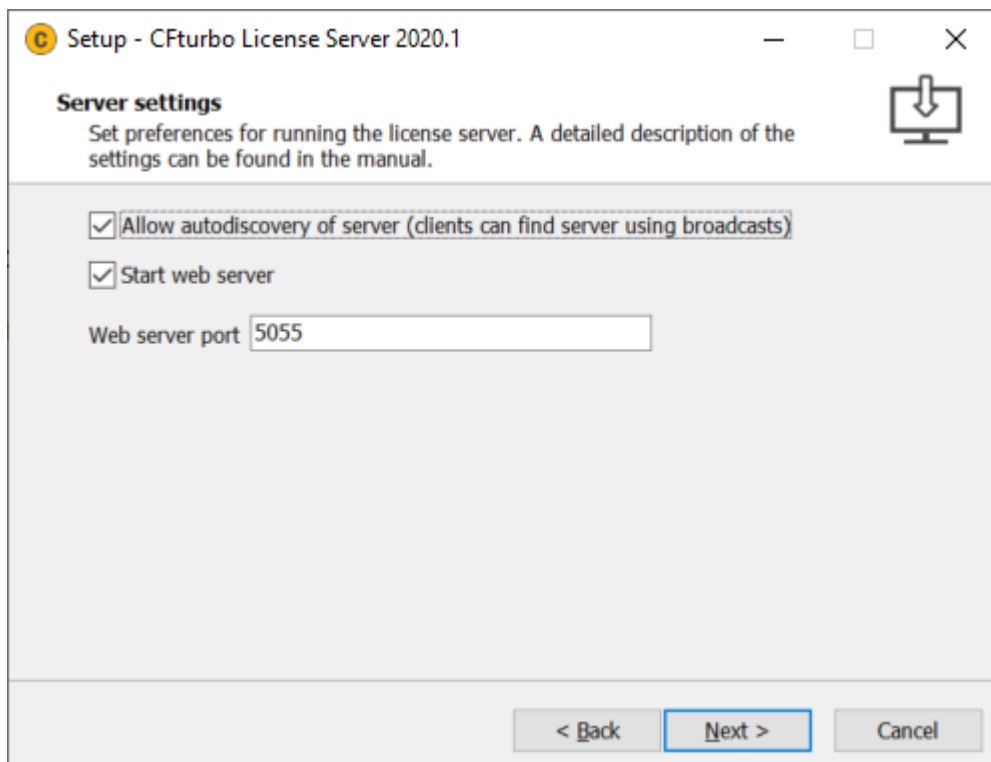
Installing the license server

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.

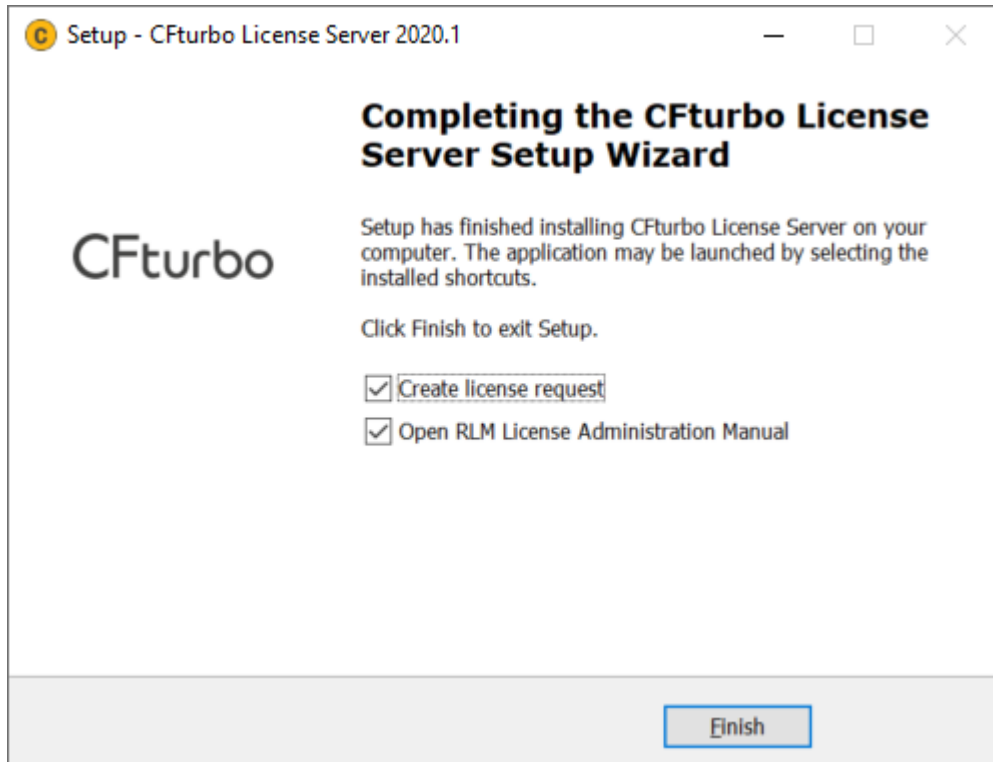
After starting the setup, selecting installation directory and start menu settings, the server parameters can be configured:



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

Requesting a floating license

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

The screenshot shows a window titled 'Licensing' with a close button (X) in the top right corner. Inside the window is a section titled 'Request Floating License'. On the left, under the heading 'Modules', there is a list of modules with checkboxes: ☒ Radial Pump Impeller, ☐ Radial Ventilator Impeller, ☐ Radial Compressor Impeller, ☐ Radial Turbine Rotor, ☐ Axial Pump Impeller, ☐ Axial Ventilator Impeller, ☐ Axial Turbine Rotor, ☒ Stator, and ☒ Volute. The 'Volute' option is highlighted with a blue background. To the right of the modules list, there are several input fields: 'Company' with the text 'CFturbo GmbH', 'Start date' with a calendar icon and the date '11.03.2020', 'Checksum' with the value '977401315', 'Machine ID' with the value 'CFIDNET2=812b42801c644277e2a9466cae64f187', and 'Concurrent users' with a spinner box set to '1'. Below these fields, there is a 'License request' section with two buttons: 'Create E-Mail...' (with an envelope icon) and 'Copy to Clipboard' (with a clipboard icon). A message below the buttons states: 'All fields must be completed for the license request.' At the bottom left of the window is the 'CFturbo' logo and the text 'CFturbo 2020.1 Copyright © 2020 CFturbo GmbH'. At the bottom right are two buttons: 'Close' (with an X icon) and 'Help' (with a question mark icon).

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).

Install license file

The license file you receive must be stored in the license server installation directory (e.g. C:\Program Files (x86)\CFturbo 10\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.

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=81f344a7a69115ce4cfd7c30c46efd1f **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](#) ^[29]).

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

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](#).

Setting the 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 (e.g. <http://support.microsoft.com/kb/310519>).

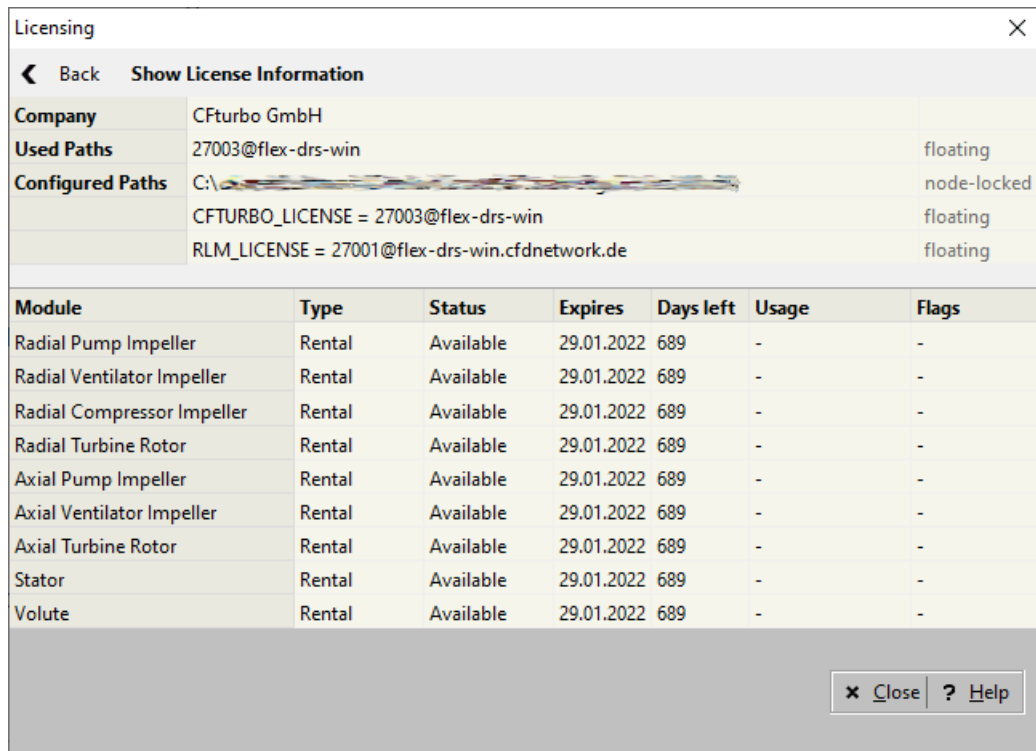
3.2.3 Show license information

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.



No node-locked license file is found in program path, a floating license path is used

3.2.4 Troubleshooting

Error messages

Problem	Message	Reasons
No valid license available yet.		See Steps for licensing ¹⁸⁾

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

CFturbo is installed in: Batch file is:	c:\Program Files\CFturbo 2020.1\ c:\tmp\Example.cft-batch
--	--

```
"c:\Program Files\CFturbo 2020.1\cfturbo.exe" -batch c:\tmp\Example.cft-batch
```

Options:

-batch <batch file>	Enables CFturbo batch mode. <batch file> can either be a CFturbo batch file (*.cft-batch) or a CFturbo project file (*.cft).
-verbose	Display log output on the command line.
-export <interface name>	If CFturbo is started with a CFturbo project file in batch mode, an export interface can be selected like in the batch file.
-log <log file>	Use specified logfile for output

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](#)¹⁷⁶.

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 version="1.0" standalone="yes"?>
<CFturboFile Version="2020.1">
  <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](#)^[256], [blade angles](#)^[389] 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.

- `<BatchAction Name="Export" ExportInterface="STEP" WorkingDir="c:\Examples\Myexports" BaseFileName="Pump1_all" ModelState="Solids only" AllComponents="" />`

The Export-action is used to export the project data utilizing the export interfaces CFturbo supports.

By default the active component (Predefined 3D model export/ Point based export) or geometry elements as configured in the active Model state (3D model export) are exported.

Depending on the export interface a selection of the components to export can either be done using the **ModelState**-attribute (3D model export) or the **ExportComponents**-subelement (Predefined 3D model export/ Point based export). For details about the supported selection options for the specific interface see [Project | Export](#) ¹⁰³.

Attribute	Value	optional	Description
Name	Export	no	Name of action
ExportInterface	e.g. "General"	no	Export interface to use. The following values are valid: <div> <div>AutoCAD</div> <div>BladeGen</div> <div>BREP</div> <div>Catia</div> <div>CfturboExchange</div> <div>CreoParametric</div> <div>DXF</div> <div>General</div> <div>IcemSTEP</div> <div>IGES</div> <div>Inventor</div> <div>MeridianContour</div> <div>NumecaAG</div> <div>Parasolid</div> <div>PerformanceData</div> <div>Pointwise</div> <div>Report</div> <div>SpaceClaim</div> <div>Simerics</div> <div>SolidWorks</div> <div>StarCCM</div> <div>STEP</div> <div>STL</div> <div>TetraVolMesh</div> <div>TurboGrid</div> <div>TurbomachineryCFD</div> </div>

			NumecaIGG NX OpenFOAM	VistaTF
WorkingDir	<existing path>	yes	Folder for exported files	
BaseFileName	<filename>	yes	File name without extension	
ModelState	<existing model state>	yes	Model state to select for export	
AllComponents	empty	yes	Select all components for export, Note: Only components which are supported by the export interface will be exported!	

The **ExportComponents**-subelement is a list of components that should be exported.

- `<BatchAction Name="Save" OutputFile="C:\Examples\Impeller\Pump1_new.cft"/>`

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.

The **OutputFile** attribute specifies the path of the file save destination.

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**.


```
1 <?xml version="1.0" standalone="yes"?>
2 <CFturboFile Version="10.3">
3   <CFturboBatchProject InputFile=".\\pump.cft">
4     <Updates>
5       <CFturboProject Type="Object" Version="10.3">
6         <CFturboDesign_RadialImpeller Type="Object" Name="Radial
          Impeller" Info="CFturbo (2/4/50)">
7           <BladeProperties Type="Object">
8             <nBl Type="Integer" Caption="Number of blades"
              Desc="Number of blades (incl. splitter)"
              OptValueRange="3:8:6">7</nBl>
9           </BladeProperties>
10          </CFturboDesign_RadialImpeller>
11        </CFturboProject>
12      </Updates>
13    <BatchAction Name="Export" ExportInterface="TurboGrid" WorkingDir=
      ".\\" BaseFileName="pump">
14      <ExportComponents Desc="Add/remove value with component's
        index in project; remove this element to export active
        component only">
15        <Value Type="Integer">2</Value>
16      </ExportComponents>
17    </BatchAction>
18    <BatchAction Name="Save" OutputFile="pump_modified" Desc="CFT
      file name of modified project"/>
19  </CFturboBatchProject>
20 </CFturboFile>
```

During runtime a log-file **pump.log** is created in the directory of **pump.cft-batch**:

```

1 2017-11-13 12:39:09 [INFO] BatchMode - CFturbo 10.3.0.0 (BETA) - 13.11.2017
2 2017-11-13 12:39:09 [INFO] BatchMode - Time: 13.11.2017 12:39:09
3 2017-11-13 12:39:09 [INFO] BatchMode - File: c:\temp\a\pump.cft-batch
4 2017-11-13 12:39:09 [INFO] BatchMode - Logfile: c:\temp\a\pump.log
5 2017-11-13 12:39:09 [INFO] BatchMode - Working directory: c:\temp\a\
6 2017-11-13 12:39:09 [INFO] BatchMode - Script parameters: -batch c:\temp\a\pump.cft-batch
7 2017-11-13 12:39:09 [INFO] BatchMode - ***
8 2017-11-13 12:39:09 [INFO] BatchMode - Opening batch file: c:\temp\a\pump.cft-batch
9 2017-11-13 12:39:09 [INFO] BatchMode - Starting batchproject for input file: .\pump.cft
10 2017-11-13 12:39:09 [INFO] BatchMode - Using input file located in batch project file
    folder: c:\temp\a\pump.cft
11 2017-11-13 12:39:10 [INFO] PM - Open input file: c:\temp\a\pump.cft
12 2017-11-13 12:39:10 [INFO] BatchMode - Update design parameters
13 2017-11-13 12:39:10 [INFO] BatchMode - Applying geometry update
14 2017-11-13 12:39:19 [INFO] Component.Spiral.Cutwater - Run cut-water trimming
15 2017-11-13 12:39:22 [INFO] Component.Spiral.Cutwater - Run cut-water patch creation
16 2017-11-13 12:39:26 [INFO] BatchMode - Hints available:
17 2017-11-13 12:39:26 [INFO] BatchMode - Main dimensions: 2: Radial Impeller: Main
    dimensions are not updated automatically. Therefore the design could be not up-to-date.
18 2017-11-13 12:39:26 [INFO] BatchMode - Blade properties: 2: Radial Impeller: Blade
    angles are not updated automatically. Therefore the design could be not up-to-date.
19 2017-11-13 12:39:26 [INFO] BatchMode - CFD setup: 2: Radial Impeller: RSI connection
    is not available at inlet.
20 2017-11-13 12:39:26 [INFO] BatchMode - Model finishing: 2: Radial Impeller: Model
    finishing is currently NOT up-to-date
21 2017-11-13 12:39:26 [INFO] TAActionExport - Export-action (TurboGrid) found
22 2017-11-13 12:39:26 [INFO] BatchMode -
23 2017-11-13 12:39:26 [INFO] BatchMode - ##### Export interface: TURBOGRID
24 2017-11-13 12:39:26 [INFO] BatchMode - No working directory set, using default:
    c:\temp\a\
25 2017-11-13 12:39:26 [INFO] Export.CAE.Interface.TurboGrid - File:
    c:\temp\a\pump_Co2_hub.curve successfully exported
26 2017-11-13 12:39:26 [INFO] Export.CAE.Interface.TurboGrid - File:
    c:\temp\a\pump_Co2_shroud.curve successfully exported
27 2017-11-13 12:39:26 [INFO] Export.CAE.Interface.TurboGrid - File:
    c:\temp\a\pump_Co2_profile.curve successfully exported
28 2017-11-13 12:39:26 [INFO] Export.CAE.Interface.TurboGrid - File:
    c:\temp\a\pump_Co2.tse successfully exported
29 2017-11-13 12:39:26 [INFO] BatchMode - Saving export files successful
30 2017-11-13 12:39:26 [INFO] BatchMode - Start processing Save-action
31 2017-11-13 12:39:26 [INFO] BatchMode - Save output file: c:\temp\a\pump_modified.cft
32 2017-11-13 12:39:26 [INFO] BatchMode - ***
33 2017-11-13 12:39:26 [INFO] BatchMode - Batch mode terminated. (00:16.967 min)

```

3.3.2 Exit Codes

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

Exit Code	Description
0	No errors or warnings occurred during batch run.

Exit Code	Description
1	Last batch run was completed with warnings but no errors.
2	Last batch run was completed with errors.

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
- Ventilator
- Compressor
- Turbine

Project structure

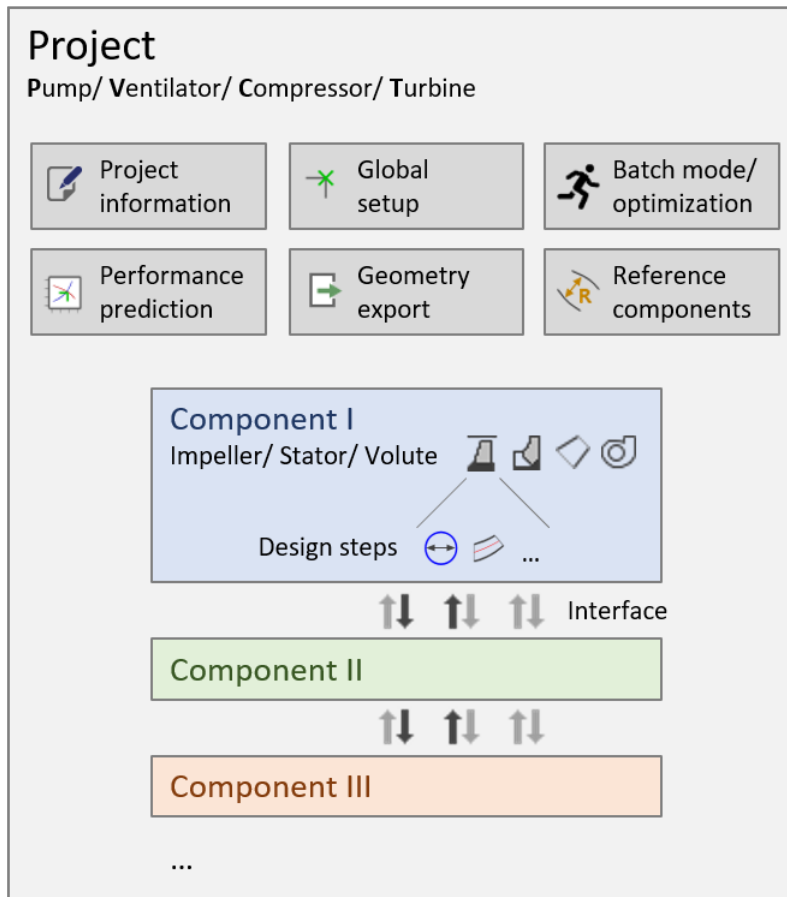
A project consists of the global parts

- [Project information](#)^[85]
- [Global setup](#)^[86]
- [Performance prediction](#)^[92]
- [Export](#)^[103]
- [Batch mode](#)^[176]
- [Reference components](#)^[181]

and the single component parts of the assembly. The following components are available:

- Up to 3 Impellers on any position
- 1 Volute as first/ last component
- any number of Stators (vaned or vaneless)

Components can be added directly in the [components view](#)^[223].



Coupling between components

The following coupling types are available:

↕	Coupling in flow direction (Default) Inlet cross section of a component is defined by the outlet cross section of previous component.
↕	Coupling reverse flow direction 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](#)^[222] directly at inlet or outlet the neighboring components.

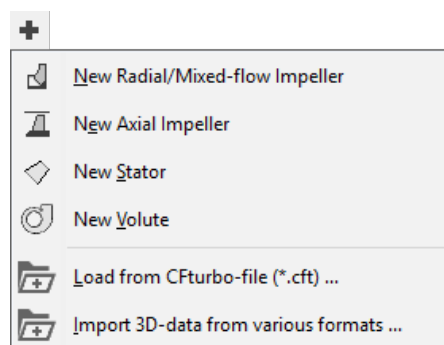
The impeller as the core component of a machine has primary sides both at inlet and outlet.

3.4.1 Add component

Components can be added at inlet or outlet side of the currently selected component.

There are 2 alternative possibilities to add components to the current project:

- Use **+** button in the [Meridian view](#)^[223]
- Use **+** button in the [component list](#)^[59] on left side



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

- up to 3 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.

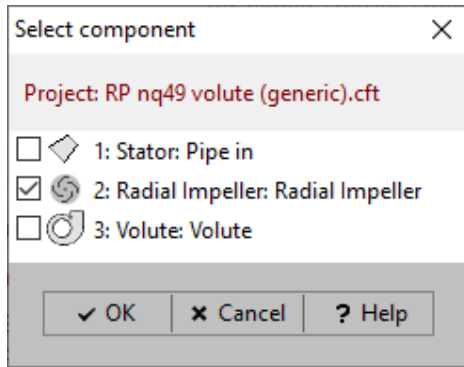
The menu is identical to the menu that is visible automatically after creating a [new project](#)^[75].

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](#)^[86] are applied for this new component.

After importing existing components, initially the component is disabled in order to preserve the original geometry. After [activation](#)^[63] the component will be updated and could be modified.

Load from CFturbo-file (*.cft)



CFturbo 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.

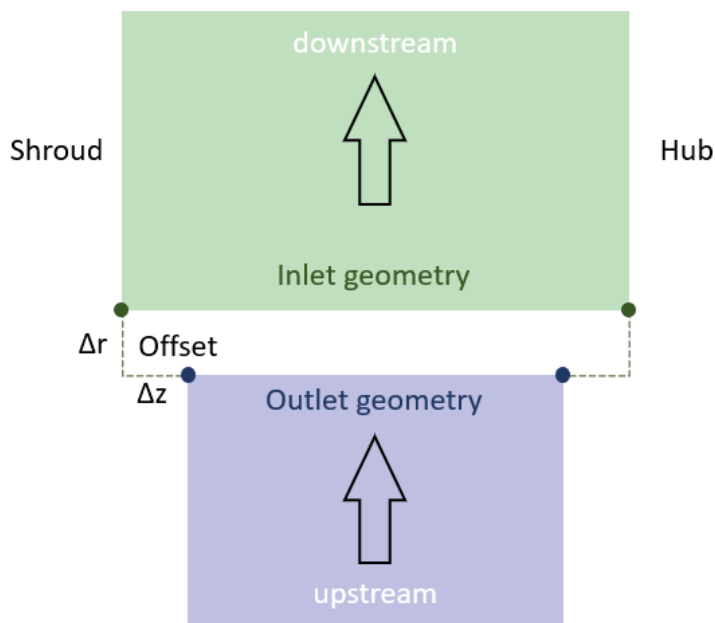
Import 3D-data from various formats

New components can be created using external geometry description.

Details can be found under [Import external geometry](#)⁷⁹.

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](#)^[63]), no adjustment will be effected - therefore an overlapping of neighboring components is possible, which is illustrated by a warning (see [Components](#)^[222]). The same applies if neighboring components are not coupled.

Coupling definition

The coupling definition at [volute inlet](#)^[514] as well as at stator [inlet](#)^[501] and [outlet](#)^[502] is made in a uniform manner.

Extent		Inlet		Outlet	
Neighboring component outlet					
Coupling: ↑↓ In flow direction (Fixed by Upstream Outlet)					
Hub	z =	40 mm	r =	137.3 mm	
Shroud	z =	25.4 mm	r =	137.3 mm	
▼					
Inlet					
<input checked="" type="checkbox"/> Center line <input type="checkbox"/> Hub, Shroud					
Offset					
Hub	Δz	0 mm	Δr	0 mm	
Shroud	Δz	0 mm	Δr	0 mm	
Absolute (incl. offset)					
Hub	z	40 mm	r	137.3 mm	
Shroud	z	25.4 mm	r	137.3 mm	

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](#)^[478] 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 (predecessor). 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](#)^[86].

Possible warnings

Problem	Possible solutions
Flow inlet does not match previous outlet Flow outlet does not match next inlet	
Inlet and outlet of flow region of two neighboring components are not matching.	<p>Check both sides (components) if the hub and shroud points are identical and no gap between the components exists.</p> <p>For impellers, an existing gap of the flow region can alternatively be closed by</p> <ul style="list-style-type: none"> • secondary flow path^[367] design in the meridional contour design step • extension^[479] (with option "connected") or RSI connection^[485] in the CFD setup

3.4.3 Automatic calculations

Some component design steps contain automatic calculations. ☒ **Automatic**

Currently these are:

- [Impeller main dimensions](#)^[244]: calculation of empirical parameters and efficiencies, calculation of dimension values
- Impeller blade properties: calculation of [blade angles](#)^[389] β_1 , β_1 (meanline design mode) or [Profile properties](#)^[466] (airfoil/ hydrofoil design mode)

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.

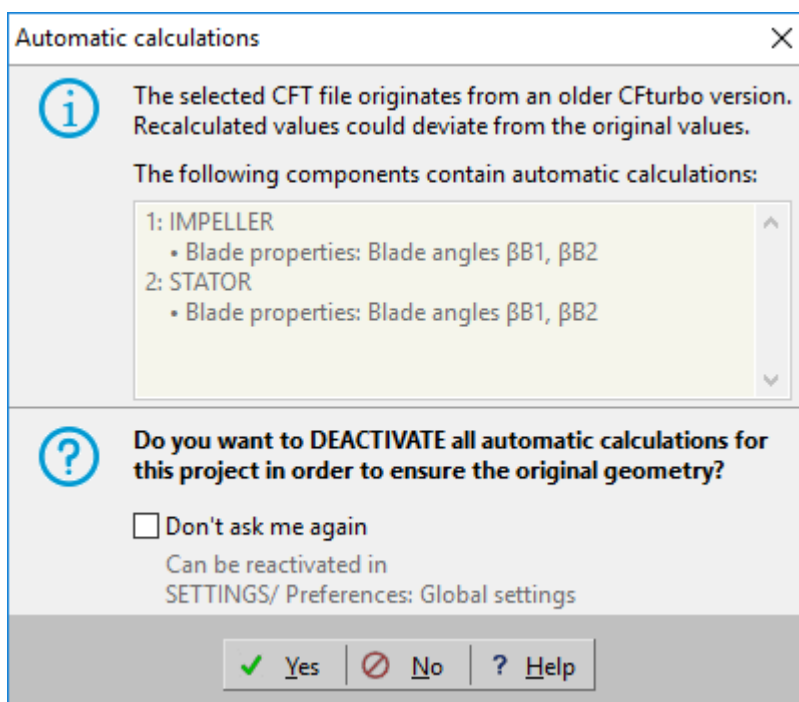
- **No automatic calculation:**

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.



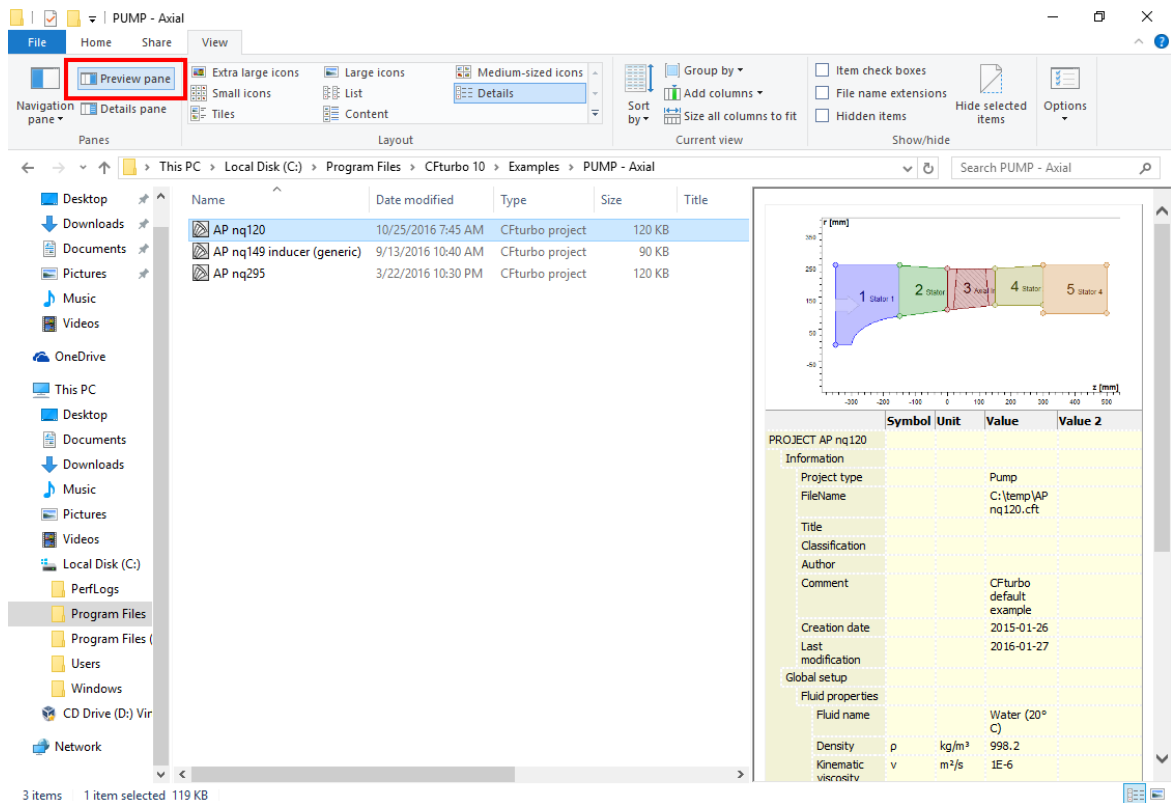
3.5 Windows Explorer integration

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

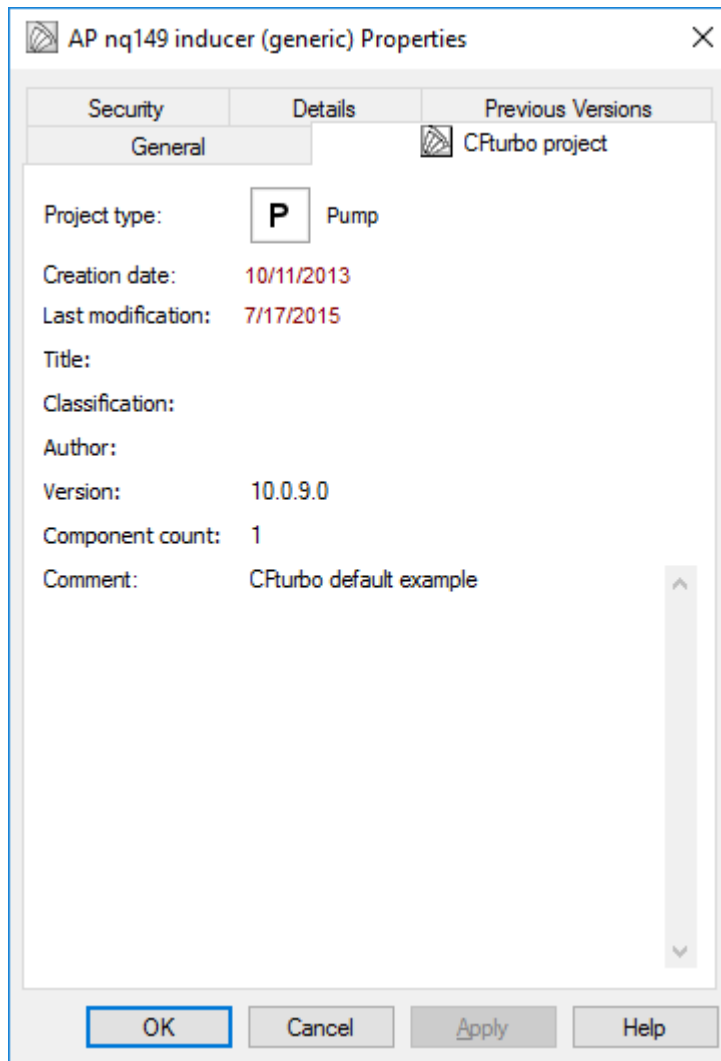
The preview pane of the Windows Explorer can be used to display a preview of the content of a file. It can be enabled on the view ribbon/menu of Windows Explorer or by pressing Alt + P and will show an excerpt of the report and an image of the meridian shape of the design.



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.

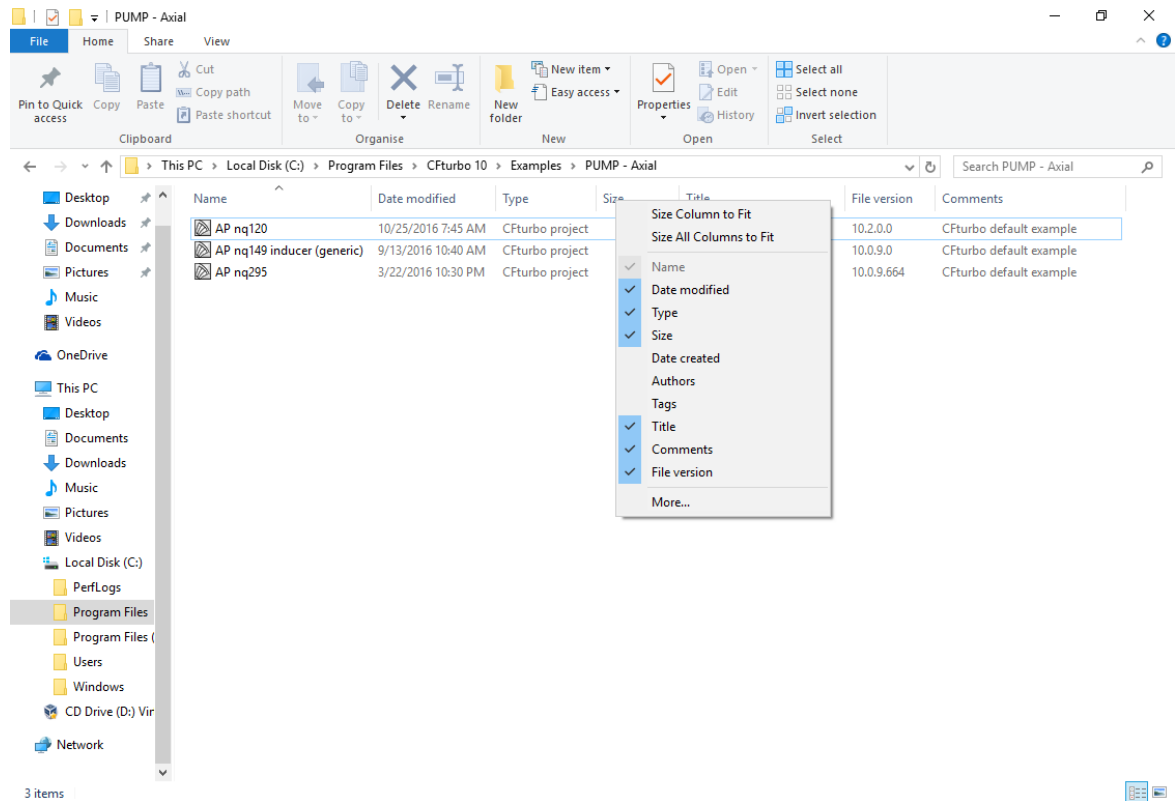


Additional columns and Windows Search

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 Troubleshooting

This chapter provides information on how problems can be handled:

→ [Error reporting](#) ⁴⁹

→ [Emergency recovery](#) ⁵²

→ [Known problems](#) ⁵³

3.6.1 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

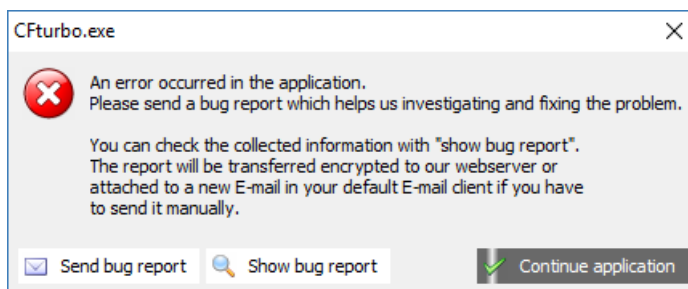
Follow the *Send assistant* to add user and contact information as well as configuring the bug report. Finally, the report will be sent to our web server encrypted.

- Show bug report

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

- Continue application (Default)

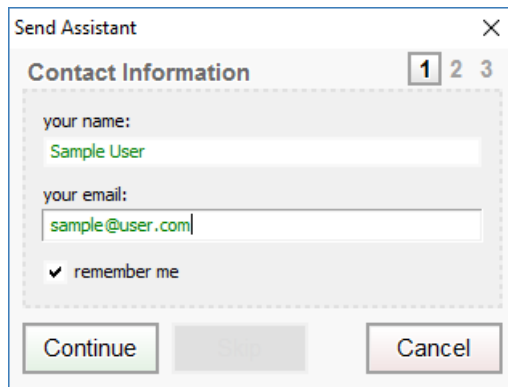
Continue working with CFturbo without sending the bug report.



Send assistant

The Send assistant will guide you sending the bug report.

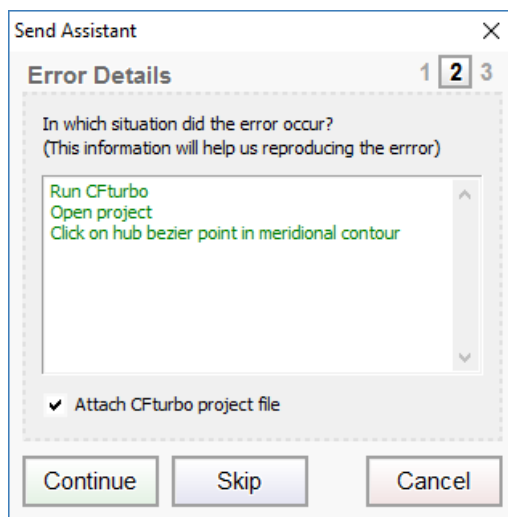
In the first step, you will be asked for your contact information so that the support team is able to contact you if additional information is needed or a solution for the problem is available.



The dialog box is titled "Send Assistant" and has a close button (X) in the top right corner. It features three tabs labeled "1", "2", and "3", with "1" being the active tab. The active tab is titled "Contact Information". Inside this tab, there are two text input fields: "your name:" with the value "Sample User" and "your email:" with the value "sample@user.com". Below these fields is a checkbox labeled "remember me" which is checked. At the bottom of the dialog, there are three buttons: "Continue" (highlighted in green), "Skip" (disabled), and "Cancel" (highlighted in red).

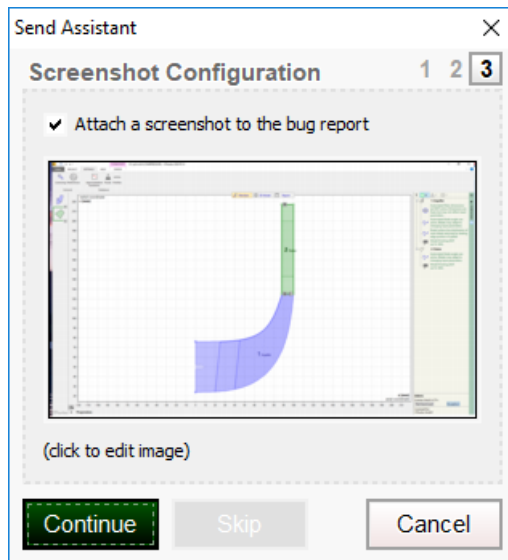
The second step asks you for the details of the situation, the error occurred in. Please note that it is extremely helpful if the error can be reproduced.

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



The dialog box is titled "Send Assistant" and has a close button (X) in the top right corner. It features three tabs labeled "1", "2", and "3", with "2" being the active tab. The active tab is titled "Error Details". Inside this tab, there is a text area with the prompt "In which situation did the error occur? (This information will help us reproducing the error)". The text area contains the following text: "Run CFturbo", "Open project", and "Click on hub bezier point in meridional contour". Below the text area is a checkbox labeled "Attach CFturbo project file" which is checked. At the bottom of the dialog, there are three buttons: "Continue" (highlighted in green), "Skip" (disabled), and "Cancel" (highlighted in red).

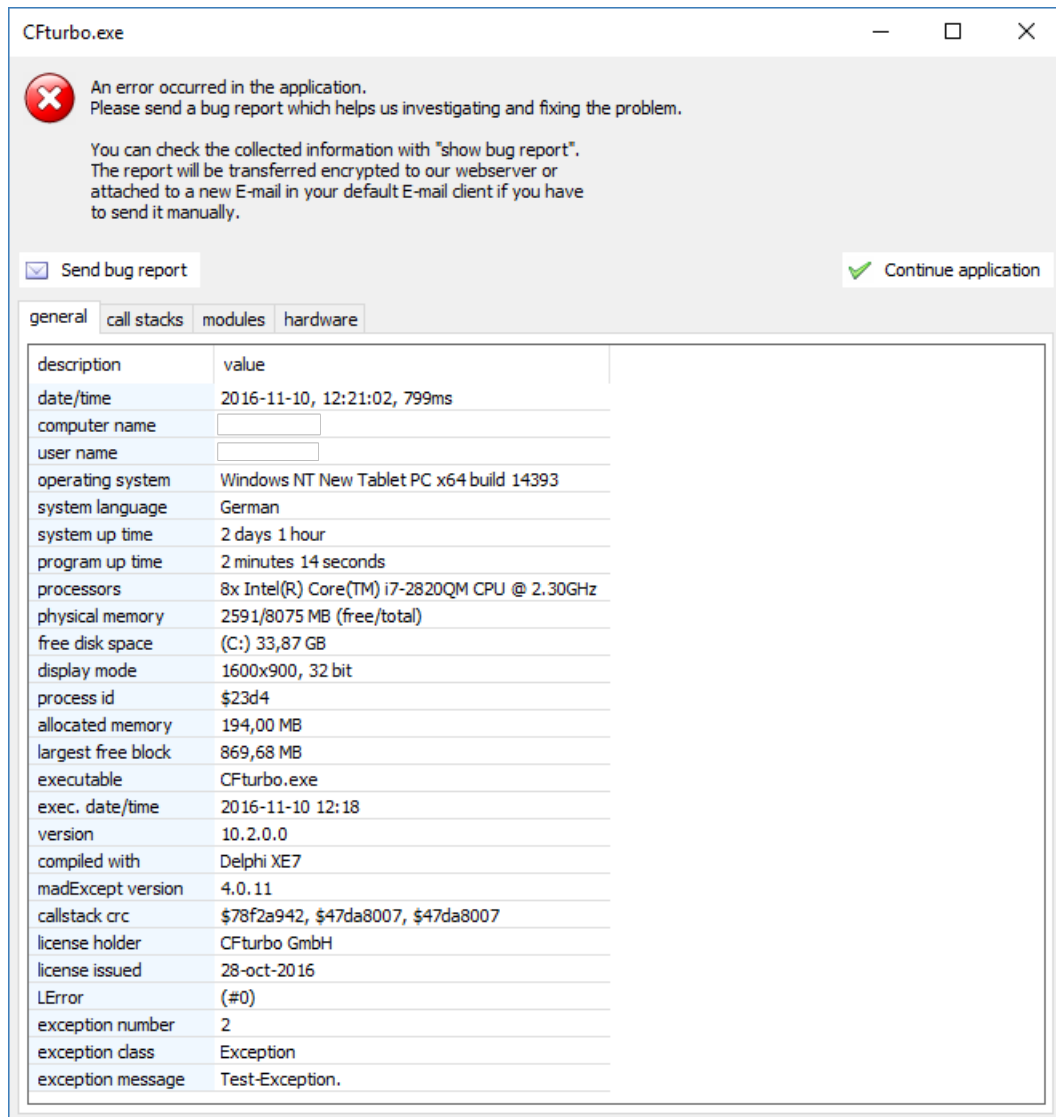
Finally you can choose if a screenshot should be attached. If Continue is clicked, the report will be sent encrypted to our web server.



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

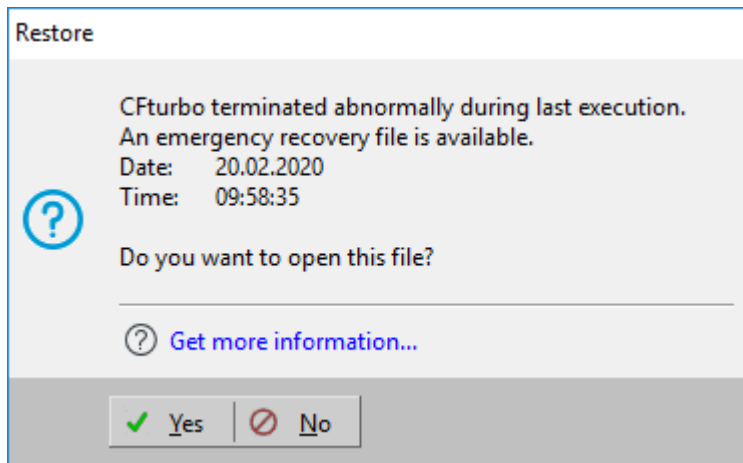
The detail view shows you the information that is collected about the error and the current state of CFturbo. Also basic system information is included.



3.6.2 Emergency recovery

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

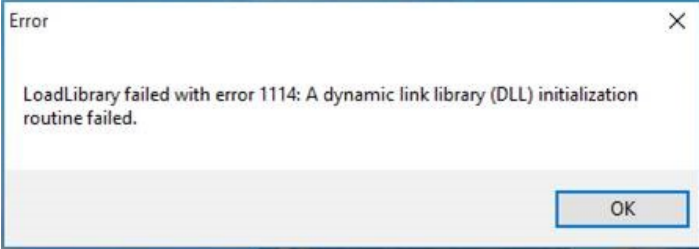
In this case the following message is displayed and one can open this last project state optionally.

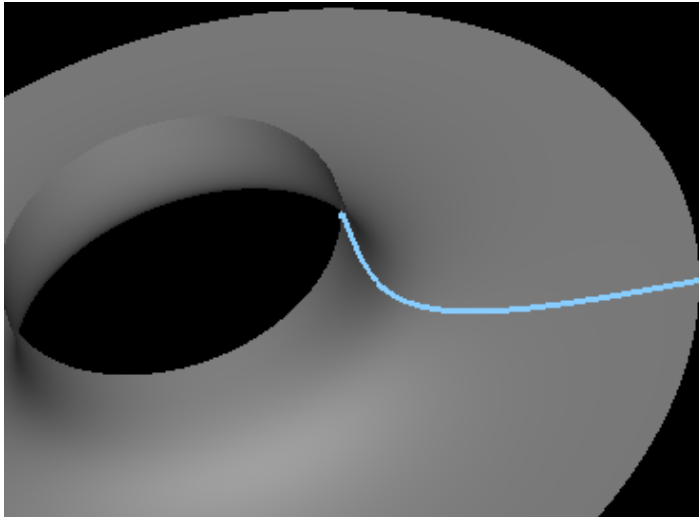


The last project state is the newest item of the [Undo](#) ¹⁰² list of the previous project.

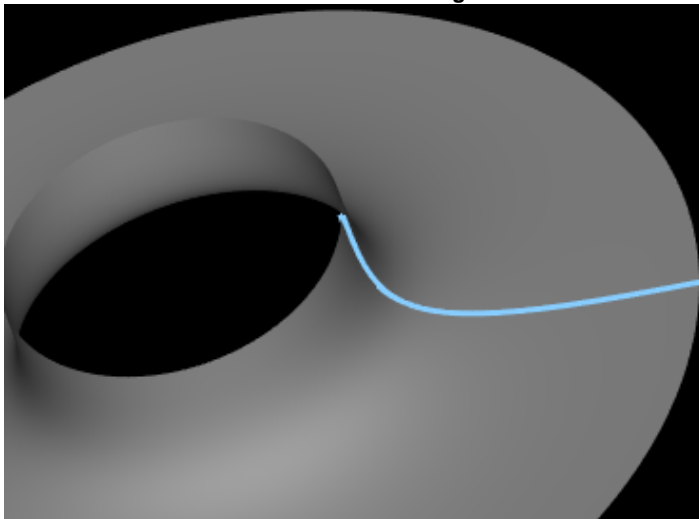
3.6.3 Known problems

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

Problem	Possible solutions
<p>When CFturbo is started, the following error message is displayed:</p>  <p>LoadLibrary failed on Windows 10</p>	<p>Update the graphics card driver.</p>
<p>The anti-aliasing of the 3D model does not work.</p>	<p>Anti-aliasing (MSAA) settings of CFturbo should not be overridden by graphics card driver. Change your driver settings to use application settings.</p>



Without anti-aliasing



With anti-aliasing

Part

IV

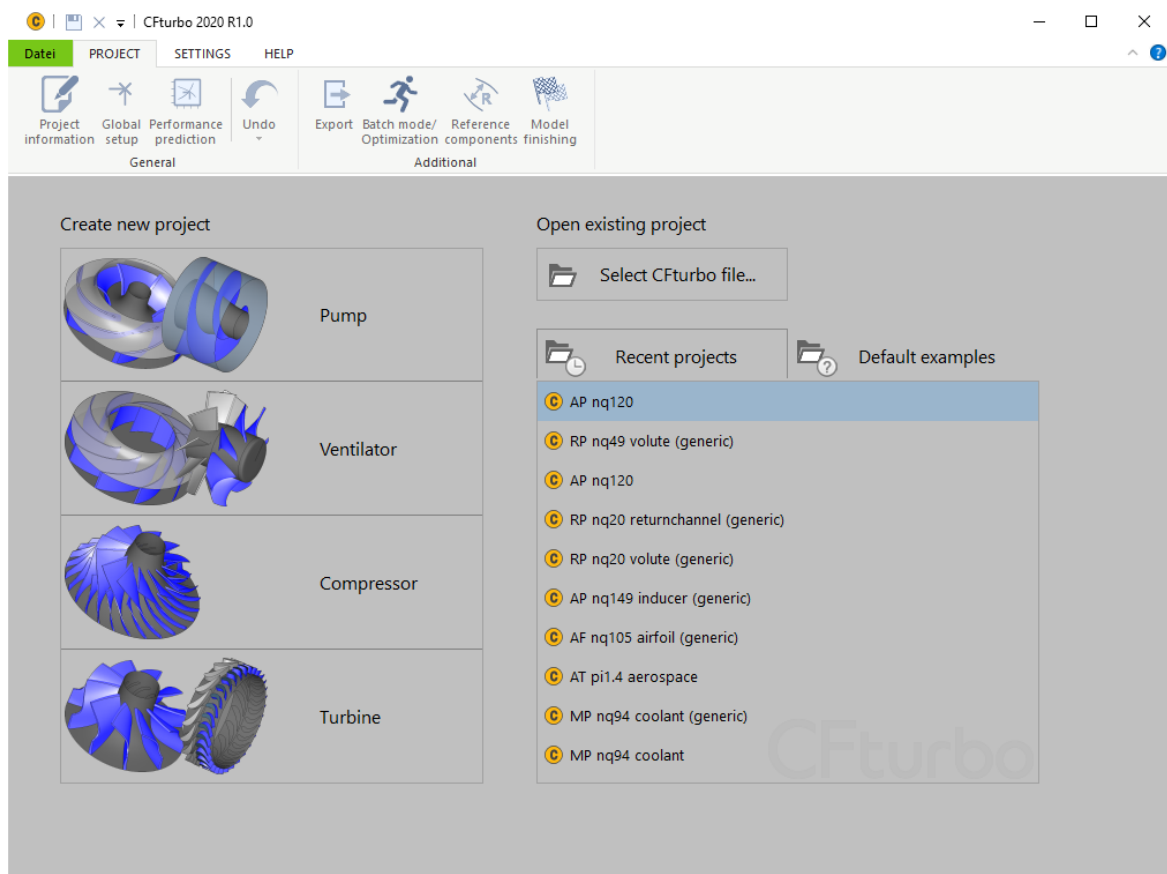
4 Getting started

This chapter describes how to start using CFturbo:

- [Start](#) ^[56]
- [Opened project](#) ^[59]
- [Component design process](#) ^[61]
- [Activate/ Rename/ Delete components](#) ^[63]
- [Remove design steps](#) ^[65]

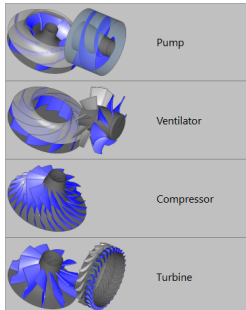
4.1 Start

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:



- Pump
- Ventilator
- Compressor
- Turbine

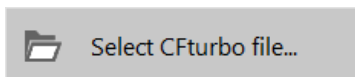
These 4 buttons correspond to the menu item [File/ New](#)^[75].

After creating a new project the [Global Setup](#)^[86] dialog is starting automatically.

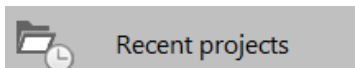
Afterwards several components can be added to the project.

Open existing project

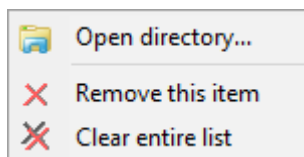
Here you can select existing projects:



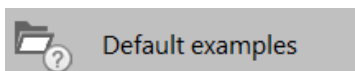
Open any CFturbo project (*.cft) via file opening dialog (corresponds to the menu item ["File/ Open"](#)^[78])



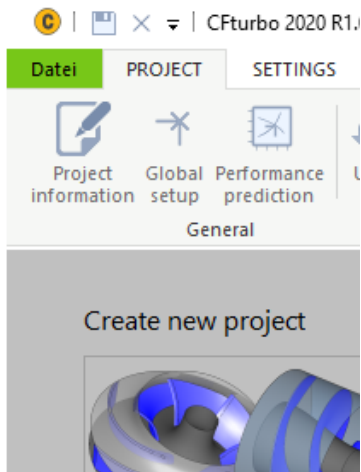
Here you can select one of the 10 recently used projects. The full filename is displayed as a hint if you move the mouse cursor over any item.



By right click on any item you can open the corresponding directory, remove the item or clear the entire list.



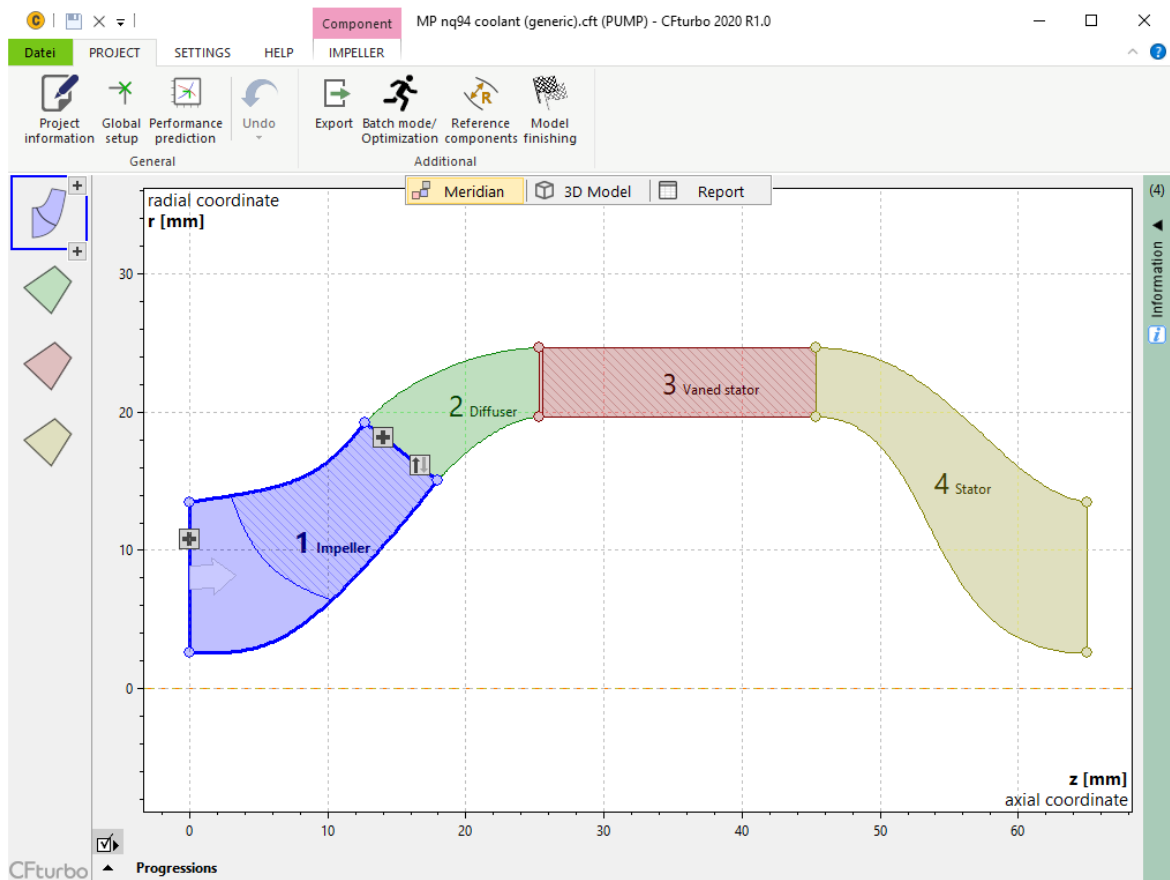
Open one of the CFturbo default examples from the installation directory.



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

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](#)^[74] 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](#)^[222]).

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:



Radial or mixed-flow impeller



Axial impeller



Stator (vaned or unvaned)



Volute

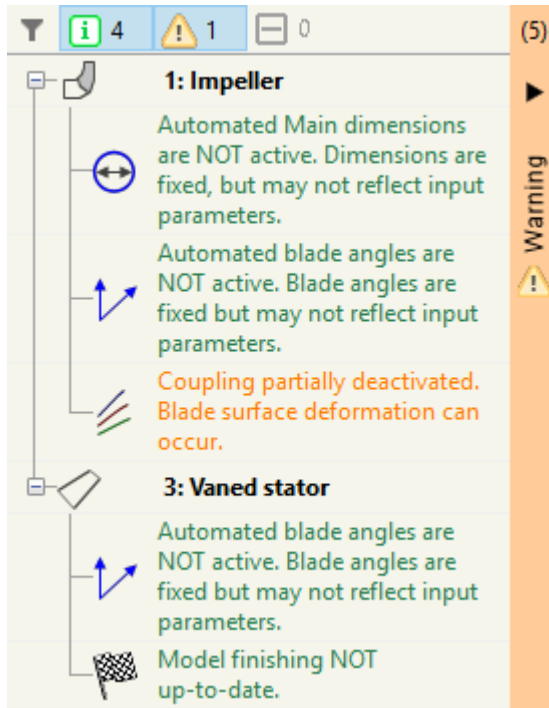
b) Three alternative views in the central part

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 shown.

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.

The type of a message (warning/ error/ information) is shown when hovering the mouse cursor over it.

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](#)^[243], [volute](#)^[513], [stator](#)^[495]).

After completing a components basic design process, optional design steps related to [model finishing](#)^[467] and [CFD setup](#)^[476] become available.

Each design step comes with its own dialog that can be accessed via the [component specific menus](#)^[217] or the components [context menu](#)^[223] 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:

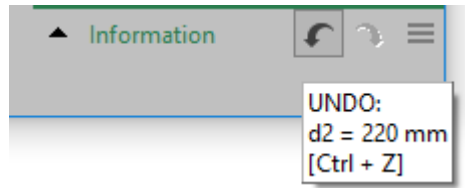
- ✓ **OK** Closes the dialog and saves user changes into the project.
- ✗ **Cancel** Closes the dialog and discards all changes made.

**Help**

Opens the help topic related to the current design step.



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 comfortable 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.



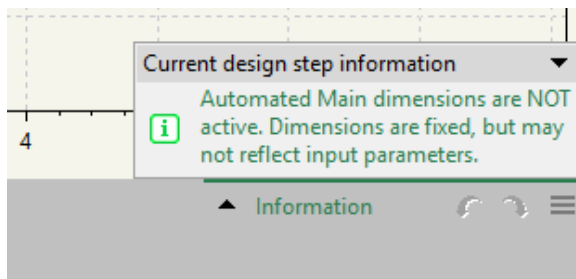
Complete all design steps

- 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](#)^[244] 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](#)^[60] 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.

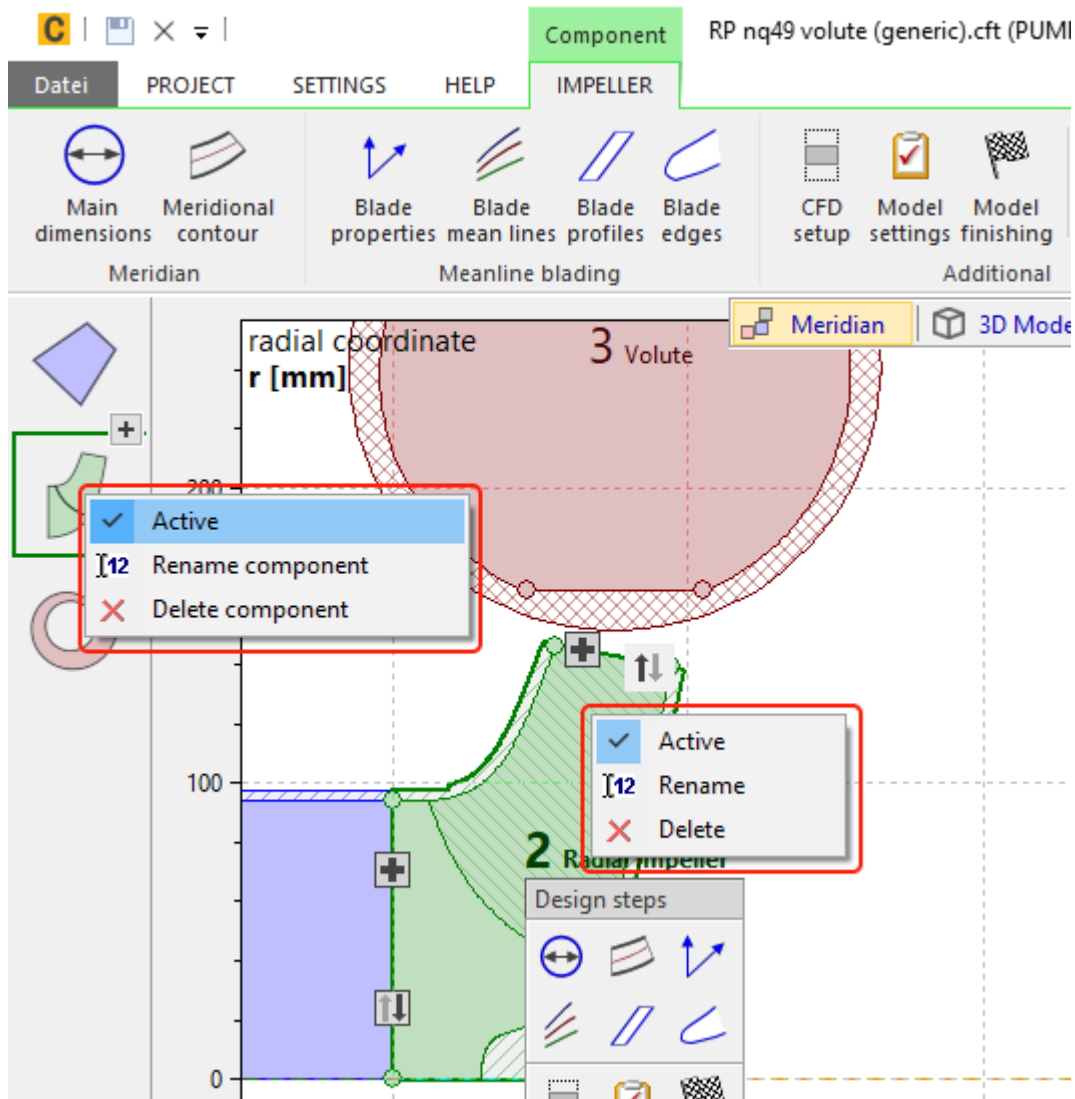
See also [Opened project/ message panel](#)^[60]. Usually you can find more information about a message in the online help by clicking on its text.

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

4.4 Activate/ Rename/ Delete components

The actions **Active**, **Rename** and **Delete** 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 meridional preview



Active

A inactive component is read only and also not going to be updated automatically. Inactive components are colored grey 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 meridional view, the 3D view and the report.

Delete

The selected component is deleted. If the Meridian-View is selected, the key on the keyboard can be used alternatively.

4.5 Remove design steps

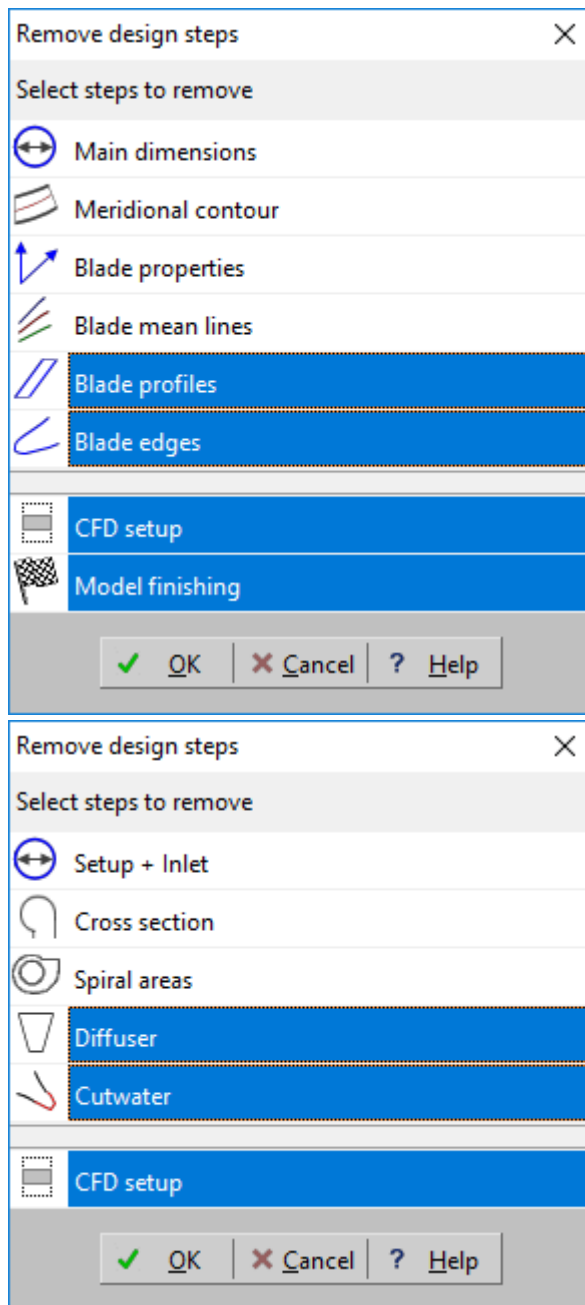
? IMPELLER/ STATOR/ VOLUTE | Additional | Remove design steps



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:

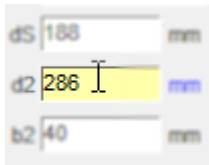
- [General handling](#)⁶⁷
- [Graphical dialogs](#)⁶⁷

→ [Progression dialog](#) 

→ [Edit fields with empirical functions](#) 

4.6.1 General handling

Value input





Value input in

- Edit controls and
- 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

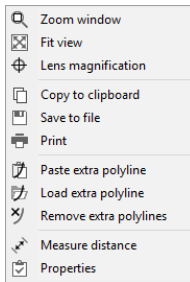
- Position and size of dialogs are saved to restore it in the same way when they are called again.

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:

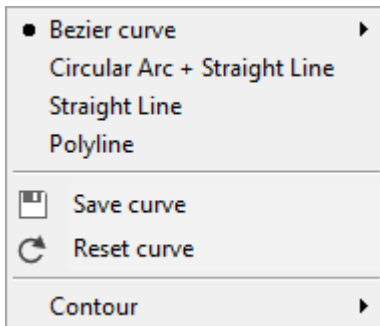


- Zoom window by mouse
- Fit view
- Lens magnification at mouse cursor position
- Copy diagram to clipboard
- Save diagram as BMP, GIF, JPG, PNG or WMF
- Print diagram
- Paste points from clipboard into an extra polyline
- Add any [polyline from file](#) (x,y points) to compare different curves (alternatively 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

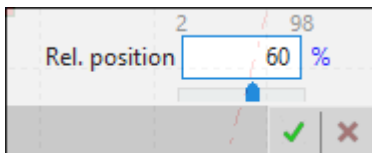
Context sensitive popup menus

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.

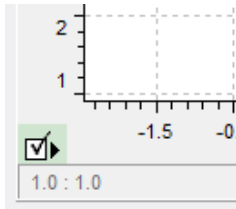


An alternate method to specifying Bezier points by the mouse, you may enter the accurate coordinates of Bezier points in a small dialog window that appears by clicking the right mouse button on the chosen Bezier point.

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

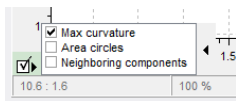
extreme values left or bottom (0) and right or top (1). Normalized relative coordinates are giving the advantageous possibility of an automatic update of the entire design if a parameter is modified.

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.

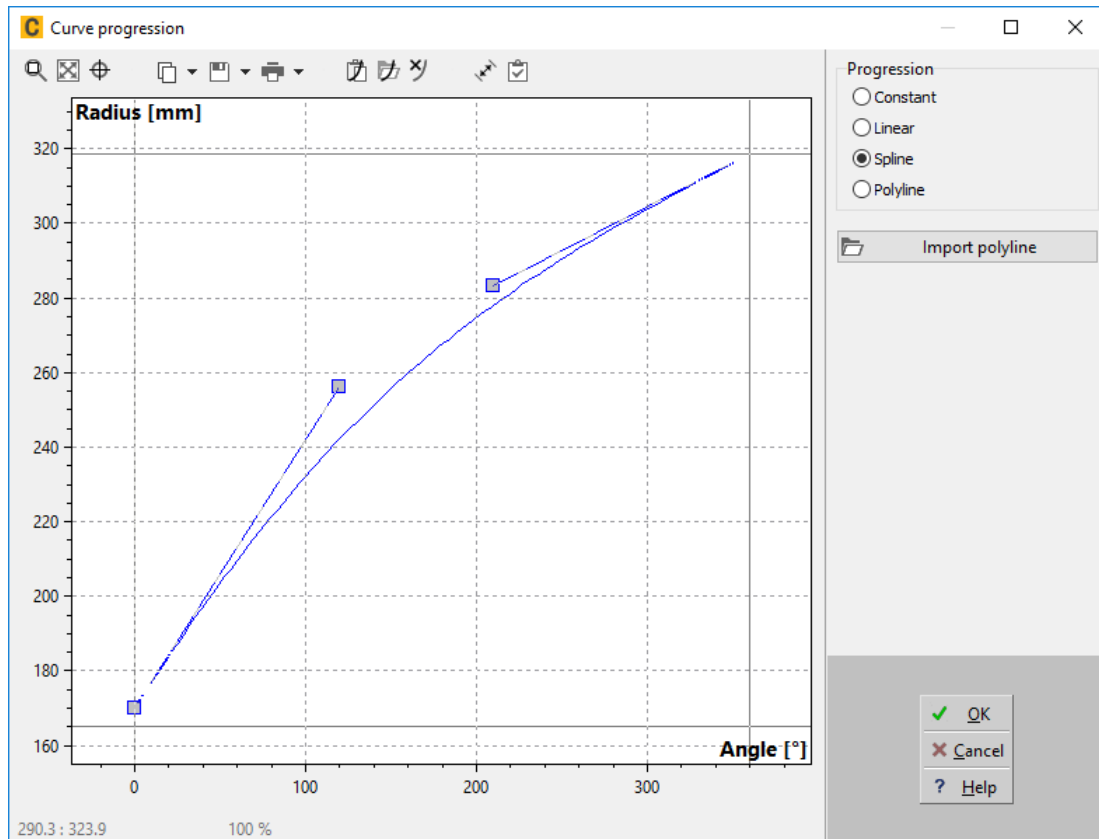
Auto fit view (for docking diagrams)



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.



Availability

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

- Cross section progression, in [Meridional contour](#) ³³⁸
- Angular positions, in [Blade mean lines](#) ⁴⁰⁵
- Spiral cross section progression, in [Spiral development areas](#) ⁵³¹

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.

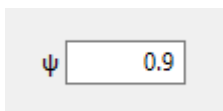
Text file format:

<pre># 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</pre>	<ul style="list-style-type: none"> • 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. <p>A sample file can be generated by right clicking the progression curve and selecting "Save polyline".</p>
---	---

4.6.4 Edit fields with empirical functions

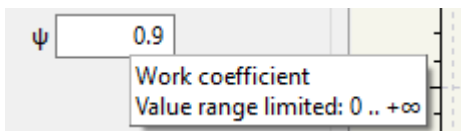
Some edit fields are connected with [empirical functions](#)¹⁹⁸. This becomes visible when activating the edit field by mouse click.

Default



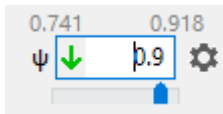
Default appearance of edit field.

Mouse-over



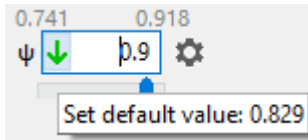
Appearance if the mouse cursor is over the edit field. Min. and max. values are displayed if a recommended range exists.

Focused



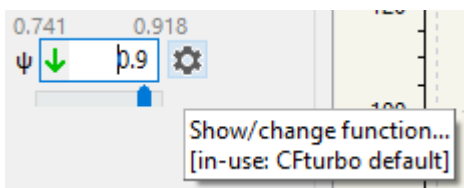
Appearance, if the edit field is focused (mouse click into the field). If a recommended range exists, min. and max. values are displayed as well as a sliding bar below.

Default value



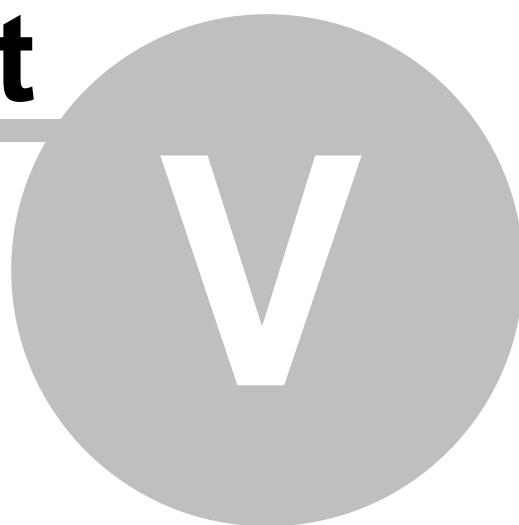
The default value can be selected by pressing the arrow button left. The numerical default value is displayed as hint.

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.

Part

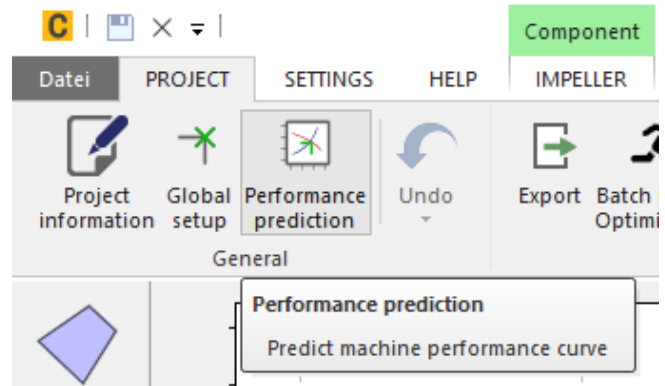


5 Menu

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

Hints

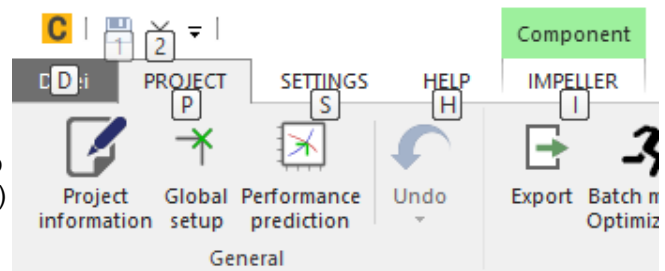
The buttons have hints if they are not self-explanatory. The hint becomes visible when the mouse cursor is on the button.



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](#)⁷⁵ left in the ribbon contains basic file handling operations.

The tab pages contain control elements grouped by functionality:


page	visible
PROJECT ⁸⁴	always
SETTINGS ¹⁸⁵	

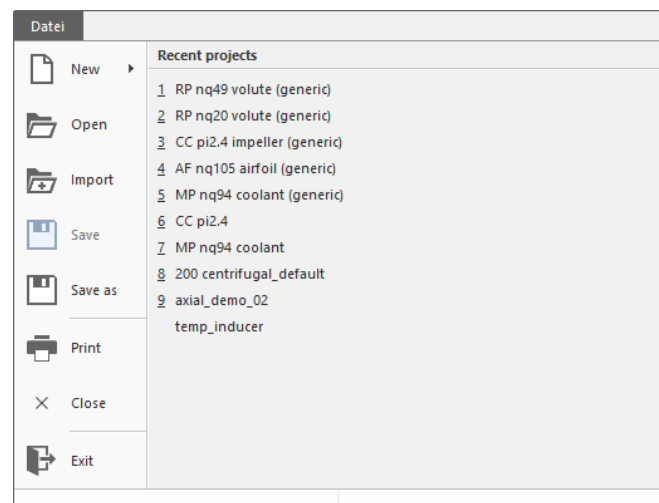
HELP 		
IMPELLER 	if the <u>selected component</u> is a	impeller 
STATOR 		stator 
VOLUTE 		volute 
3D MODEL 	if the <u>corresponding view</u> is selected	3D view 
3D-MODEL - BLADES 		
REPORT 		Report view 

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](#) 



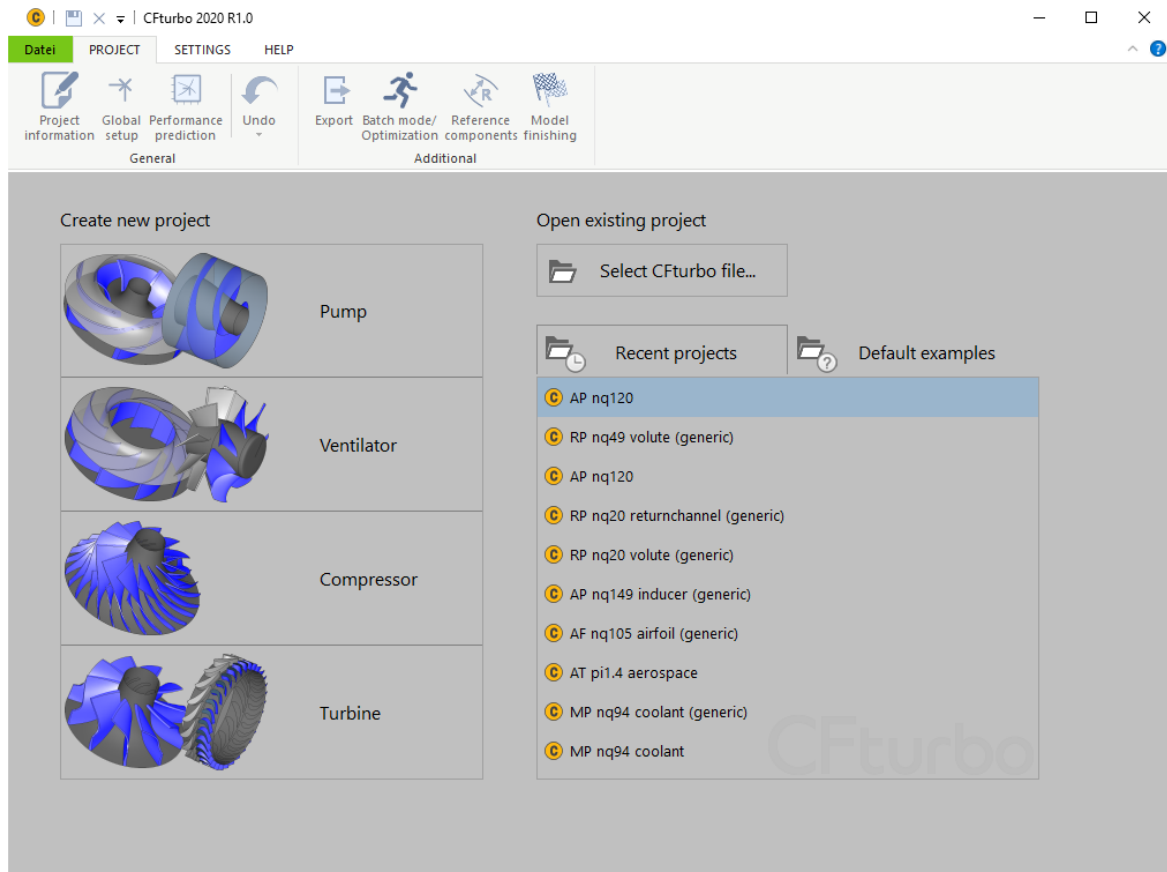
5.1.1 Create new design

? **File | New** 

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

- Pump
- Ventilator
- Compressor
- Turbine

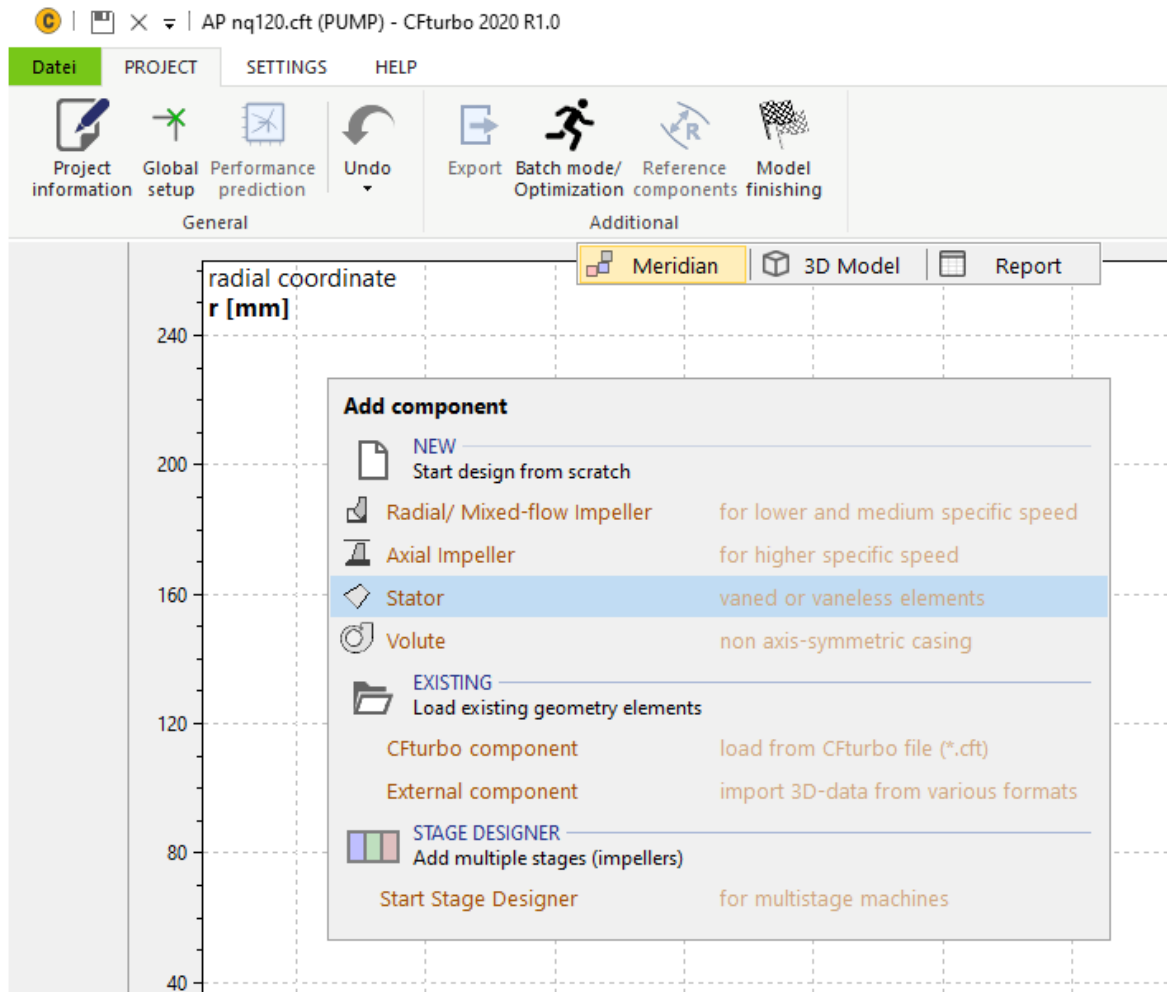
Equivalent to using this menu, the buttons in the **Create new project** area can be used, see [Start](#)^[56].



The [Global Setup](#)^[86] 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.

- General information about adding new components: see [Add component](#)^[41]
- Specific for multi-stage machines: see [Stage designer](#)

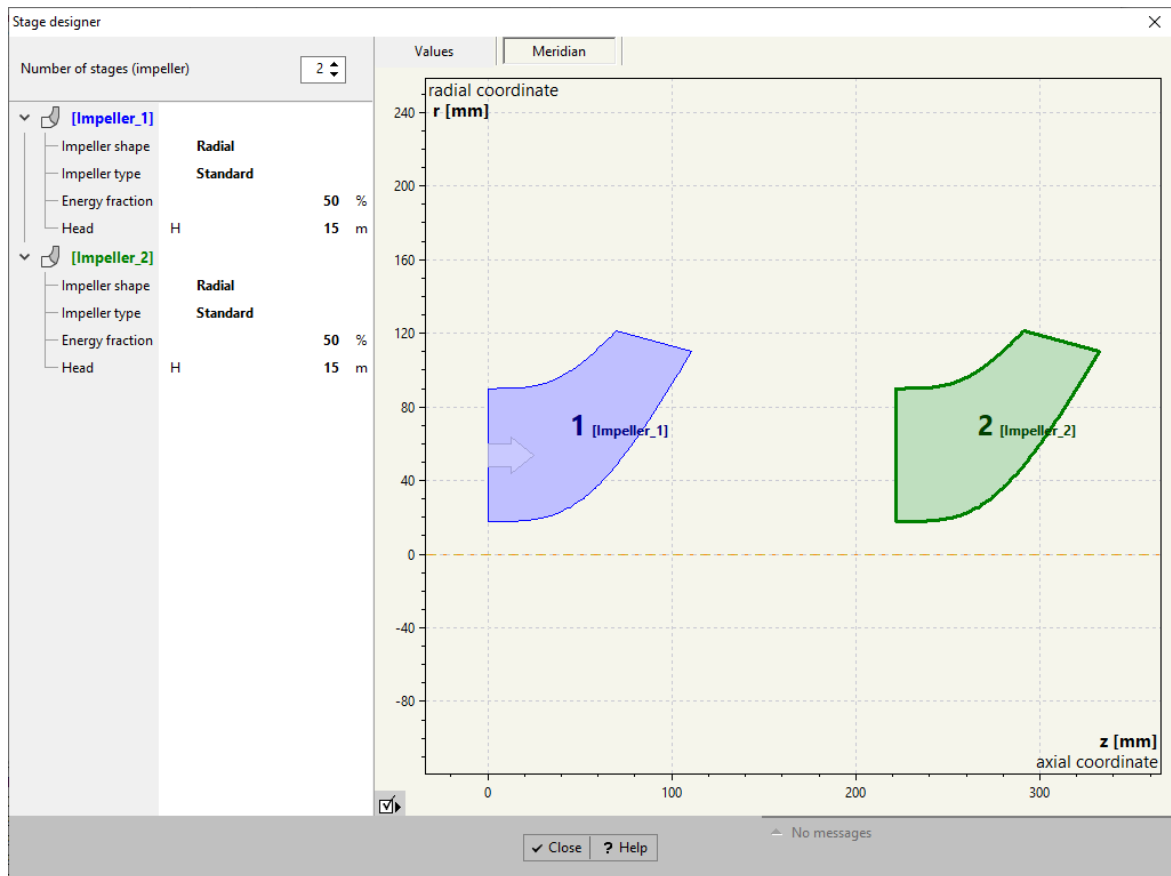


5.1.1.1 Stage designer

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](#)^[86] 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](#)^[244]. 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](#)^[56].

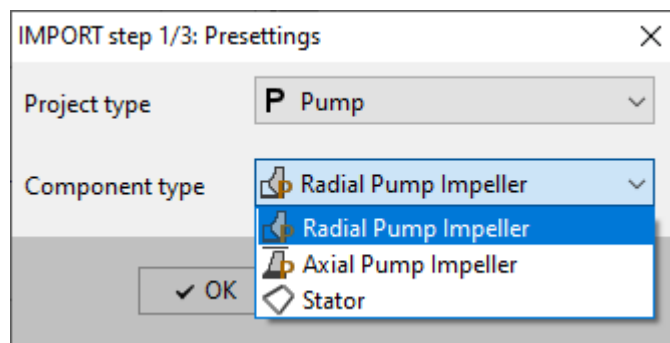
The user can modify the filename by the **Save as** function in order to save modified designs under different file names.

5.1.3 Import external geometry

? File | Import

This menu item allows direct import of external geometry description. The import process contains 3 steps:

1. Select project and component type



2. [Global setup](#)  for the new project

IMPORT step 2/3: Global setup

Design point ⓘ

Flow rate Q 150 m³/h

Head H 50 m

Revolutions n 3000 /min

Fluid

Name Water (20°C) ⓘ

Inlet conditions

Pressure (total) pt 1 bar

Temperature T 20.0 °C

Optional

Rotation direction Right (clockwise) ▾

Add'l. Hydraulic efficiency η_{h+} 100 % ⓘ

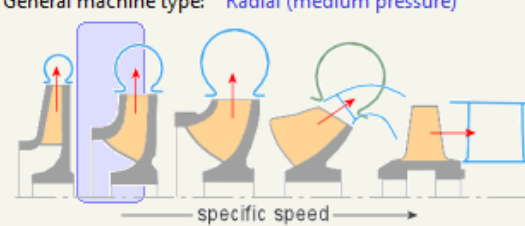
Pre-swirl

Swirl angle Swirl number Swirl energy

$\alpha_s = \tan^{-1}(c_{mS}/c_{uS})$ α_h 90.0 °

α_s 90.0 °

General machine type: Radial (medium pressure)



Specific speed (EU)	nq	33
Specific work	Y	490.5 m²/s²
Power output	PQ	20.4 kW
Mass flow	m	41.592 kg/s
Total-to-total pressure difference	Δp_t	4.8962 bar

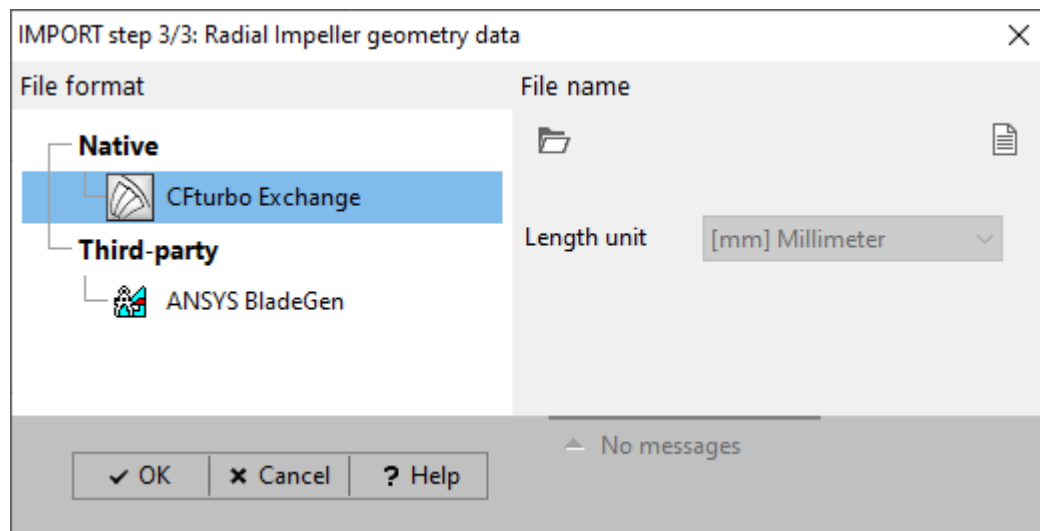
✓ OK ✕ Cancel ? Help

⬆ No messages

It is important that the direction of rotation is specified in accordance to data to be imported.

If the import is not initiated via File menu but by the **+** button (see [Add component](#)⁴¹⁾) within an existing project, then the next step is obsolete and is not available.

3. Select import format and file



5.1.3.1 Native format

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 lines and blade thickness data must be provided for both main and splitter blade. Only symmetric blades are supported.

Span positions	Relative position between hub and shroud (0...1). Array of at least 2 float numbers.
Mean line data	Array of at least 2 curves. Each curve contains an array of 3D-points (r, T, z coordinates).
Thickness data	<p>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:</p> <p>x = relative point position on mean line y = blade thickness at this position</p> <p>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.</p>

An example file can be easily generated by [exporting](#) ¹⁰³ any CFturbo component using the "CFturbo Exchange" export interface.

5.1.3.2 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:

	Possible values	Description
Number of blades	Positive Integer	Number of main blades
Splitters flag	0 or 1	0 if no splitter blades
Pitch fraction	Positive float number lower than 1	Position of splitter blades (ignored for main blade)
Number of layers	Integer greater than 1	Number of layers

Thickness flag	N or T	<p>N = Normal thickness values T = Tangential thickness values</p> <p>For more details see Thickness definition^[439].</p>
Span fraction	Float number between 0 and 1	Relative position of a span section
Number of points	Integer greater than 1	Number of points describing geometrical data for a span section
r	Nonnegative float number	Radial coordinate value for a point
theta	Float number	Circumferential blade angle value for a point in radians
z	Float number	Axial coordinate value for a point
thickness	Nonnegative float number	<p>Blade thickness value for a point.</p> <p>Parts of meridional contour without blades are represented using a zero thickness value</p>

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

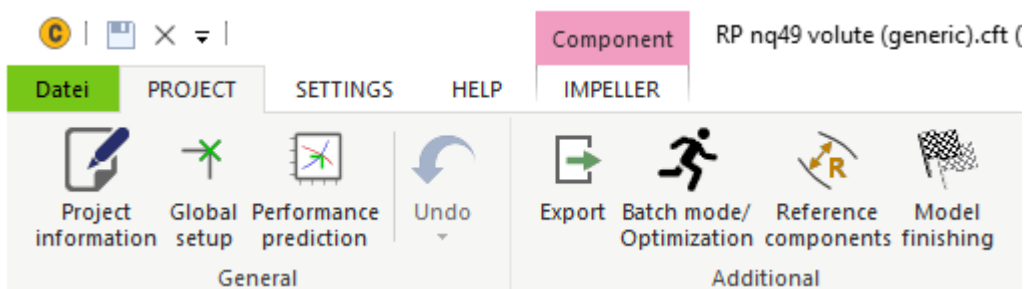
r theta z thickness

An example file can be easily generated by exporting any example using the [BladeGen](#)^[131] export interface.

5.2 PROJECT

A project can consist of several components (see [Project structure and interfaces](#)^[39]). All components can be designed separately, whereas they influence each other on the interfaces due to geometrical constraints and fluidic coupling.

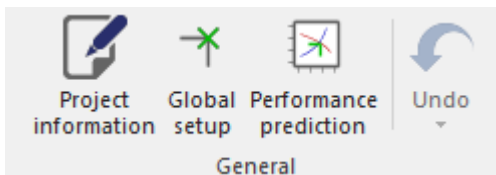
The Project menu is split into [general](#)^[84] and [additional](#)^[103] functions.



5.2.1 General

? PROJECT | General

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



→ [Project information](#)^[85]

→ [Global setup](#)^[86]

→ [Performance prediction](#)^[92]

→ [Undo](#)^[102]

5.2.1.1 Project information

? PROJECT | General | Project Information

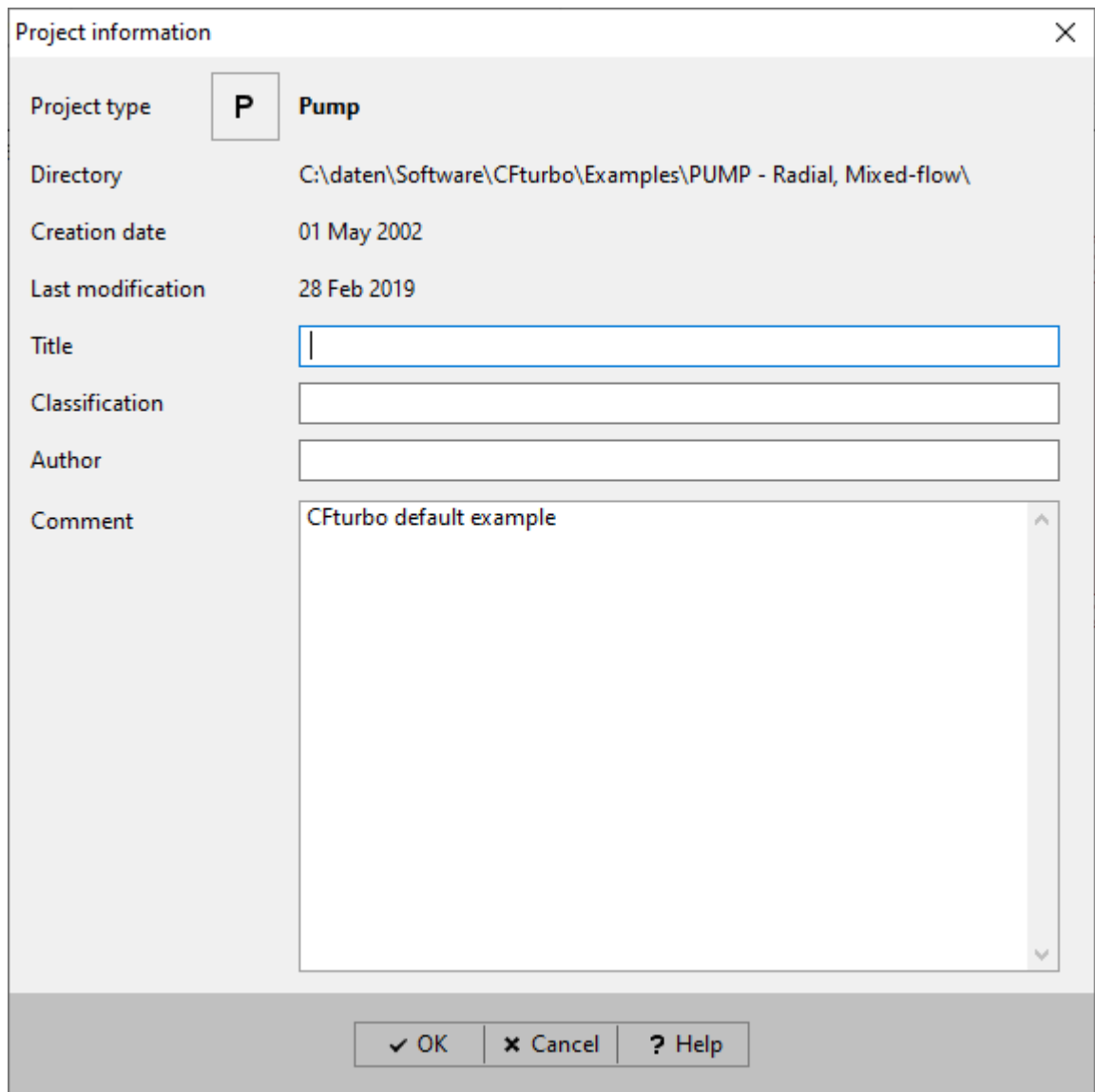


For identification of the project can be specified:

- 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.



The image shows a 'Project information' dialog box with a close button (X) in the top right corner. The dialog contains the following fields:

- Project type:** A button labeled 'P' and the text 'Pump'.
- Directory:** C:\daten\Software\CFturbo\Examples\PUMP - Radial, Mixed-flow\
- Creation date:** 01 May 2002
- Last modification:** 28 Feb 2019
- Title:** An empty text input field.
- Classification:** An empty text input field.
- Author:** An empty text input field.
- Comment:** A text area containing 'CFturbo default example' with a vertical scrollbar on the right.

At the bottom of the dialog are three buttons: '✓ OK', '✗ Cancel', and '? Help'.

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 below, for compressors on the right side.

Global setup

Design point

Flow rate: $Q = 454 \text{ m}^3/\text{h}$

Head: $H = 30 \text{ m}$

Revolutions: $n = 1770 \text{ /min}$

Fluid

Name: Water (20°C)

Inlet conditions

Pressure (total): $p_t = 1 \text{ bar}$

Temperature: $T = 20.0 \text{ °C}$

Optional

Rotation direction: Right (clockwise)

Add'l. Hydraulic efficiency: $\eta_{ht} = 90 \%$

Pre-swirl

Swirl angle: $\delta_r = 1 - c_{us}/u_s$

Swirl number: $\delta_{rh} = 1$

Swirl energy: $\delta_{rs} = 1$

General machine type: Radial (low pressure)

Specific speed (EU): $nq = 49$

Specific work: $Y = 294.3 \text{ m}^2/\text{s}^2$

Power output: $PQ = 37.05 \text{ kW}$

Mass flow: $m = 125.88 \text{ kg/s}$

Total-to-total pressure difference: $\Delta p_t = 2.9377 \text{ bar}$

OK Cancel Help

Global setup

Design point

Volume flow (total): $Q_{ts} = 4852.8 \text{ m}^3/\text{h}$

Total pressure ratio: $\pi_{tt} = 2.375$

Revolutions: $n = 27000 \text{ /min}$

Gas

Name: Air

Model: Perfect

Inlet conditions

Pressure (total): $p_{t5} = 1 \text{ bar}$

Temperature (total): $T_{t5} = 20.0 \text{ °C}$

Optional

Rotation direction: Right (clockwise)

Add'l. Total-to-total efficiency: $\eta_{tt} = 100 \%$

Pre-swirl

Swirl angle: $\delta_r = 1 - c_{us}/u_s$

Swirl number: $\delta_{rh} = 1$

Swirl energy: $\delta_{rs} = 1$

General machine type: Radial (medium pressure)

Specific speed (EU): $nq = 36$

Specific work: $Y = 82586 \text{ m}^2/\text{s}^2$

Total-to-total pressure difference: $\Delta p_t = 1.375 \text{ bar}$

Power output: $PQ = 132.3 \text{ kW}$

Mass flow: $m = 1.6016 \text{ kg/s}$

Sonic speed (total): $at_1 = 343.26 \text{ m/s}$

Total density: $\rho_{t1} = 1.1882 \text{ kg/m}^3$

OK Cancel Help

Design point

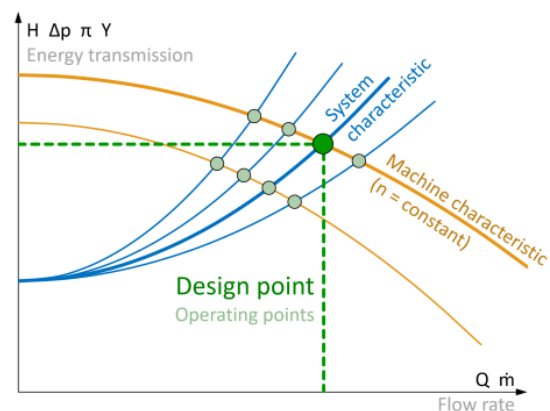
Here you have to enter the design point data:

(1) Flow rate:

- **for pumps, ventilators:** volume flow Q or mass flow m
- **for compressors:** mass flow m or volume flow Q (referring to total state on suction side)
- **for turbines:** mass flow m

(2) Energy transmission:

- **for pumps:** head H or total pressure difference p_t
- **for ventilators:** total pressure difference p_t
- **for compressors:** total pressure ratio π_t or total pressure difference p_t or specific work Y



- **for turbines:** total pressure ratio π_{tt} or actual power output P_D or total-to-static pressure ratio π_{ts}

(3) Number of revolutions n

Fluid/ Gas

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](#)^[20].

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/ Boundary 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 axial turbines. For pumps and ventilators 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.

For radial-inflow turbines the static pressure at the suction flange (pressure in the connection flange of the work piece attached to the turbine at the outlet) has to be specified instead of the total pressure at inlet.

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. **[not for axial turbines]**
- Pre-Swirl **[for pumps, ventilators, compressors only]**
Here you may define the inflow swirl at hub and shroud. The following definitions are available:

	Flow angle	Swirl number	Swirl energy number
	$\alpha = \arctan(c_m/c_u)$	$\delta_r = 1 - c_u/u$	$\delta_Y = uc_u/Y$
Positive swirl	$< 90^\circ$	$r < 1$	$Y > 0$
Negative swirl	$> 90^\circ$	$r > 1$	$Y < 0$
No swirl	$= 90^\circ$	$r = 1$	$Y = 0$

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:

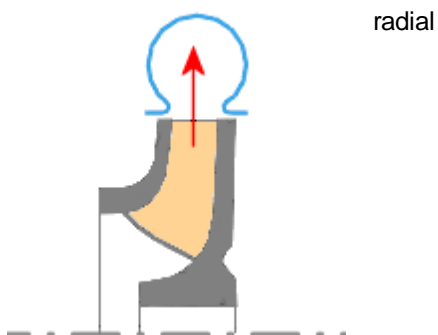
$$\delta_r = 1 - \frac{c_m}{u \tan \alpha} = 1 - \frac{4Q}{\pi^2 (d_S^2 - d_H^2) u \tan \alpha}$$

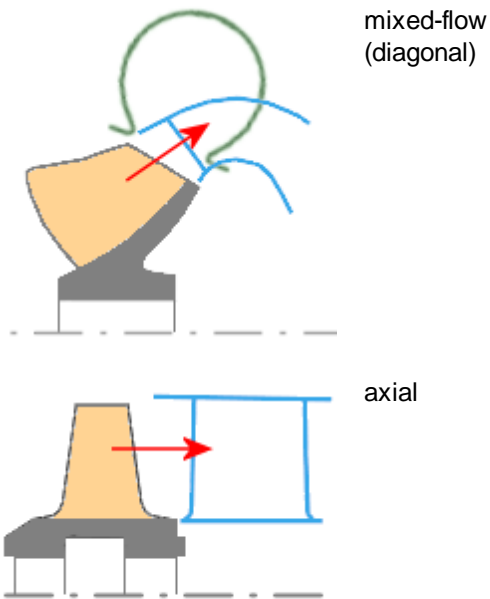
$$\delta_r = 1 - \frac{\delta_Y Y}{u^2} \quad \delta_Y = \frac{u^2 (1 - \delta_r)}{Y}$$

The conversion $\delta_r - \alpha$ is only valid for certain diameters d_H and d_S .

Information

Except for radial-inflow turbines the general meridional shape of the machine, depending on the specific speed, is displayed in the right **Information** area:





Furthermore some calculated variables are displayed:

Specific speed	points to machine type and general shape of impeller (see Specific speed ^[192] definitions)
Specific energy Y	<p>Pumps, Ventilators: $Y = gH = p_t / \rho$</p> <p>Compressors (perfect gas model): $Y = \left(\pi_t^{\frac{\kappa-1}{\kappa}} - 1 \right) c_p T_{t,s}$</p>
Power output P_Q	<p>$P_Q = \dot{m}Y$</p> <p>Pumps, Ventilators: $P_Q = \rho gHQ$</p>
Mass flow \dot{m}	<p>Pumps, Ventilators:</p> <p>Compressors:</p> <p>(density according to gas model)</p>

Total pressure difference Δp_t	Pumps, Ventilators: $\Delta p_t = \rho g H$ Compressors: $\Delta p_t = p_{t,2} - p_{t,S} = p_{t,S} \cdot (\pi_t - 1)$
--	--

Compressor:

Total pressure ratio	$\pi_t = p_{t,2} / p_{t,S}$
Inlet speed of sound (total)	$a_{t,1} = \sqrt{\kappa R Z T_{t,S}}$ (perfect gas model)
Volume flow (total)	$Q_{tS} = \frac{\dot{m}}{\rho_{tS}(p_{tS}, T_{tS})}$ (density according to gas model)
Inlet density (total)	$\rho_{tS} = \rho_{tS}(p_{tS}, T_{tS})$ (density according to gas model)
Outlet density (total)	$\rho_{t2} = \rho_{t2}(p_{t2}, T_{t2})$ (density according to gas model)
Outlet temperature (total)	$T_{t2} = T_{tS} \left(1 + \frac{\gamma}{c_p T_{tS}} \right)$ (perfect gas model)

Turbine:

Total speed of sound at inlet a_{t1}	$a_{t1} = \sqrt{\kappa \cdot R_{Gas} \cdot Z \cdot T_{t1}}$ (perfect gas model)
--	---

General remarks

- 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 an impeller for extremely low specific speed values ($nq < 10$). 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 ($nq > 400$) 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 nq - in parallel.
- Please note: CFturbo® is preferably used between **10 < nq < 400** – radial, mixed-flow and axial impellers.

Possible warnings

Problem	Possible solutions
Energy transmission of all impellers is different to globally defined value.	
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 Main dimensions ^[244]) to get altogether 100% of the initially defined value of the Global setup.

5.2.1.3 Performance prediction

? PROJECT | General | Performance prediction



The Performance prediction is an empirical based estimation of the performance map of the machine. Currently it is not available for axial turbines.

Two approaches are used to estimate the characteristics:

[Euler](#):^[96] Losses are estimated and subtracted from the theoretical Euler-line.

[Casey/Robinson](#):^[101] Characteristics will be derived from similarity considerations. For details see [Casey/Robinson](#)^[568]. 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:

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

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 radial impeller and the area averaged diameter for axial impeller respectively. The latter reads as:

$$d_{M2} = \sqrt{\frac{1}{2} (d_{S2}^2 + d_{H2}^2)}$$

Variables

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

Variable	Pump	Ventilator	Compressor	Turbine
H	head	-	-	-
p	pressure difference (total-total)			
p _{ts}	-	pressure difference (total-static)	-	
		$\Delta p_{ts} = \Delta p - \frac{\rho}{2} c_2^2$		
ψ	work coefficient			
H/H _{opt}	head ratio	-	-	-
p/ p _{opt}	total pressure difference ratio			

tt	-	-	pressure ratio (total-total)
ts	-	-	pressure ratio (total-static)
St	stage efficiency		
St*	stage efficiency incl. motor		-
v	volumetric efficiency		-
P	required driving power		-
	$P = \frac{\rho \cdot g \cdot H_{\text{Decr}}}{\eta_{\text{mech}} \eta_{\text{mot}} \eta_{\text{sf}}} \cdot (Q + Q_{\text{leak}})$	$P = \frac{Y_{\text{Decr}} \cdot \rho}{\eta_{\text{mech}} \eta_{\text{mot}}} \cdot (Q + Q_{\text{leak}})$	
Q	volume flow		
	$Q = \frac{\dot{m}}{\rho}$	$Q = \frac{\dot{m}}{\rho_2}$	$Q = \frac{\dot{m}}{\rho_1}$
φ_m	meridional flow coefficient		
	$\varphi_m = \frac{c_{m2}}{u_2}$		$\varphi_m = \frac{c_{m1}}{u_1}$
Q/Q _{opt}	flow ratio		
Q _t	-	-	volume flow total
	-	-	mass flow

\dot{m}_{red}	-	-	<p>reduced mass flow</p> $\dot{m}_{\text{red}} = \dot{m} \frac{\sqrt{T_{\text{Ref}}}}{p_{\text{Ref}}}$
\dot{m}_{corr}	-	-	<p>corrected mass flow</p> $\dot{m}_{\text{corr}} = \dot{m} \frac{\sqrt{T_{01}/T_{\text{Ref}}}}{p_{01}/p_{\text{Ref}}}$

All combinations of flow and energy variables are possible.


It is common practice in the case of 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

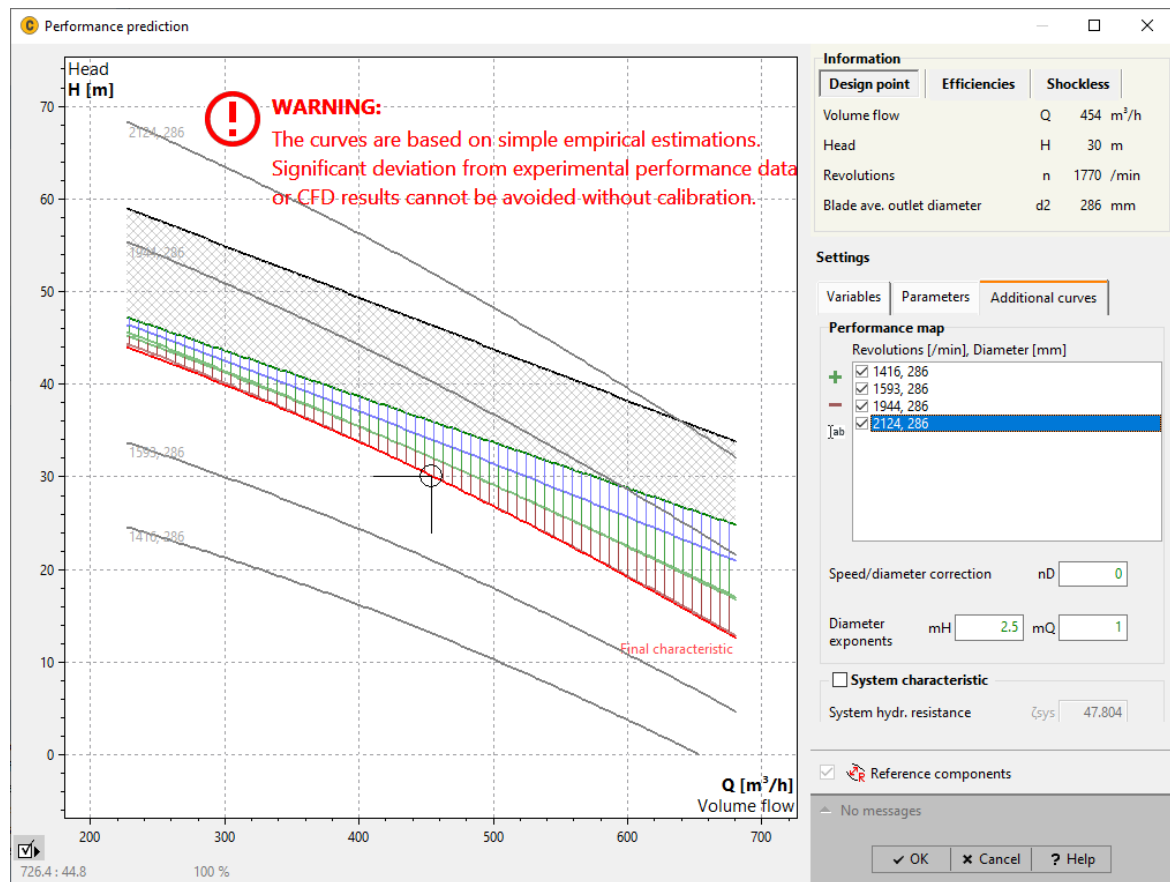
For comparison purposes with the present design saved designs can be loaded (soft button "configure").

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).

5.2.1.3.1 Euler based

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:


Kind	Description	Parameter
Decrease d power	Based on the Euler-Equation and the decreased power that is calculated in the Blade properties ^[37] . In the design point the decreased power line is shifted by a pressure head loss equivalent to the decreased power ($H_{Decr} = H_{th} - H_{Decr}$). The decreased power line can be parallel to the	<p>cl:</p> <p>cl = 1...parallel position,</p> <p>cl = 0...intersection with Euler-Line at $p = 0$.</p>

	Euler-Line as well as positioned that way, that it intersects the Euler-Line at $p = 0$.		
Hydraulic losses	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 $Q = 0$.	ζ_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)
Shock losses (by turbulence and separation)	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.	ζ_s : General approach: $\Delta H_{Shock} = \zeta_s \cdot F \cdot (Q - Q_{opt})^2$	$F = \frac{100}{g \cdot 0.5 \cdot (A_{in} + A_{out})}$

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 " 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 ζ_h/ζ_s can be set in the panel Parameter, see table below, second column.

Settings

Energy and flow rate variables plus flow rate limits (reset default flow rate with )

Coefficients influencing the decreased power (cl) and the hydraulic as well as shock losses (ζ_h , ζ_s)

Additional curves with different speeds and diameter plus system characteristic

The image displays three panels from the CFturbo 10 software interface:

- Settings - Axis definition:** Shows 'x' as 'Volume flow' and 'y' as 'Total pressure diff.'. A 'Flow rate rel. to design point' is set to 50%.
- Settings - Decreased output:** Shows a 'Radial Impeller' with a value of 0.85621 and a 'Decreased power alignment cl' of 0.5.
- Settings - Loss coefficients:** Includes a 'Calculate ζ' button, an 'Automatic' checkbox, and a table for loss parameters.

	ζ _h	ζ _w	c _w
Pipe in	0.014731	0	
Radial Impeller	0.019392	0.012948	
Volute	0.018493	0.012347	
- Settings - Performance map:** Shows a list of 'Revolutions [/min], Diameter [mm]' with values 1416, 286; 1593, 286; 1944, 286; and 2124, 286. The 'Speed/diameter correction' nD is 0. 'Diameter exponents' mH is 2.5 and mQ is 1. 'System characteristic' is checked, with 'System hydr. resistance' ζ_{sys} at 47.804 and 'Static part' p_{stat} at 0 bar.

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_{Blind} due to recirculation at a flow of $Q = 0$. This blind flow Q_{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 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 ventilators, 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 exists, 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:

$$\dot{m}_{ch} = A \cdot a_c \cdot \rho_c \cdot (1 - B)$$

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 nD , see table [Settings](#)^[97], last column. The resulting losses will be:

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

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: $H' = H (d'/d)^{m_H}$ and $Q' = Q (d'/d)^{m_Q}$. The exponent m_H should be within 2..3, m_Q should be 1 or slightly bigger.

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

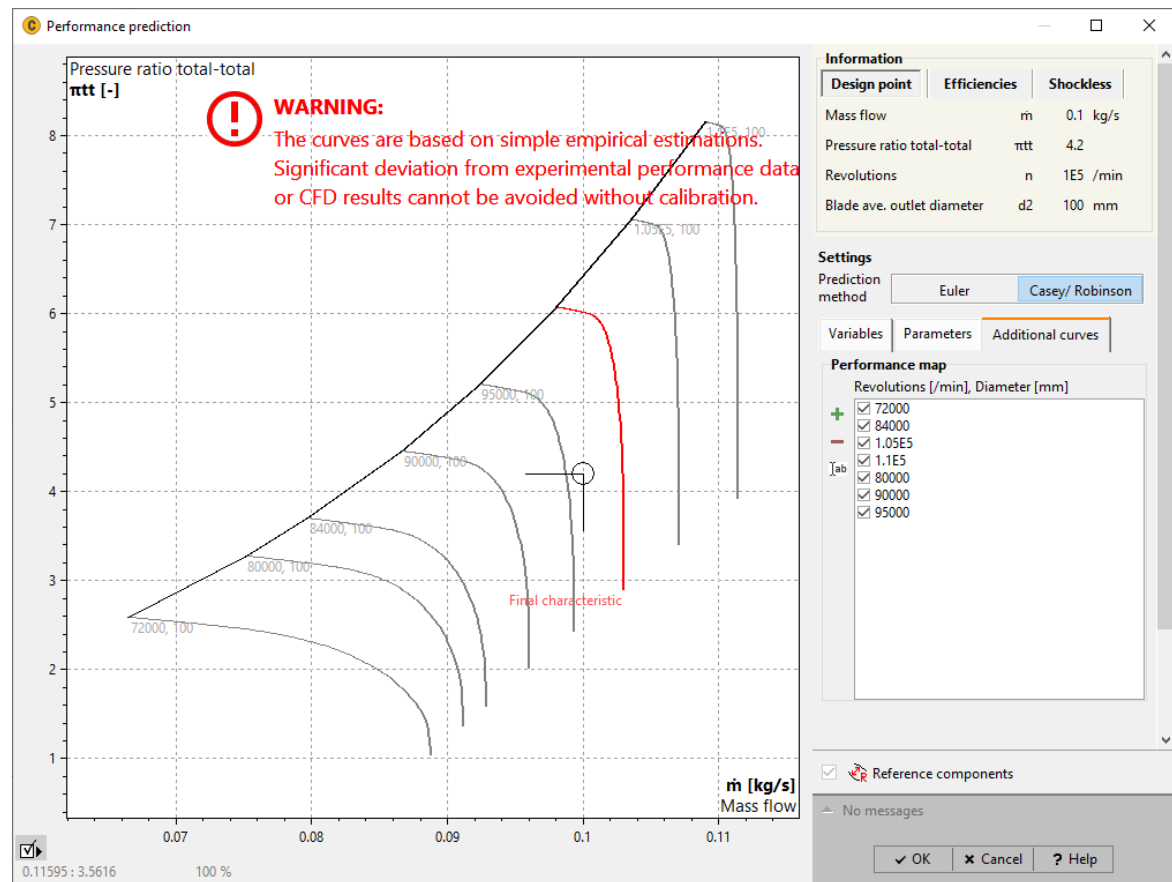
$$\text{Loss}(D) = \text{Loss}(D_{\text{Design}}) \left[1 - nD \cdot \left(1 - \frac{D}{D_{\text{Design}}} \right)^2 \right]$$

System characteristic - pumps, ventilators 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.

5.2.1.3.2 Casey/Robinson

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

Settings

VariablesParametersAdditional curves

Axis definition

x Volume flow
y Total pressure diff.

Flow rate rel. to design point
50 % ... 150 %

Settings

Prediction method EulerCasey/ Robinson

VariablesParametersAdditional curves

Disc friction

Disk friction factor kdf 0.005

Characteristics width

A 0.5 B 1 C 4.5

Surge location

As 0.5 Bs 1.1 Cs 4.5

Coefficients

	low	high
$D\eta$	2	1.7
$G\eta$	2	0.3
$H\eta$	2	3.5
φ_d/φ_c	0.5	1
φ_s/φ_c	0.5	1

Settings

Prediction method EulerCasey/ Robinson

VariablesParametersAdditional curves

Performance map

Revolutions [/min], Diameter [mm]

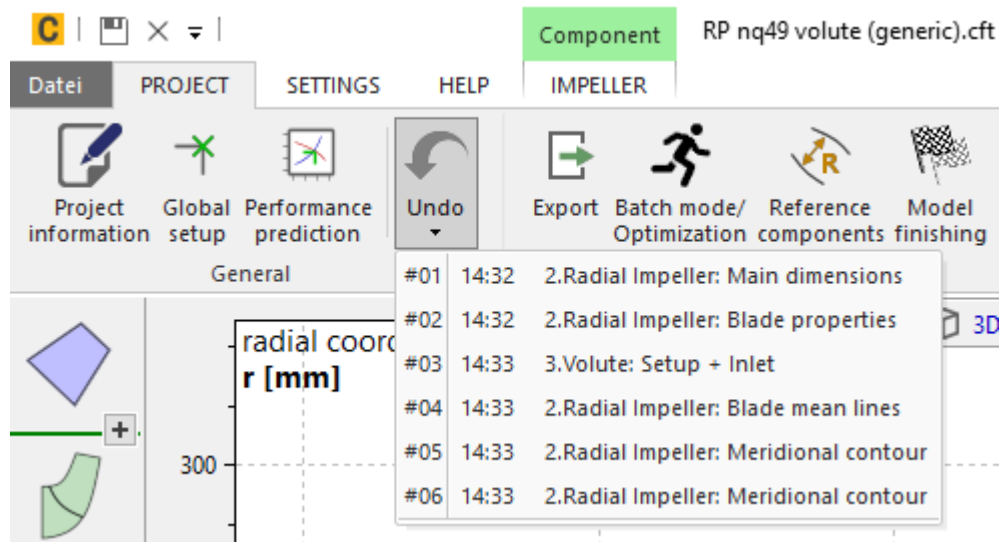
+ 72000
85000
1.05E5
1.1E5
95000
90000
80000

5.2.1.4 Undo

? PROJECT | General | Undo 

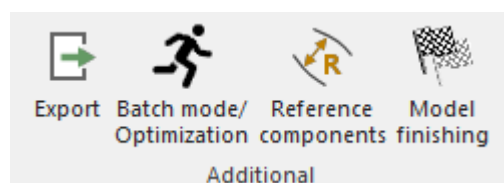
The design history can be opened by clicking the undo-button. It contains all modifications from opening of the project or session in chronological order.

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](#) ¹⁰³

→ [Batch mode/ Optimization](#) ¹⁷⁶

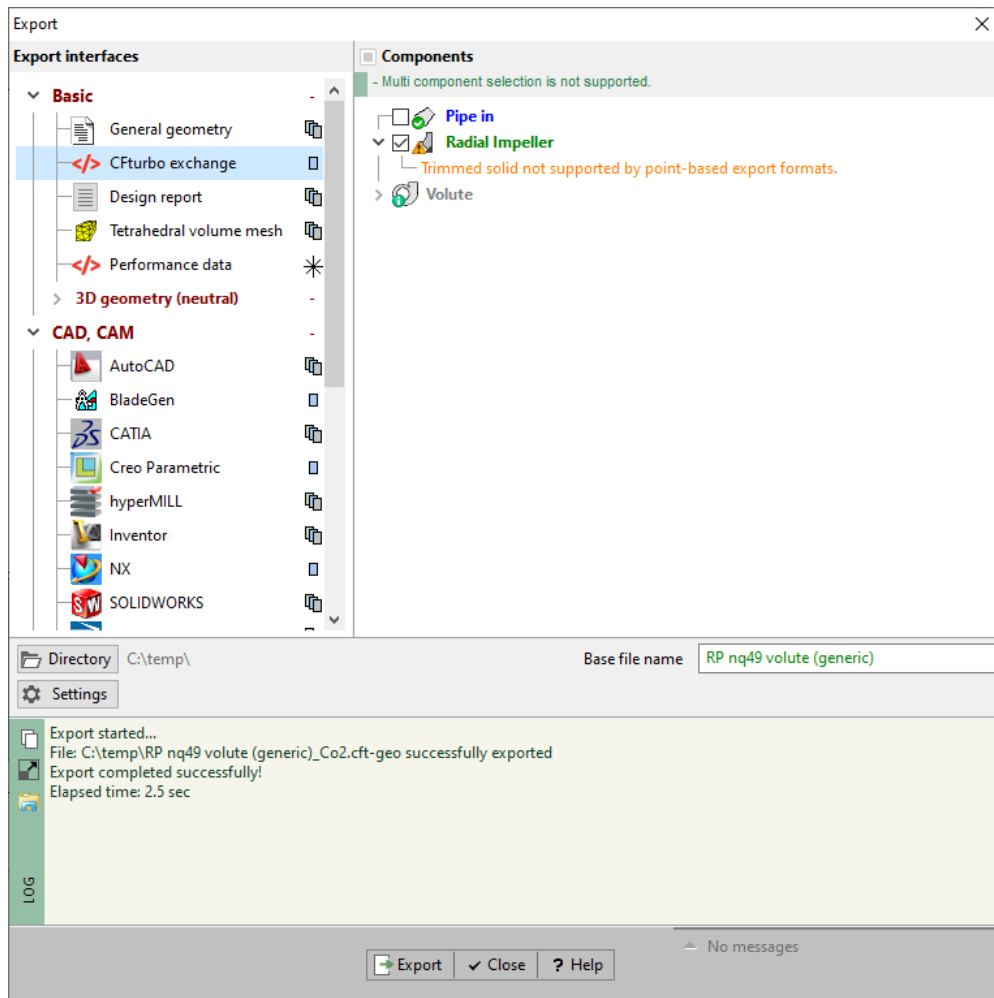
→ [Reference components](#) ¹⁸¹

→ [Model finishing](#) ¹⁸⁴

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

Interfaces

Available interfaces are grouped into three blocks: [Basic](#)^[111], [CAD/ CAM](#)^[114] and [CFD](#)^[117].

Generally, there are 3 types of export formats available: "3D model export", "Predefined 3D model export" and "Point based export":

	3D model export	Predefined 3D model export	Point based export
Format	IGES, STEP, STL, Parasolid, BREP	ANSA, CCM+, ICEM-CFD, Pointwise, SimericsMP, SimericsMP+, Tetrahedral volume mesh	All the rest
Content	all visible parts of the 3D model	predefined set of parts of the 3D model	predefined set of points/splines (independent of the 3D model)
Point density	variable ¹⁾	variable ¹⁾	variable ²⁾
Units	[mm]	[mm]	variable ²⁾

¹⁾ Point density can be configured in the **Model settings/ 3D model** of each component ([Impeller](#)⁴⁸⁶, [Stator](#)⁵¹¹, [Volute](#)⁵⁶³).

²⁾ Point density and export unit can be configured in the **Model settings/ Point export** of each component ([Impeller](#)⁴⁸⁶, [Stator](#)⁵¹¹, [Volute](#)⁵⁶³).

If the [blade shape](#)³⁷¹ is ruled surface then points of mean lines as well as profiles (pressure and suction side) are not affected by the [model settings](#)⁴⁸⁶ for the point based export.

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

Remarks about the 3D model export

It is recommended to export solids or solid faces if they are available, because then the individual faces best fit to each other. Particularly, this is the only sensible option after 'solid trimming' has been done during [Model finishing](#)⁴⁸⁷.

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.

Export

Above the log area some settings for the selected export interface can be specified, like export destination and the base name of exported files. Additional parameters can be available depending on the selected interface.

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

	Problem	Possible solutions
CFD setup	Segment required (see CFD setup).	
	CFD setup not accomplished.	Execute CFD setup ⁴⁷⁸ (generates a segment).
	Blade tip projection to casing recommended (see CFD setup).	
	Blade tip projection not accomplished.	Check "Blade projection" in CFD setup ⁴⁷⁸ .
	Gap between leading/trailing edge and inlet/outlet required. Select stator on inlet/outlet side. Alternatively, the CFD extension can be activated (see CFD setup).	
	Some space around blade edges is required for meshing. This can be generated by	Try to increase the distance between leading/ trailing edge and meridional inlet/ outlet by

	Problem	Possible solutions
	<p>creating a CFD extension or by selecting a neighbouring stator component.</p> <p>Note for TurboGrid: a vaneless stator has to be selected, which has to be considered as part of the rotating domain in TurboGrid.</p>	<p>a) moving leading/ trailing edge in meridional contour if edge is not fixed on inlet/ outlet ³⁵⁶.</p> <p>b) selecting a neighbouring stator if possible.</p> <p>or</p> <p>c) activating CFD-Extension in CFD setup/ Extension ⁴⁷⁹.</p>
	<p>Gap between leading/trailing edge and inlet/outlet recommended.</p> <p>CFD extension can be activated (see CFD setup).</p>	
	Some space around blade edges is recommended.	<p>Try to increase the distance between leading/ trailing edge and meridional inlet/ outlet by</p> <p>a) moving leading/ trailing edge in meridional contour if edge is not fixed on inlet/ outlet ³⁵⁶.</p> <p>or</p> <p>b) activating CFD-Extension in CFD setup/ Extension ⁴⁷⁹.</p>
	<p>Small gap between blade/leading edge and inlet/outlet may cause problems.</p> <p>Increase it in case of import problems.</p>	
	See message.	<p>Try to increase the distance between leading/ trailing edge and meridional inlet/ outlet by</p> <p>a) moving leading/ trailing edge in meridional contour if edge is not fixed on inlet/ outlet ³⁵⁶.</p> <p>or</p> <p>b) activating CFD-Extension in CFD setup/ Extension ⁴⁷⁹ (only for impellers).</p>
Finishing	Trimmed solid required (see Model finishing).	
	Up-to-date trimmed solids required.	Execute Model finishing ⁴⁸⁷ with option "Solid trimming".

	Problem	Possible solutions
	Extended blade not supported (see Model finishing).	
	See message.	Execute Model finishing ^[487] with option "No model finishing" or "Solid trimming".
	Model finishing not up-to-date.	
	See message.	Execute Model finishing ^[487] .
	Model finishing not yet selected. "Solid trimming" is recommended.	
	See message.	Execute Model finishing ^[487] with option "Solid trimming".
	Trimmed solid not supported by point-based export formats.	
	See message.	-
	Beware: "Solid" and "Solid faces" are handled differently in various target systems.	
	To be taken into account if a mixed selection of solids and solid faces was selected in the 3D model tree ^[234] .	-
Blades	Blades required (see Main dimensions).	
	Components without blades are no supported by this interface.	-
	Single blade designs not supported.	
	See message.	-
	Wrap angles larger than 360° not supported.	
	See message.	-
	Blades with thickness definition "perpendicular to mean surface" are not supported.	
	See message.	Choose another blade thickness definition method

	Problem	Possible solutions
		(see thickness definition ⁴³⁹ in Blade Profiles).
Volute	"Fillet-Cut-water" feature not supported for point based export formats.	
	See message.	-
	"Flow domain" export may not work.	
	The STEP export of "Flow domain.Solid" or "Flow domain.Solid faces.Spiral" might be defective if the spiral face spans a wrap angle of 360°. This occurs for internal volutes.	Select "Spiral.Surface" instead in the 3D model tree ²³⁴ .
	Volutes without cut-water not supported.	
	See message. Note for ICEM-CFD : geometry can be exported but cannot be meshed using CFturbo2ICEM.	-
Model settings	Model geometry does not fit a cube of size (-500, -500, -500) to (500, 500, 500). Choose another export unit to scale geometry appropriately.	
	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.
	Current point export settings may cause problems in Autodesk Inventor due to high number of points.	
	See message.	Change number of points in Model settings/Point export ⁴⁸⁶ .
General	Completion of all design steps required.	

	Problem	Possible solutions
	Only for CFD-Applications. One or more design steps were not finished.	Complete all design steps.
	Special license for this interface required.	
	License for this interface not found.	Check the license information in SETTINGS/ Licensing ¹⁸⁵ .
	Export not possible due to license restrictions.	
	The corresponding module is not licensed or CFturbo is running with a trial license.	Only designs corresponding with licensed modules or unmodified default examples using a trial license can be exported.
	Material domain is not available	
	Components without material domain are not supported by this interface.	Add material domain, see Hub/Shroud solids ³⁶⁰ .
	Model state contains no geometry for export.	
	See message.	Select via "Set parameters" a proper model state containing desired parts to be exported. Imports can only be exported via the context menu of the 3D model tree.
	Meshing process ran out of memory.	
	A too fine-grained mesh was produced. This resulted in exceeding virtual memory.	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.
	Performance prediction not possible for axial turbine designs.	
	See message.	-

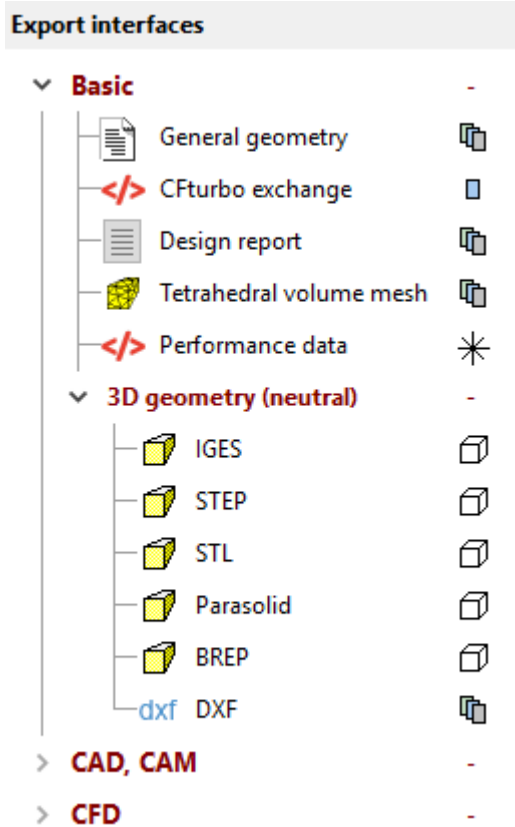
	Problem	Possible solutions
	Performance prediction not possible for projects without impellers.	
	See message.	-
	Invalid viscosity value.	
	See message.	Set a valid viscosity value in fluid manager ^[201] .
	Some objects didn't get meshed properly: [Face names]	
	The exported mesh is not watertight due to some invalid areas inside the mesh. These areas are equivalent to the specified faces in the warning message.	Check the specified faces in the 3D-model due to invalid geometry. Vary the meshing parameters.
	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.	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.
	Real gas properties ignored by default. Configure *.rgp file manually.	
Only for Vista TF. See message.	-	

5.2.2.1.1 Basic

? PROJECT | Export | Basic



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





Export preconditions

Export availability is independent of the design progress.

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

Menu item	Description		Component type			
Design report	*.html, *.rtf, *.csv, *.txt	design report	I	S	V	MC
	Design information as text file; Summary of most important design parameters see Report ^[240]					
Performance data	*.cft-pp	XML file	The project is always exported in its entirety			
	File contains results of Performance prediction ^[92] . Two curves are exported: p versus ṁ and s _t versus ? . Units are those in accordance to preferences ^[190] .					

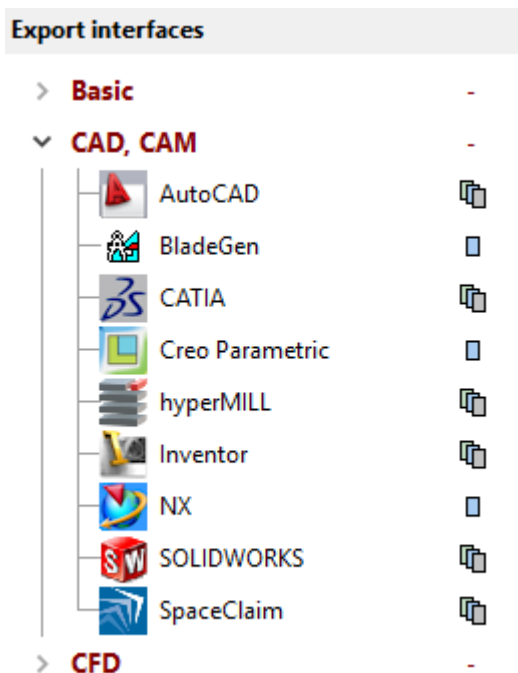
General geometry	*.geo-txt *.geo-xml	plain text file or XML file	I	S	V	MC
	File contains detailed geometry data of the design for any further processing. Impeller/stator: Meridional section: z, r of hub, shroud, leading edge Blade mean lines, Blade profiles: x, y, z: cartesian coordinates r: radius t: 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) : blade angle s: blade thickness L: 3D length la: lean angle Volute: Spiral cross sections, Diffuser cross sections: x, y, z (cartesian coordinates) Contour lines in circumferential direction: x, y (cartesian coordinates)					
CFturbo exchange	*.cft-geo	XML file	I	S	V	MC
	File contains geometry data that can be re-imported in CFturbo (component data exchange format) see Add component ^[41]					
IGES	*.igs	neutral format (Initial Graphics Exchange Specification)	Exported geometry is selected via model state			
	File contains designed geometry as surface model.					
STEP	*.stp	neutral format (Standard for the Exchange of Product model data)				
	File contains designed geometry as volume model. Solid vs. Solid faces: They are handled differently by various target systems. In case of import problems, it is advisable to try the other variant as well. <u>Specifics:</u> For <i>SOLIDWORKS</i> , try with and without STEP import option: "B-REP mapping".					

STL 	*.stl	neutral format (Standard Triangulation Language)				
	File contains designed geometry as triangulated surface model.					
Parasolid	*.x_b, *.x_t	neutral format				
	File contains designed geometry as volume model. Available as binary (*.x_b) or ASCII file format (*.x_t).					
BREP	*.brep	native format of Open CASCADE based applications (Boundary Representation)				
	File contains designed geometry as volume model.					
DXF	*.dxf	neutral format (Drawing Interchange File Format)	I	S	V	MC
	File contains designed geometry of the selected component as 3D polylines.					
Tetrahedral volume mesh 	*.msh, *.vol, polyMesh	available file formats: Fluent, Netgen, OpenFOAM, Abaqus	I	S	V	MC
File or folder contains designed geometry as tetrahedral volume mesh for simulation.						

5.2.2.1.2 CAD, CAM

? PROJECT | Export | CAD, CAM

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




Export preconditions

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

Component type	Export available from design step
Impeller, stator with blades	"Mean lines"
Stator without blades	"Meridional contour"
Volute	"Spiral development areas"

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

Menu entry	Description	Component type			
AutoCAD 	*-.txt	I	S	V	MC
	Lisp script <i>xyz2spline</i> (part of CFturbo) creates splines from imported points. <ul style="list-style-type: none"> Select "AutoCAD Classic" Workspace Load "xyz2spline.lsp" under Manage Load Application Run command "xyz2spline" and select *.txt file 				
BladeGen	*.rtzt	I	S	V	MC

	<p>The file contains complete 3D impeller geometry point-by-point.</p> <ul style="list-style-type: none"> • File Open: select file type „Meanline File (*.rtzt)“ • select *.rtzt file 				
CATIA ¹³³	*.catvbs	I	S	V	MC
	<p>The macro generates a surface model + generating splines.</p> <ul style="list-style-type: none"> • Tools Macro Macros • Select macro library and macro, Run 				
Creo Parametric ¹³⁴	*.ibl, *.pts	I	S	V	MC
	<p>*.ibl contains geometry defined by 3D points.</p> <p>*.pts files are exported for impellers only and contain information about blade thickness defined by 2D points</p> <ul style="list-style-type: none"> • Home New Part <name> (if no file is open) • Model Get data Import • select *.ibl or *.pts file 				
hyperMILL	*.stp	I	S	V	MC
	<ul style="list-style-type: none"> • File Open 				
Inventor ¹⁵⁰	*.bas	I	S	V	MC
	<p>The macro generates a surface model + generating splines.</p> <ul style="list-style-type: none"> • Tools Visual Basic Editor • VB <ul style="list-style-type: none"> ◦ File New project ◦ File Import file, select *.bas ◦ Tools Macro, select “Main”, Run 				
NX	*.dat	I	S	V	MC
	<p>Some files per component are created.</p> <ul style="list-style-type: none"> • New New Project file <name> (if no file is open) • Application Modeling <p>To import curves (hub, shroud, material domain, secondary flow path and volute contour curves):</p> <ul style="list-style-type: none"> • File Import Points from file • select *.dat file • Use "fit curve" to make a spline through desired points <p>To generate surfaces (blade, volute, diffuser):</p> <ul style="list-style-type: none"> • Insert surface Through points • Row degree <= number of blade profile sections • Column degree <= Row degree-1 • Points from file • select *.dat file 				

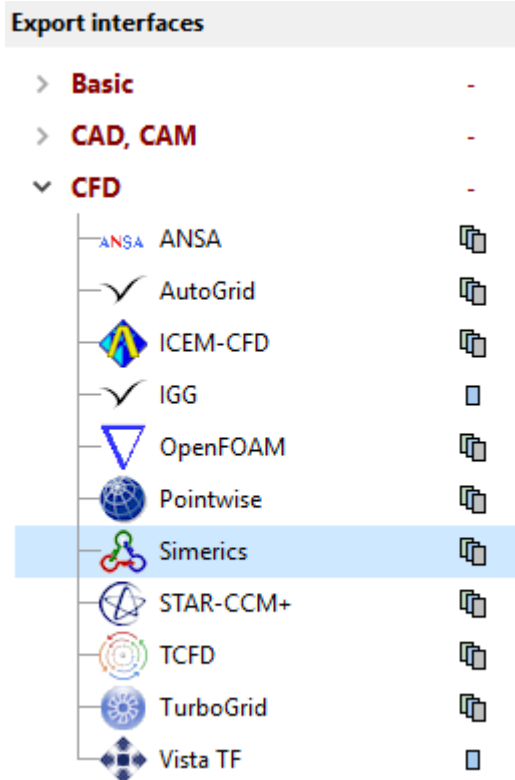
	Please note: If the mentioned menu options are not available, the appropriate commands have to be created: a) "Tools/Customize" or right click on any toolbar/menu, "Customize..." b) "Commands", "Insert/Curve/Spline..." or "Insert/Surface/Through Points..." c) Integrate selected item via Drag and Drop in a menu or toolbar				
SOLIDWORKS	*.swb	I	S	V	MC
	The macro generates a surface model + generating splines. • Tools Macro Run: select *.swb				
SpaceClaim <small>153</small>	*.stp				
	File contains designed geometry as volume model. • File Open: select *.stp file				

5.2.2.1.3 CFD

? PROJECT | Export | 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.


Component type	Export available from design step
Impeller, stator with blades	"Blade edges"
Stator without blades	"Meridional contour"
Volute	"Diffuser geometry"

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












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







Menu entry	Description	Component type			
ANSA	*.igs	I	S	V	MC

	<ul style="list-style-type: none"> File Open Select *.igs file 				
AutoGrid  159	*.geomTurbo	I	S	V	MC
	<ul style="list-style-type: none"> File New Project "Initialize a New Project from a geomTurbo File" Select *.geomTurbo file 				
ICEM-CFD  162	*.tinXML, *.stp	I	S	V	MC
	A STEP file with named geometries is created. The names are visible in ICEM-CFD if the file is imported via <i>Workbench Reader</i> . Parameters are saved in a separate XML file.				
IGG	*.dat	I	S	V	MC
	<p>Multiple data files are generated: section.dat, diffusor.dat, curves.dat</p> <ul style="list-style-type: none"> File Import IGG Data Select *.dat file Repeat steps for remaining files 				
Pointwise	*.step	I	S	V	MC
	<ul style="list-style-type: none"> File Import Database Select *.step file 				
SimericsMP  165	*.spro, *.stl	I	S	V	MC
SimericsMP+  165	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 Open project				
STAR-CCM+	*.stp	I	S	V	MC
	<ul style="list-style-type: none"> File Import Import Surface Mesh... Select *.stp file 				
TurboGrid  163	*.curve	I	S	V	MC
	<p>4 files are created, a session file (<filename>.tse) and <filename>_hub.curve, <filename>_shroud.curve, <filename>_profile.curve.</p> <p>Load the saved session file <filename>.tse:</p> <ul style="list-style-type: none"> File New Case Session Play Session <p>or</p> <p>Open the curve files (<filename>_hub.curve, <filename>_shroud.curve, <filename>_profile.curve) manually:</p>				

	<ul style="list-style-type: none"> • Launcher: select directory, start ANSYS TG • File New Case • File Load Curves • input number of blades, define z axis as rotational axis, select cartesian coordinate system and length unit, select *.curve file 				
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.tcfcd -allrun & More info at CFD support website	I	S	V	MC
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.	I	S	V	MC

Possible warnings

Problem	Possible solutions
Inlet CFD-interface does not match previous component.	
  Impeller - Impeller   Impeller - Stator   Impeller - Volute   Stator - Impeller	If possible, activate the RSI connection in CFD setup of the impeller  .
  Stator - Stator	Inlet geometry was defined in such a way that it does not match the inlet of the previous

Problem	Possible solutions
  Stator - Volute	component. See geometric coupling ^[42] for more details.
Outlet CFD-interface does not match next component.	
  Stator - Stator	Outlet geometry was defined in such a way that it does not match the outlet of the next component. See geometric coupling ^[42] for more details.
  Stator - Volute	

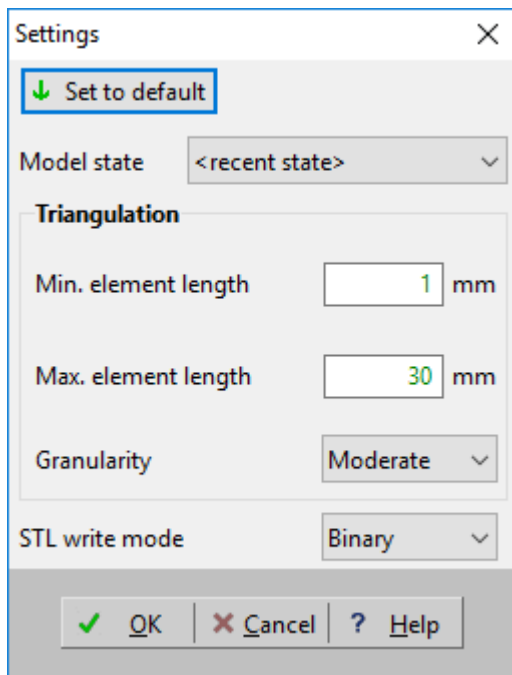
5.2.2.1.4 Specifics

The following topics contain specific information about how to import the geometry designed by CFturbo into some CAE applications:

- [STL](#)^[122]
- [Tetrahedral volume mesh](#)^[123]
- [AutoCAD \(Autodesk\)](#)^[124]
- [BladeGen \(ANSYS\)](#)^[131]
- [CATIA \(Dassault Systèmes\)](#)^[133]
- [Creo Parametric \(PTC\)](#)^[134]
- [Inventor \(Autodesk\)](#)^[150]
- [SpaceClaim \(ANSYS\)](#)^[153]
- [AutoGrid \(NUMECA\)](#)^[159]
- [ICEM-CFD \(ANSYS\)](#)^[162]
- [TurboGrid \(ANSYS\)](#)^[163]
- [SimericsMP/ SimericsMP+ \(Simerics\)](#)^[165]
- [TCFD \(CFD Support\)](#)^[175]

5.2.2.1.4.1 STL

Some parameters are available via **"Set parameters"** to influence the quality / resolution of the STL geometry.



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.

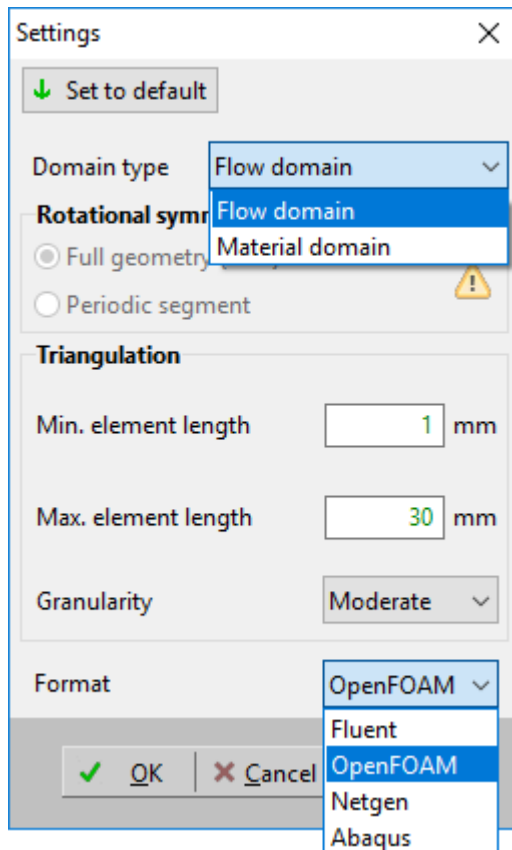
STL write mode: Format (Binary / ASCII) for writing STL files.

Possible warnings

Problem	Possible solution
Model state contains no surfaces and no solids.	
See message.	STL files describe triangulated surfaces. Points and curves cannot be represented. Select via "Set parameters" a proper model state containing surfaces or solids to be exported.

5.2.2.1.4.2 Tetrahedral volume mesh

In addition to the [parameters for triangulation](#)^[122], the following parameters are available:



Flow vs. Material domain

The user can select between:

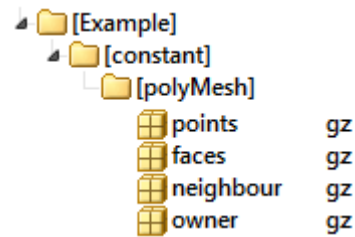
- Flow domain: for CFD purposes, flow domain watertight surfaces are exported
- Material domain: for FEM purposes, material domain watertight surfaces are exported

Export Format

Three export formats can be selected:

Fluent: *.msh file is exported

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



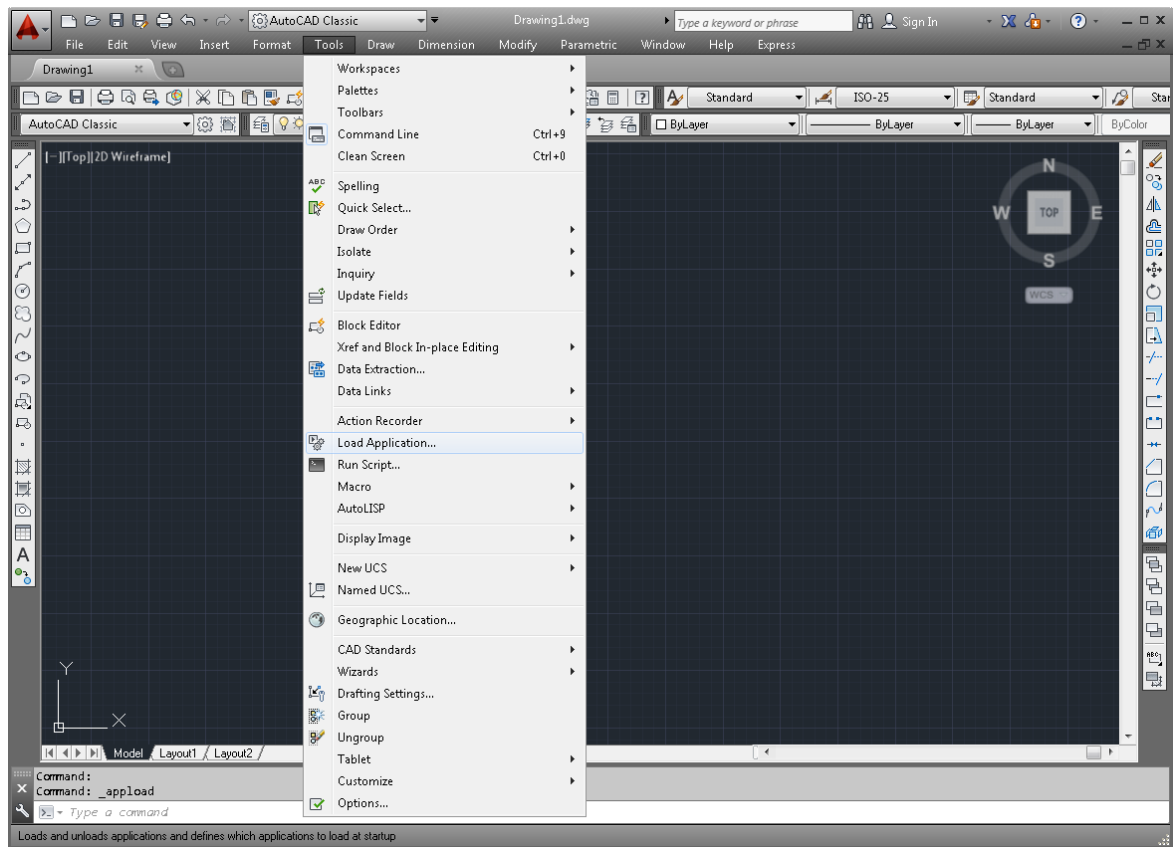
Netgen: *.vol file is exported

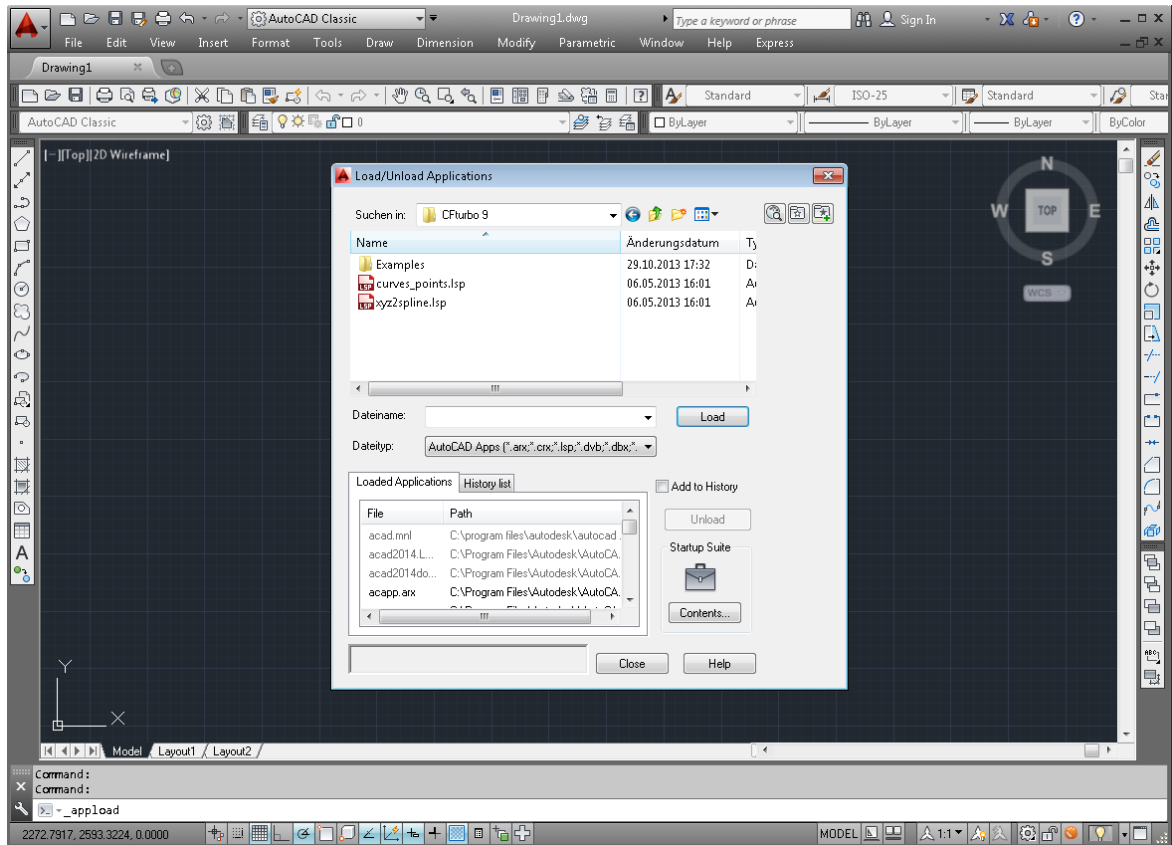
5.2.2.1.4.3 AutoCAD (Autodesk)

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.



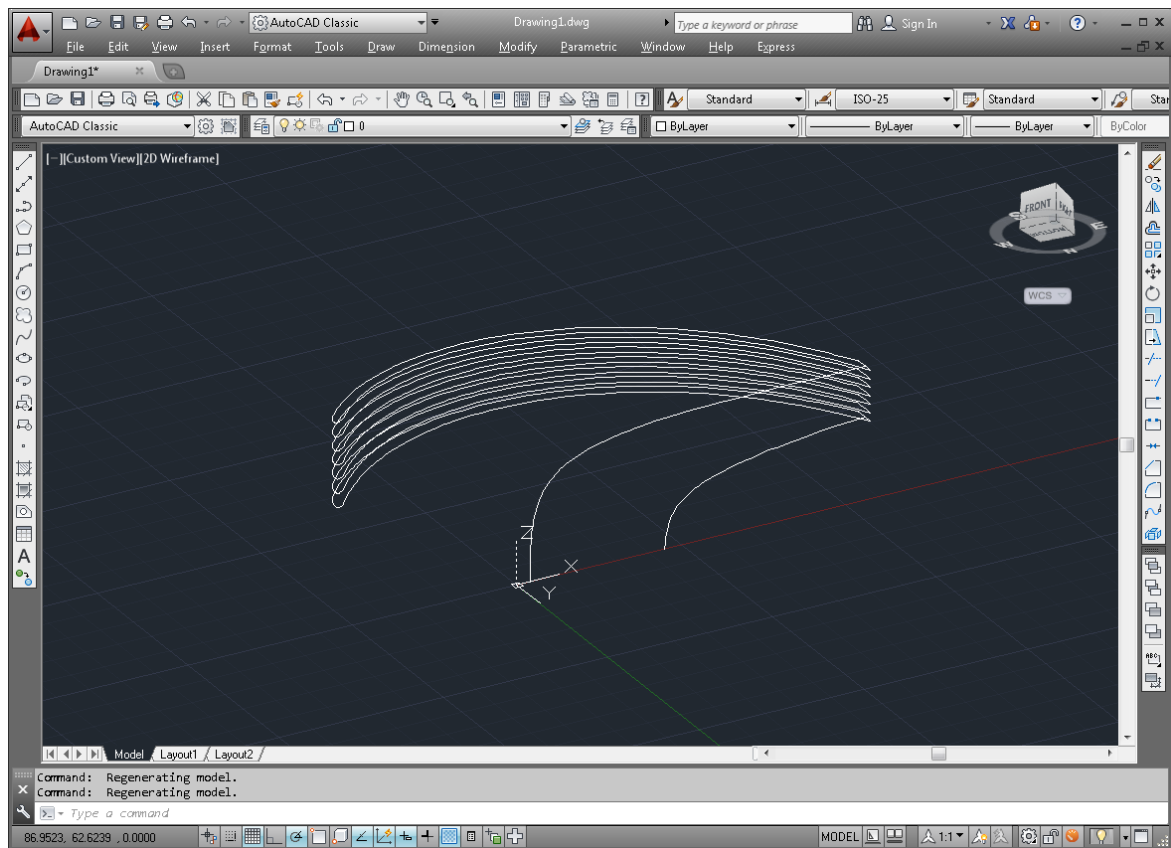


Selection of xyz2spline.lsp file

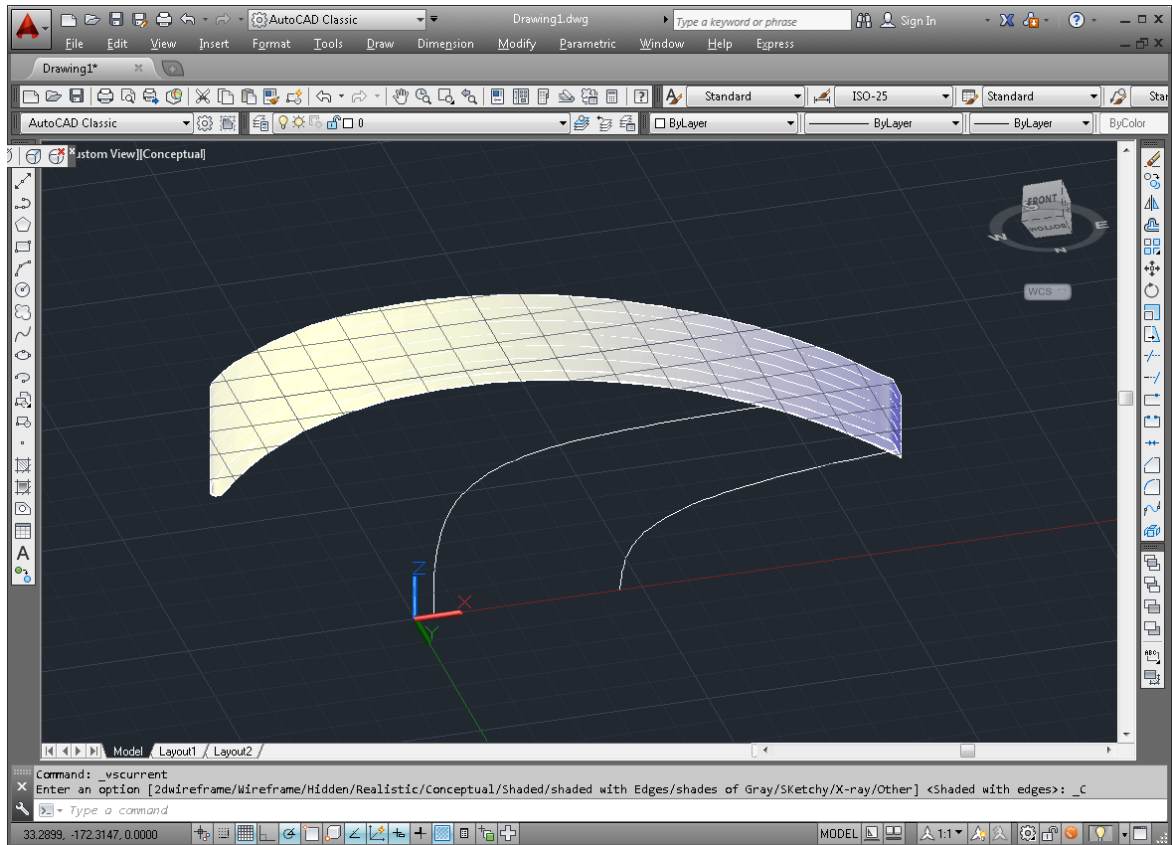
Construction of Impeller

Creating the blades

- Use the command `_loft` to create surfaces from curves



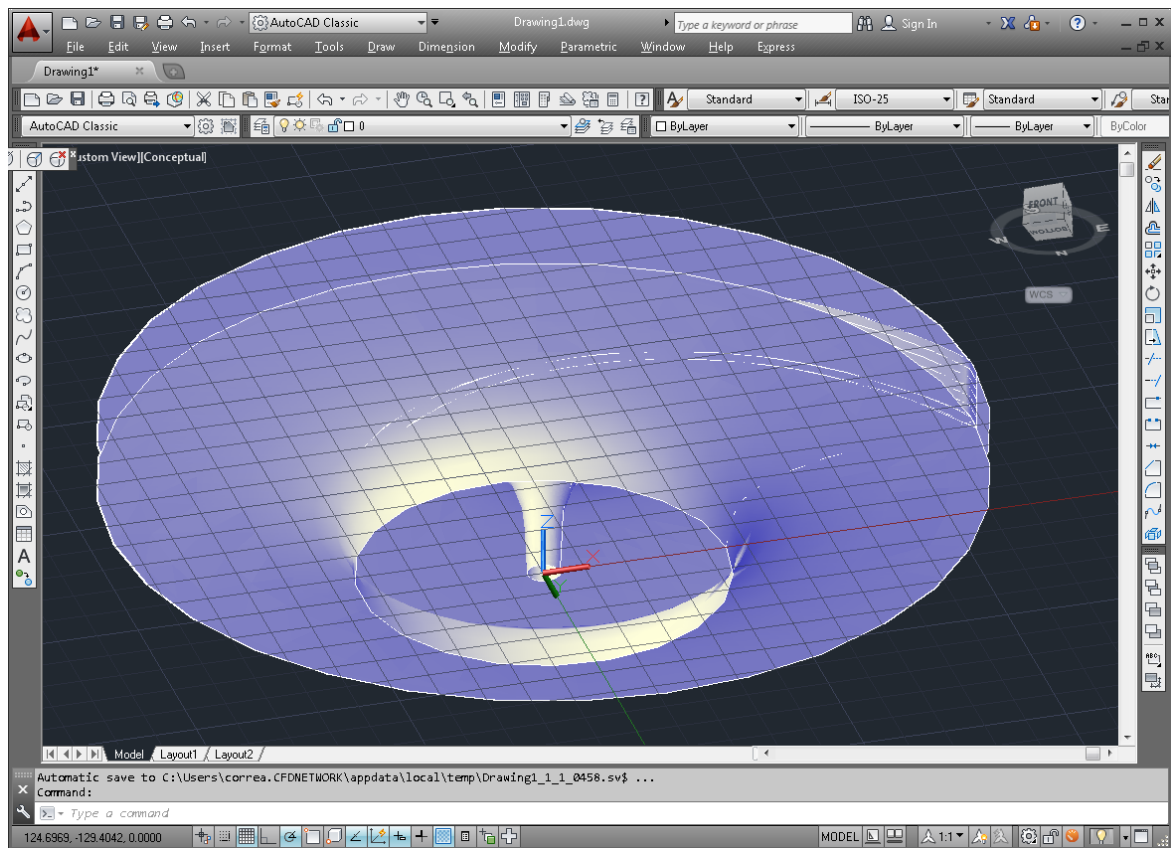
Sample-view after data import



Blade surface generated by using the _loft command

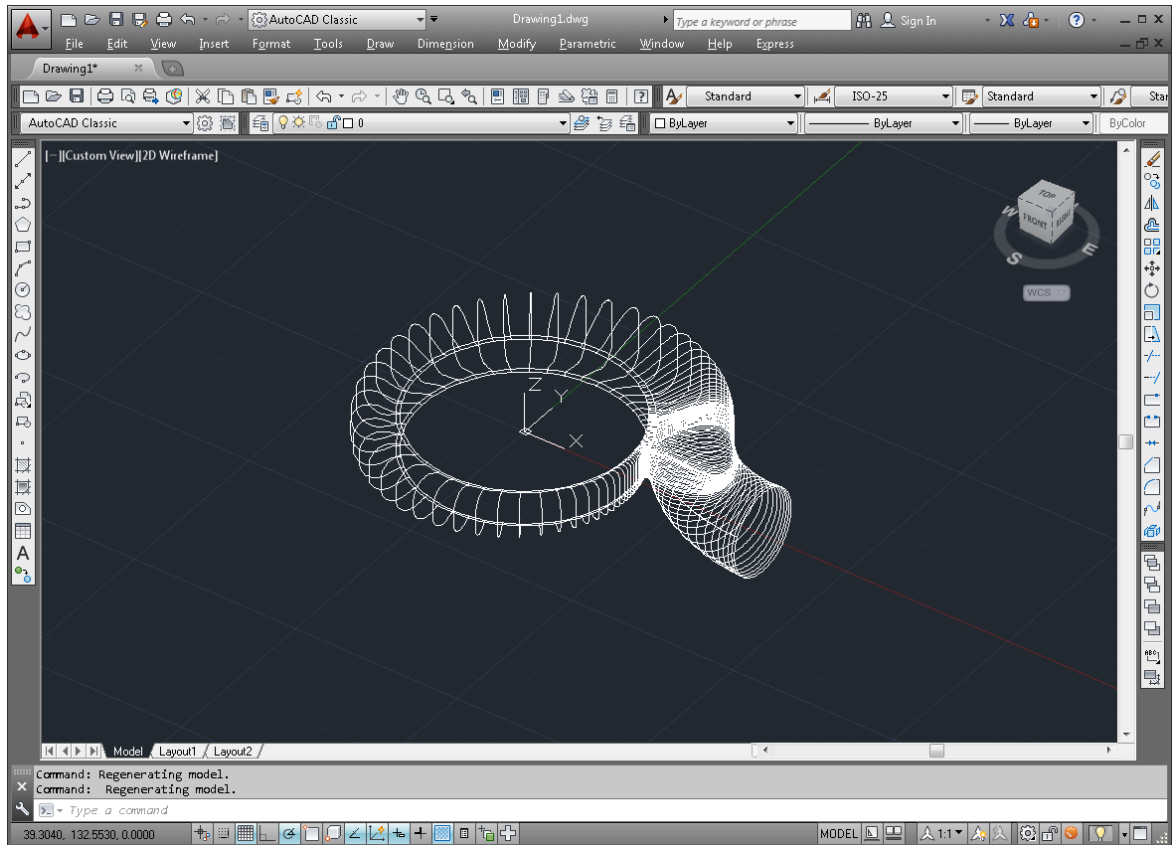
Creating rotational surfaces (Hub, Shroud)

- 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 [SStart angle/Reverse/EExpression] <360>:360



Hub and Shroud surfaces

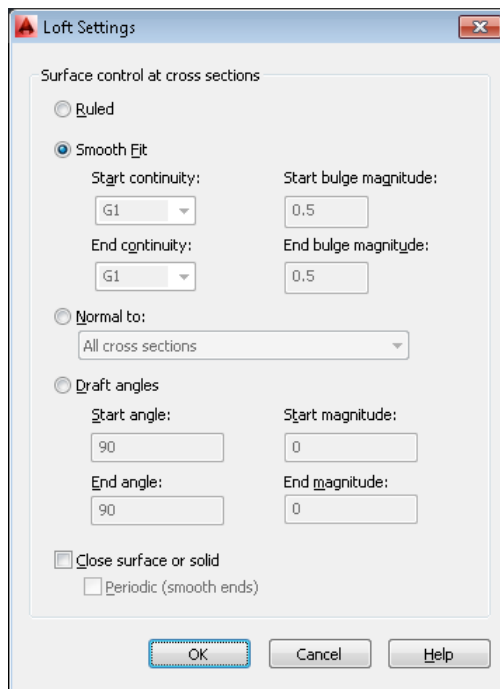
Construction of Volute



Sample-view after data import

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



Settings for lofted surface

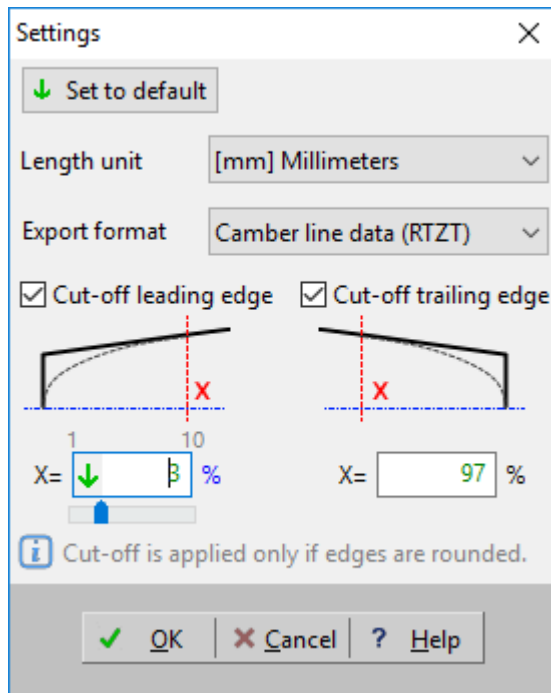
4. Repeat steps 1 to 4 for remaining parts of the volute

5.2.2.1.4.4 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 export parameters:

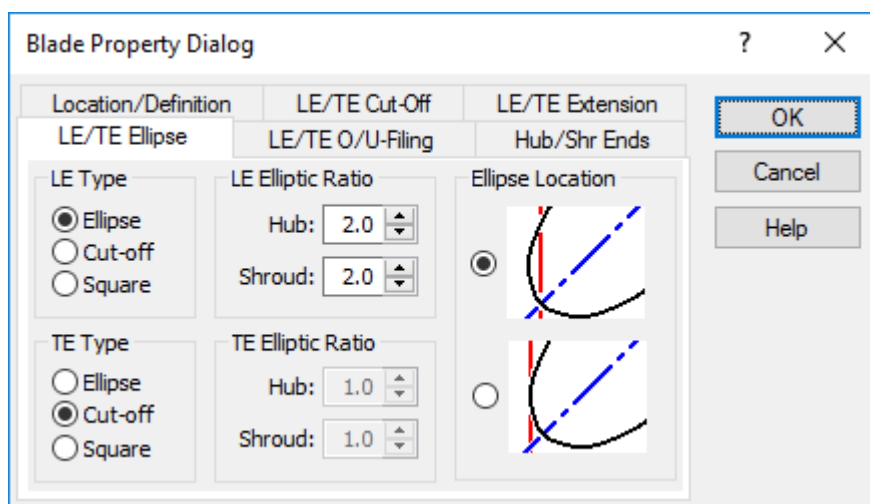


If the exported RTZT-File is expected to be imported in BladeGen, enabling cut-off of rounded edges is recommended, since blade edges shape must be specified using Blade Property Dialog within BladeGen. Therefore, the shape of the blade edges designed in CFturbo and the ones created in BladeGen could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

BladeGen import

In BladeGen, by choosing Ellipse type and setting a ratio value for leading or trailing edges, 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

Problem	Possible solution
Blades with asymmetric thickness distribution not supported.	
Blades with asymmetric thickness distribution will be imported in BladeGen in such a way that the thickness distribution is symmetric with respect to the camberline.	-
Edge shape should be specified in BladeGen.	
BladeGen settings will override any blade edges thickness distribution imported from CFturbo (more information above).	-

5.2.2.1.4.5 CATIA (Dassault Systèmes)

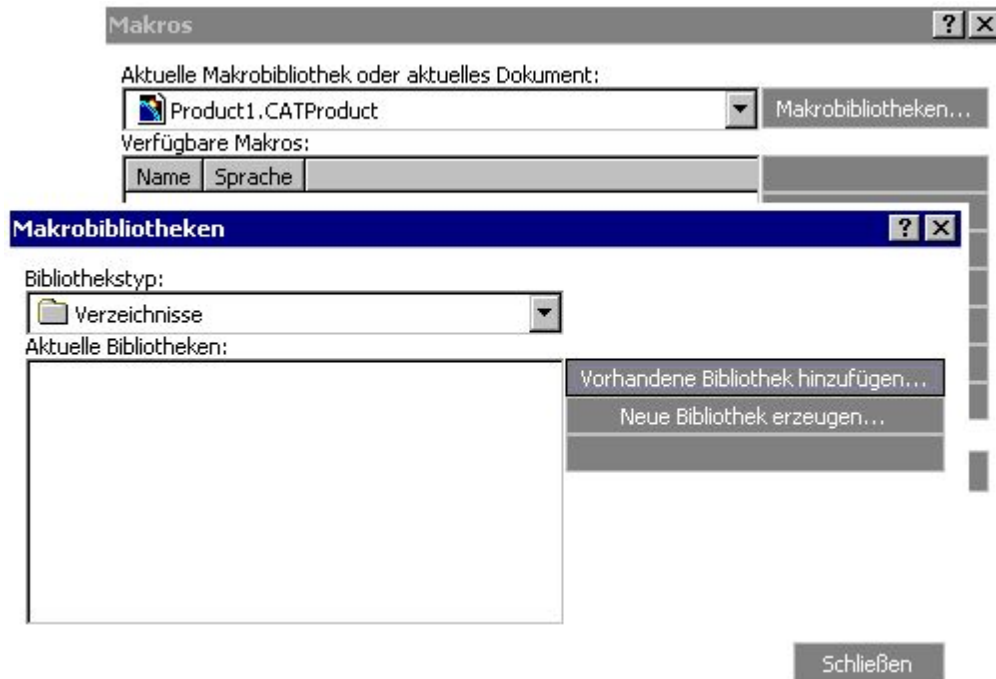
The data-import is realized by a macro that is created for each geometry individually by CFturbo. The macro is loaded and executed in Inventor.

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...>)



- Select macro library and execute macro

5.2.2.1.4.6 Creo Parametric (PTC)

The following files are exported by CFturbo for impellers:

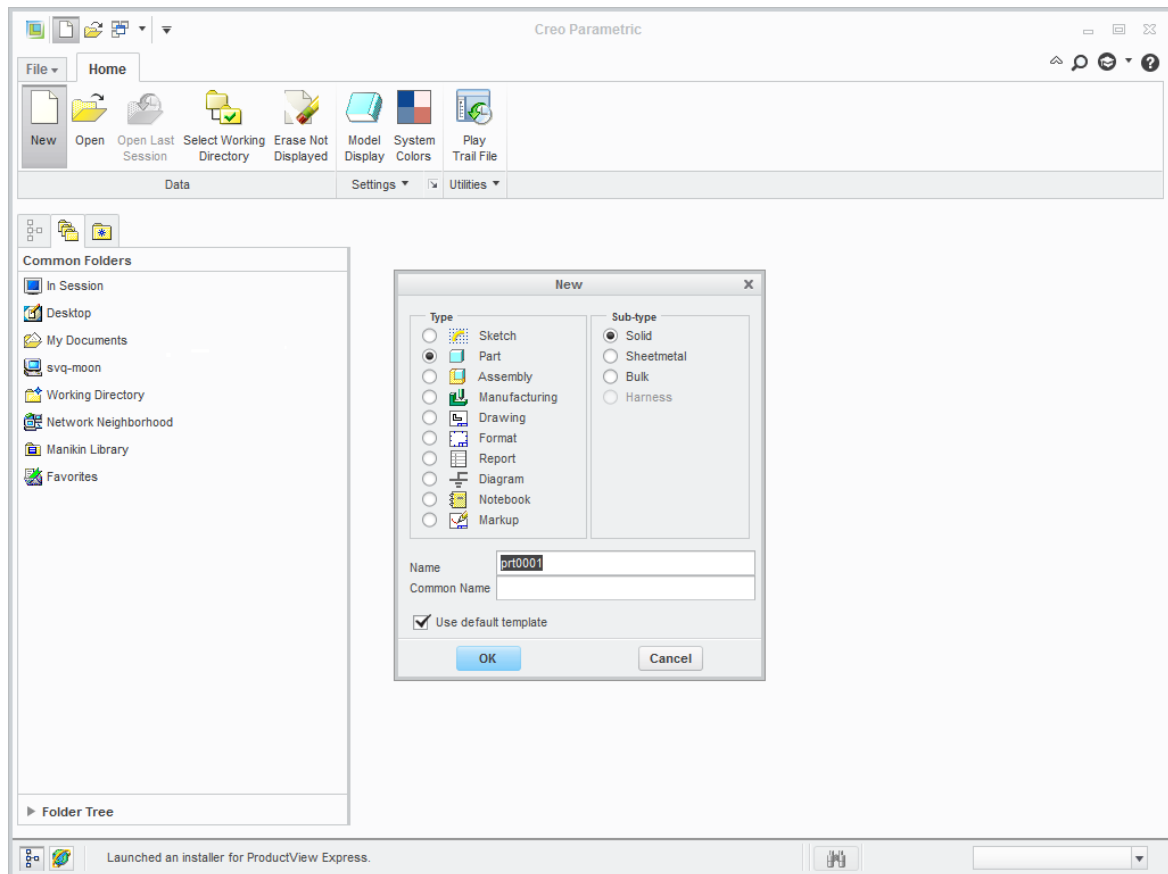
- *-hub.ibl, *-shroud.ibl: points of hub and shroud
- *-profile.ibl: points for blade profiles
- *.ibl: all points for hub, shroud and blades

The following files are exported by CFturbo for volutes:

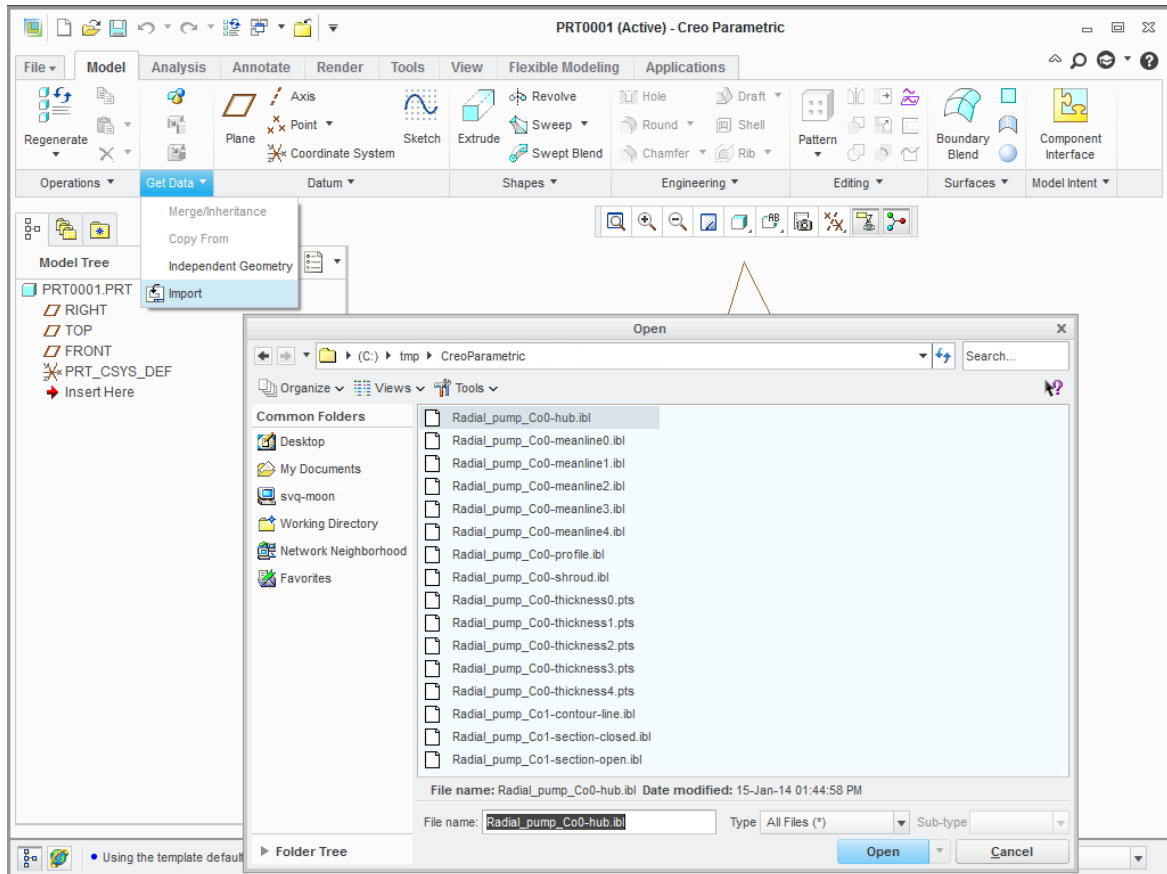
- *-contour-line.ibl: spiral contour points
- *-section-closed.ibl: points for all spiral, cut-water and closed diffuser sections
- *-section-open.ibl: points for all open diffuser sections

Import of curves

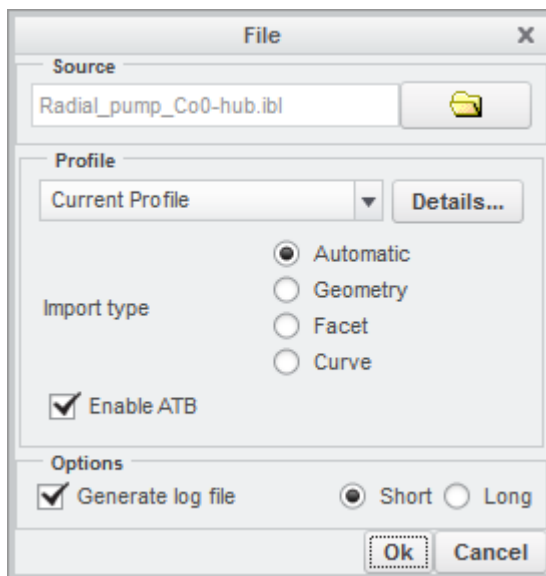
1. Home | New | Part



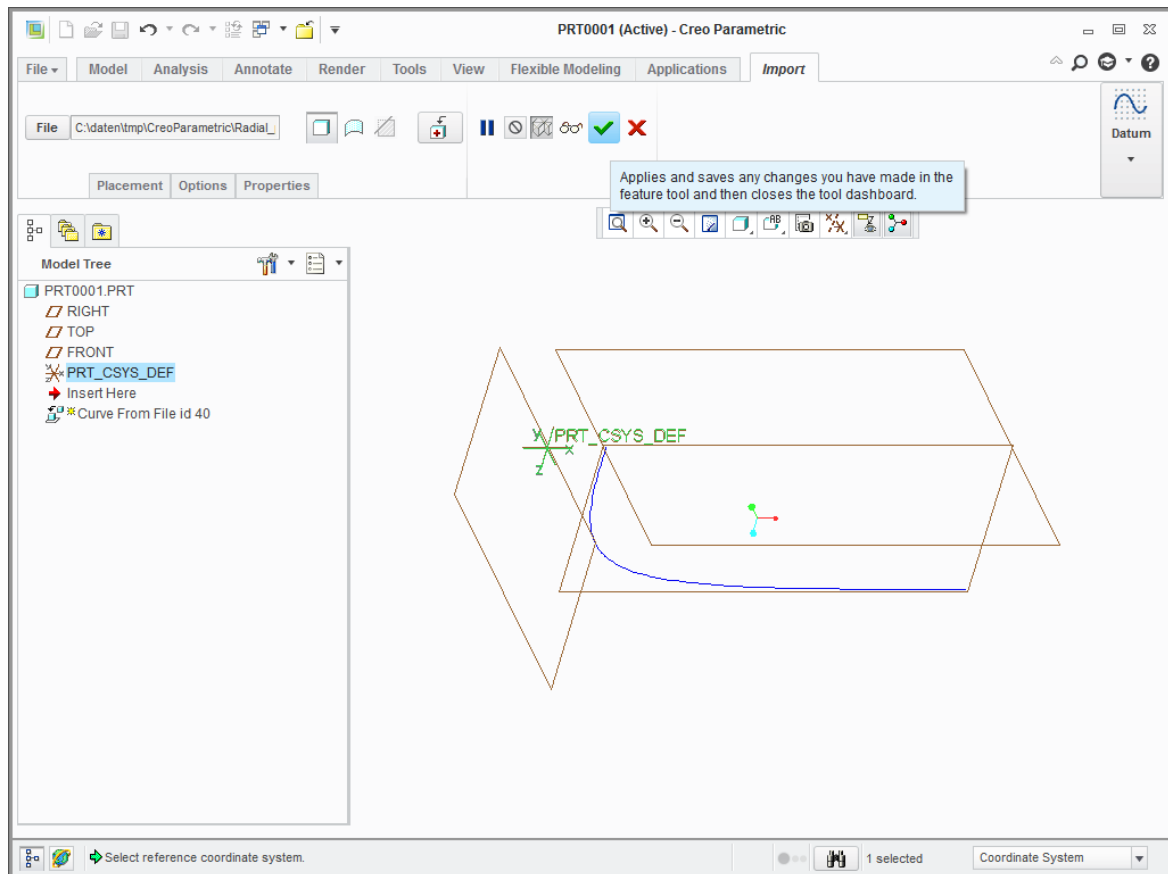
2. Model | Get Data | Import. Select *.pts or *.ibl file



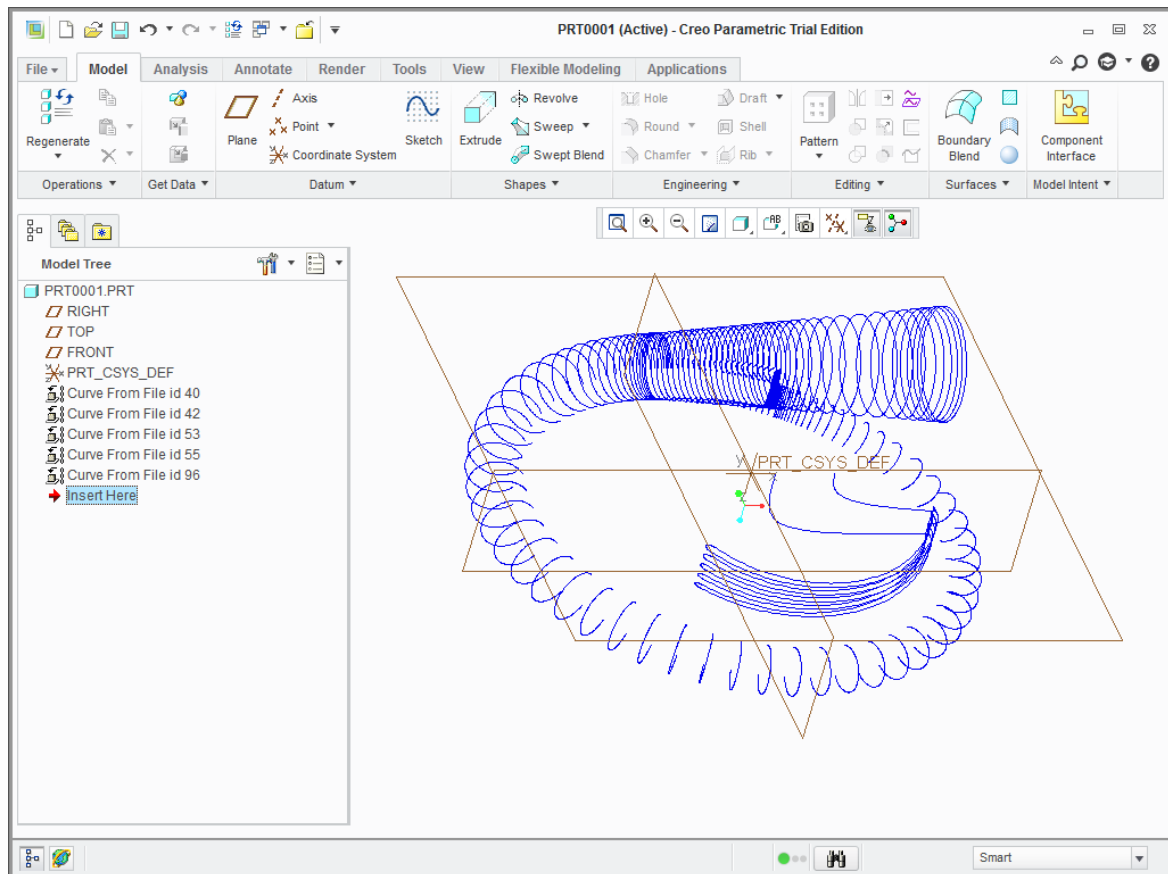
3. In "File" dialog, select desired import options



4. Confirm to finish import process

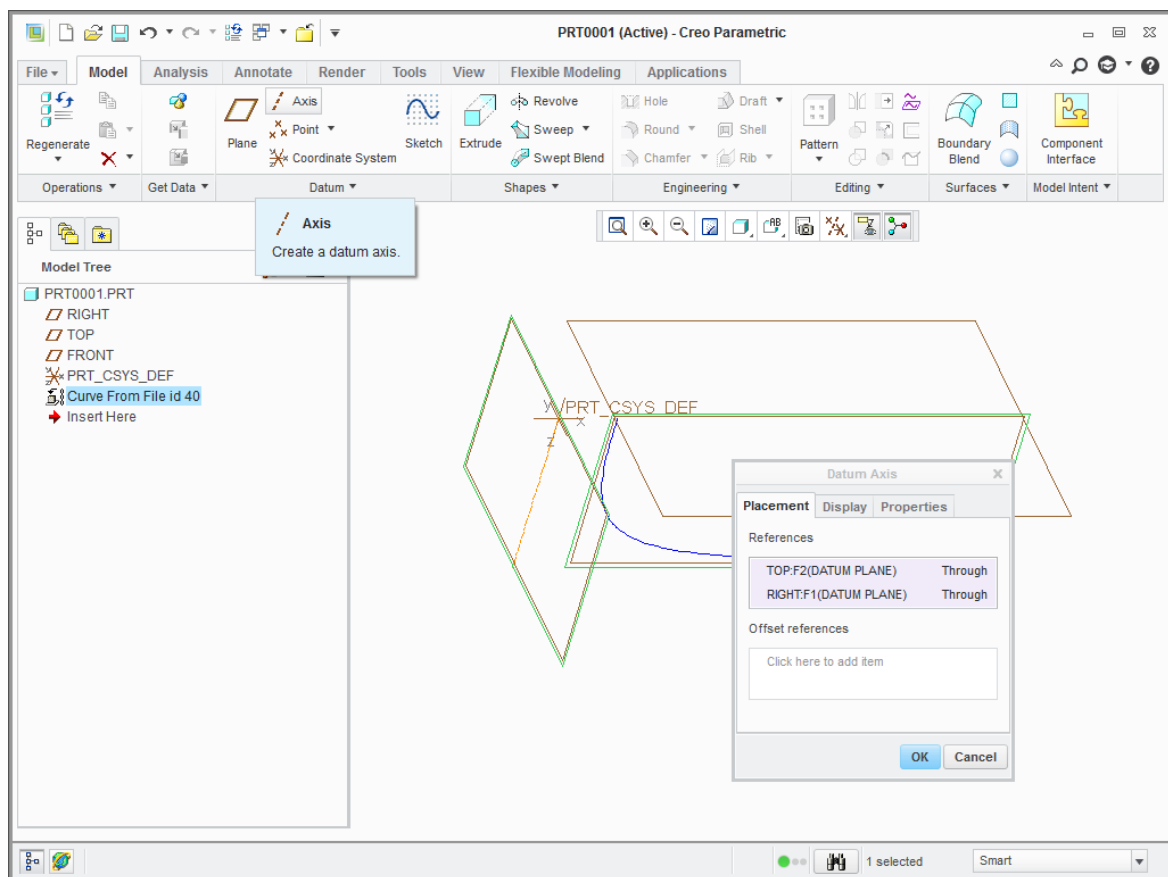


All curves can be imported in this way



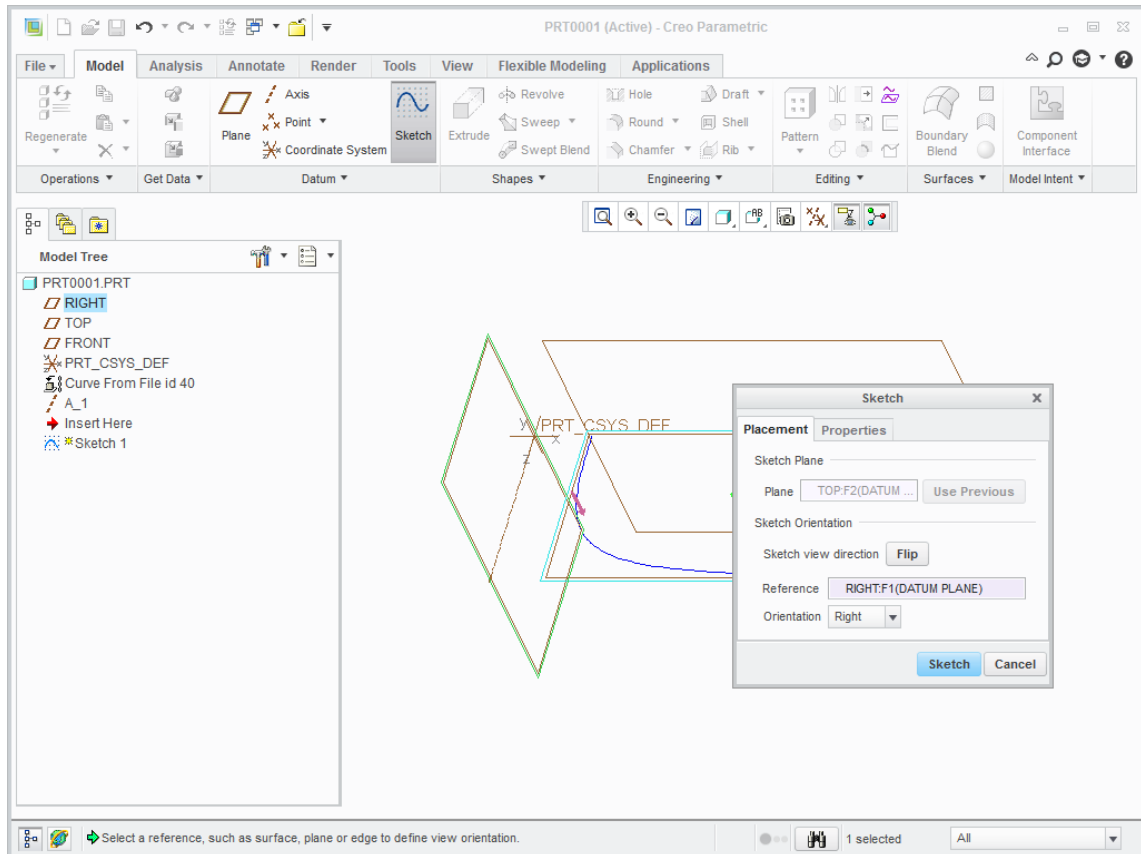
Creating revolution surfaces

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

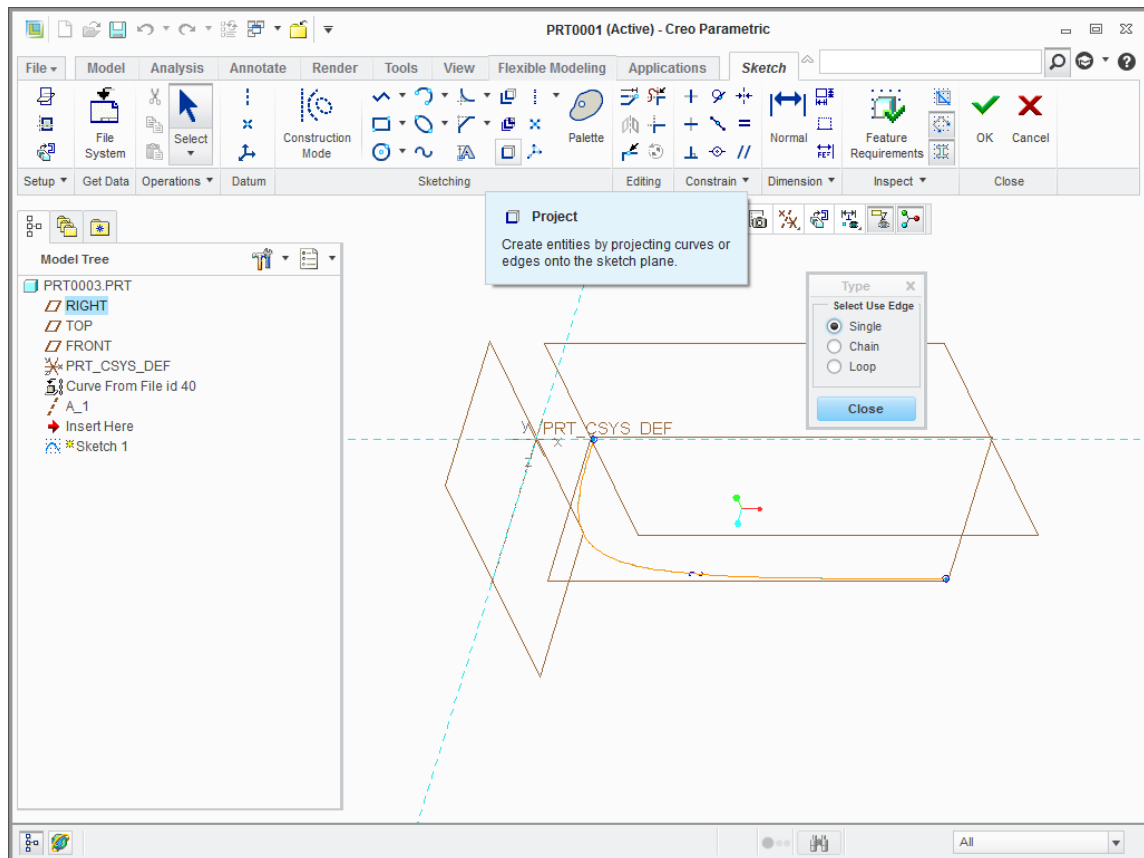


2. Model | Datum | Sketch: create a new sketch

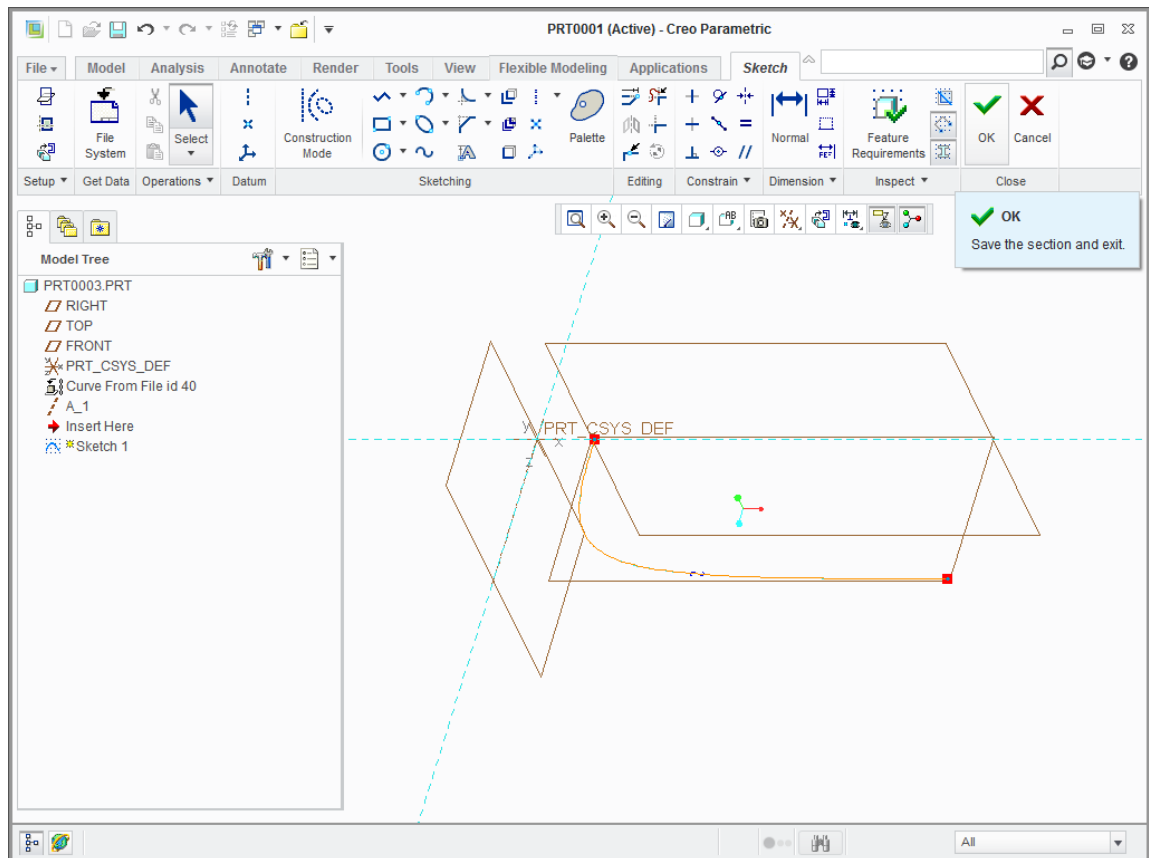
- Select the plane containing the curve to be revolved. Reference and orientation items are set automatically after selection.



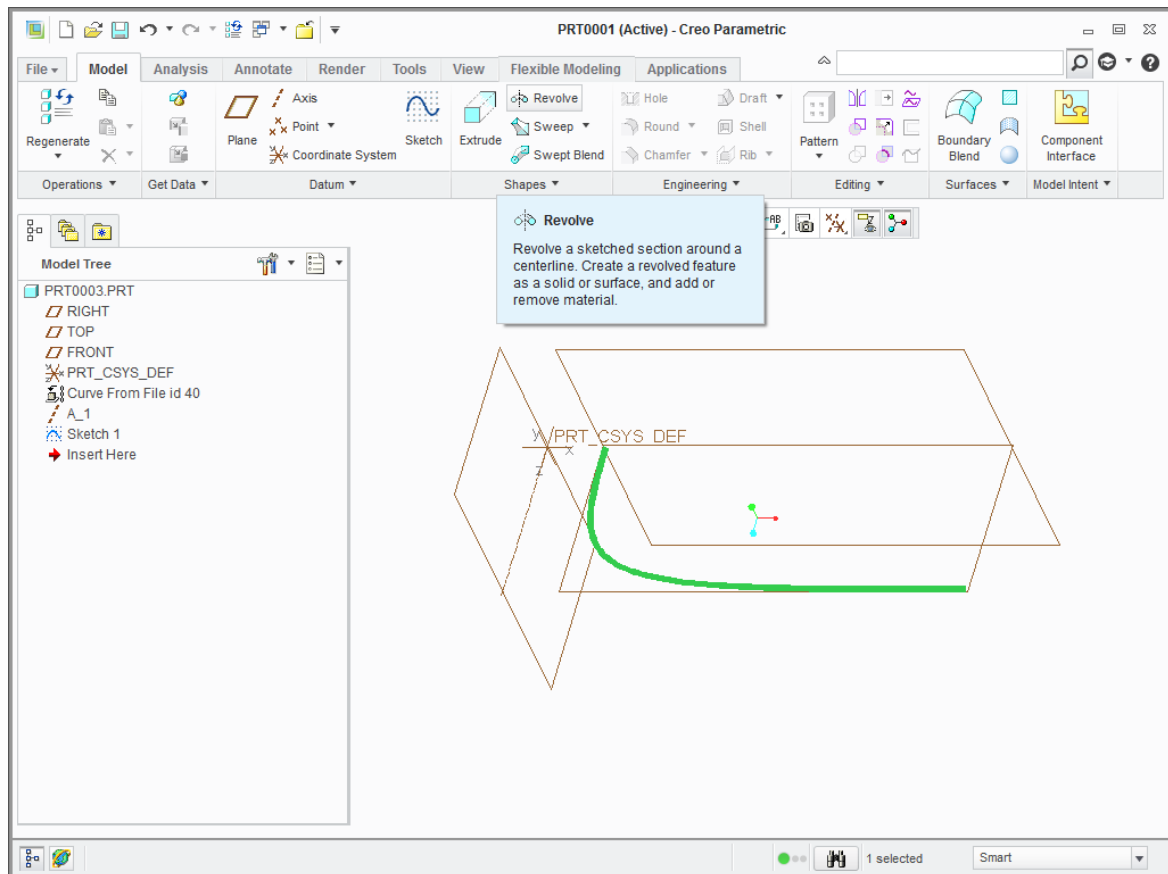
- Sketch | Sketching | Project: do a projection of the curve selecting the curve. Select option "Single" and click on "Close"



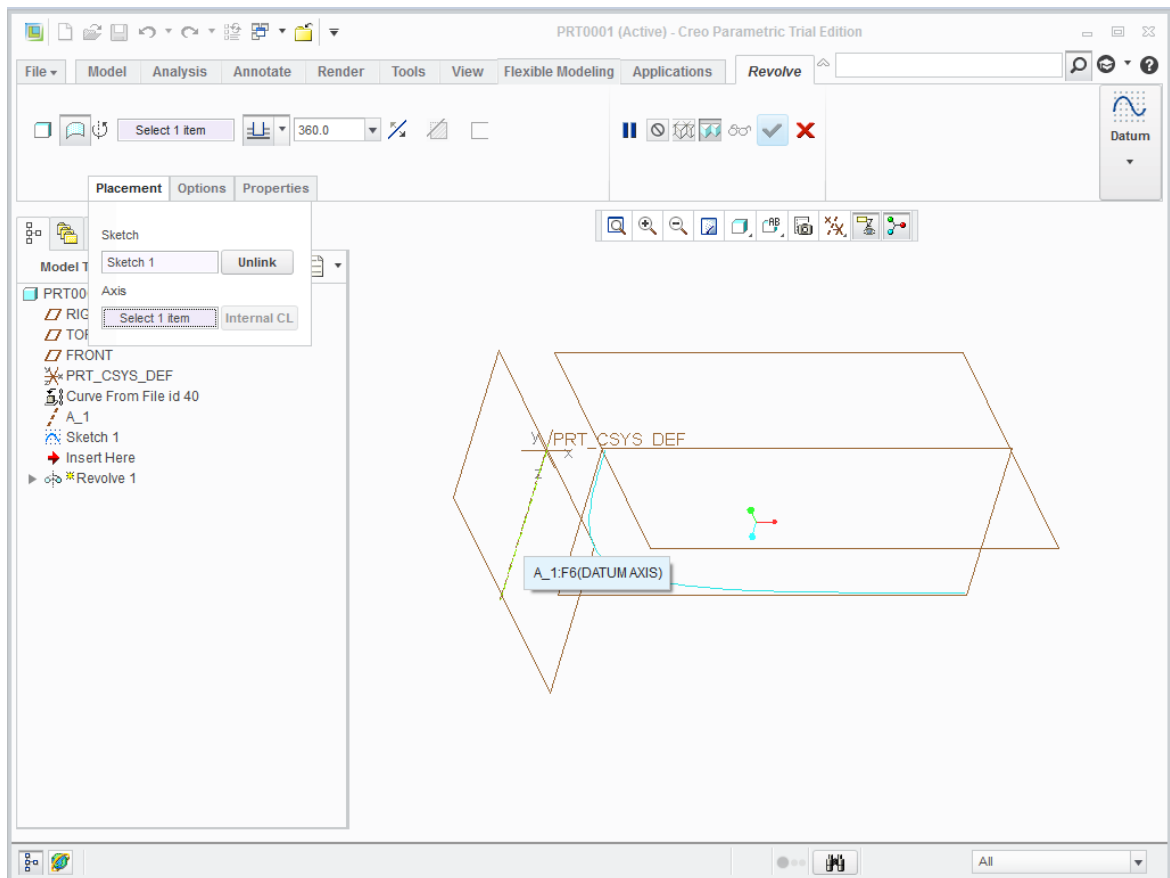
- Finalize sketching task by clicking on "OK"

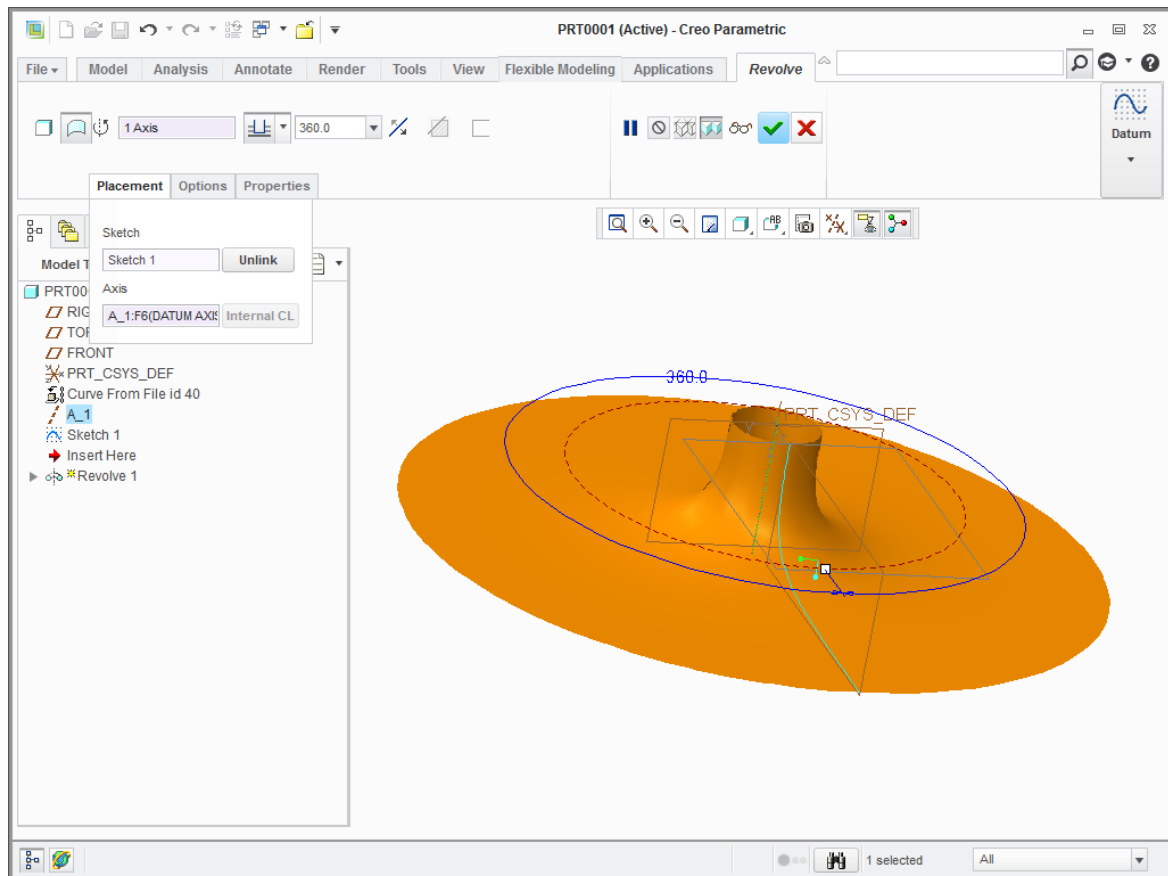


3. Select the curve and click on Model | Shapes | Revolve

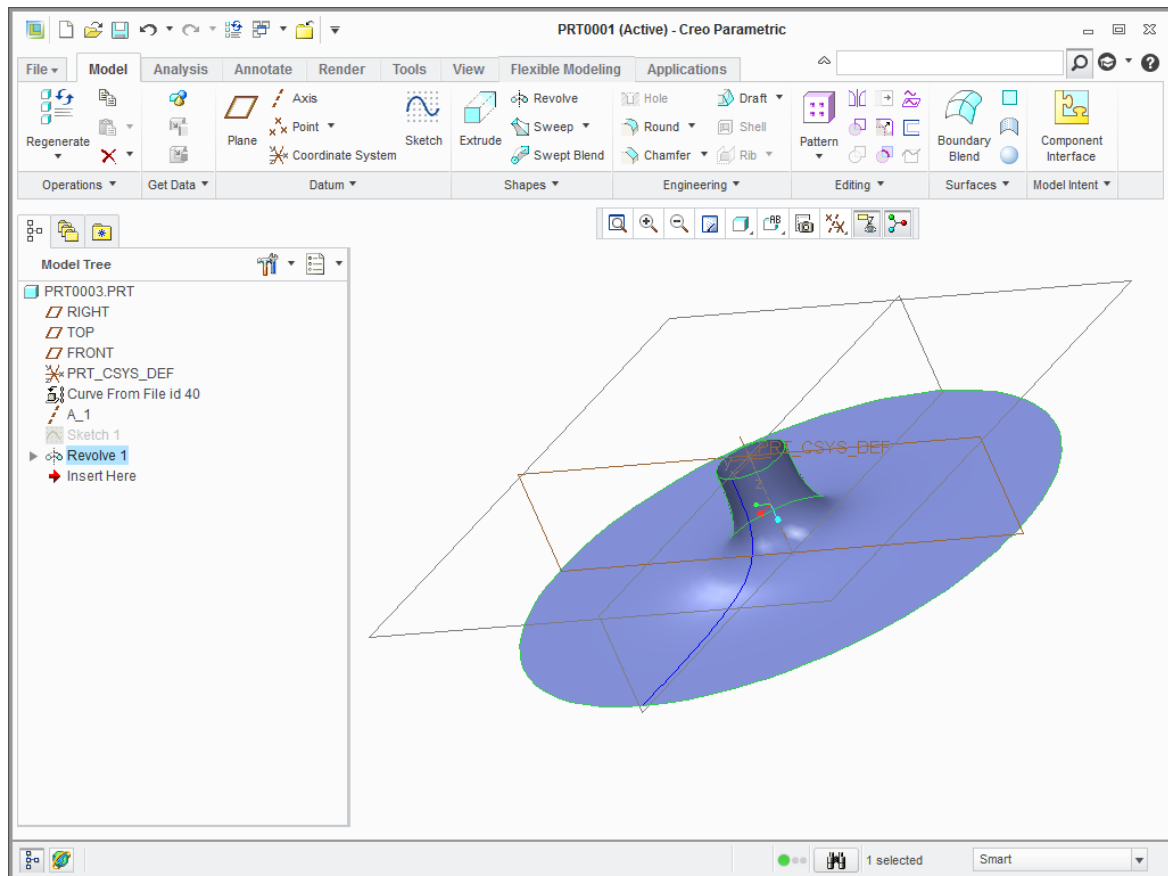


4. Click on field "Axis" under tab "Placements" and select the revolution axis. Surface of revolution will be generated





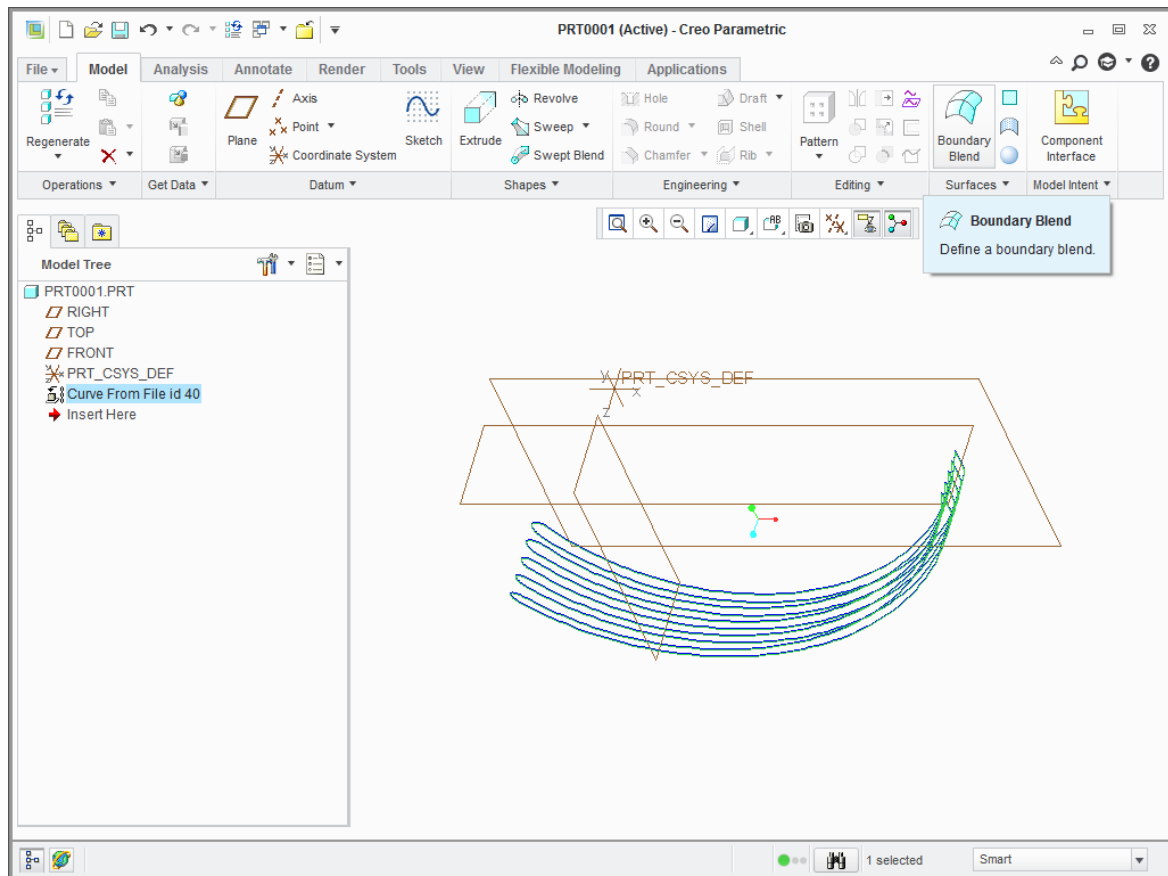
5. Finalize revolve task by clicking on "OK"



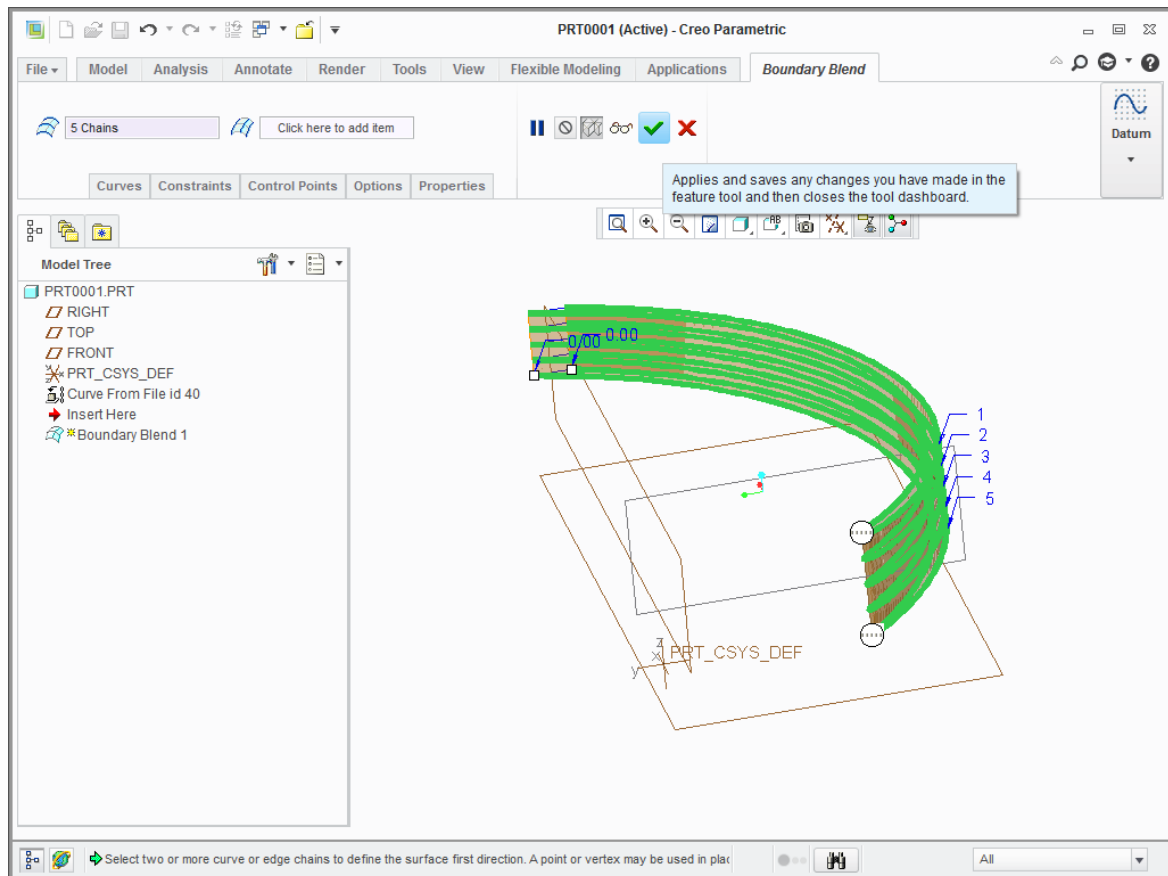
Creating lofted surfaces

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

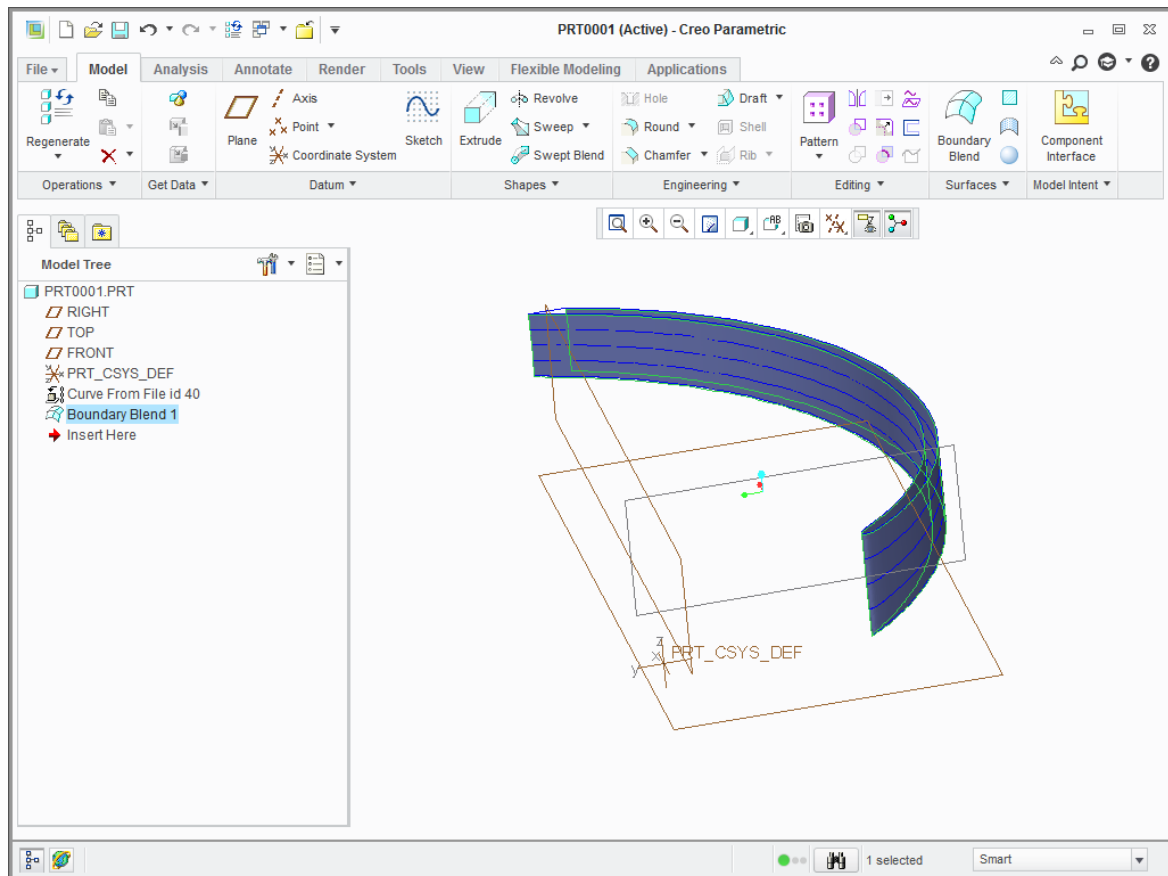
1. Model | Surface | Boundary Blend

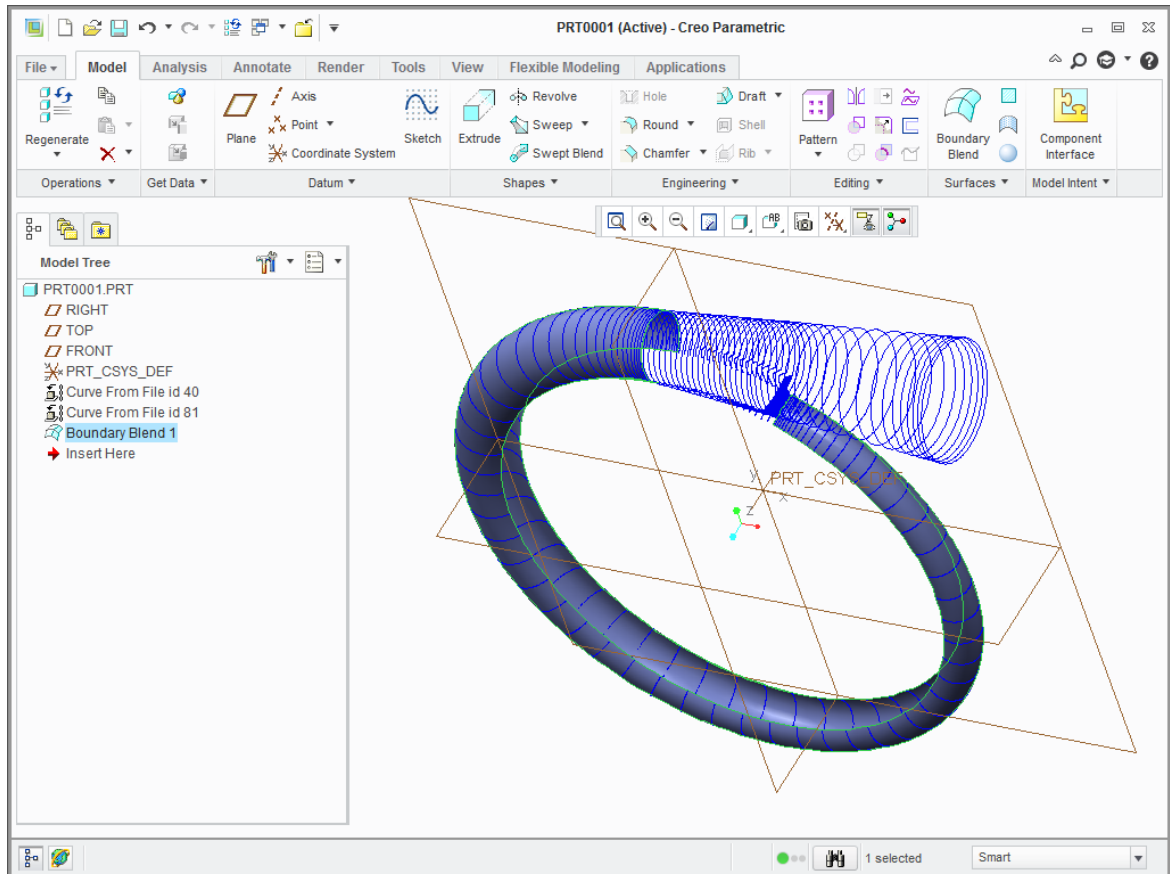


2. Select desired curves (use Ctrl for multi-selection)



3. Finalize Boundary Blend task by clicking on "OK"



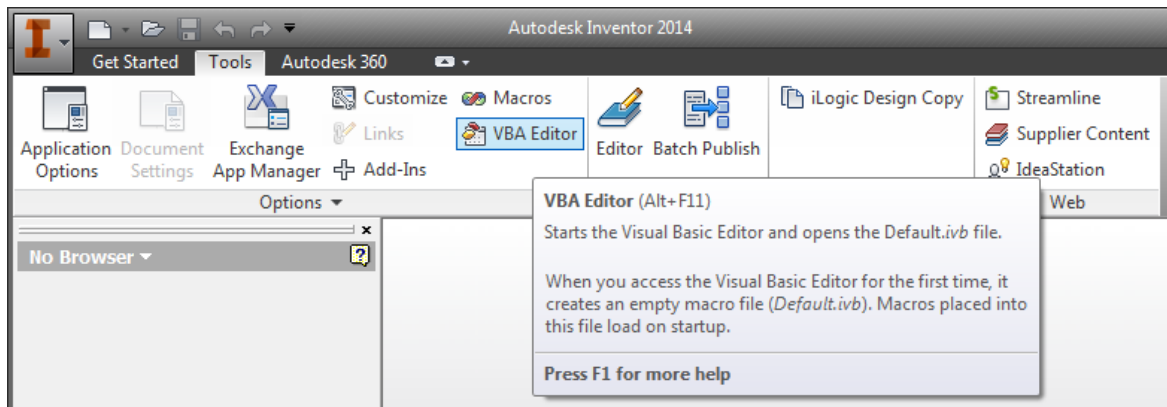


5.2.2.1.4.7 Inventor (Autodesk)

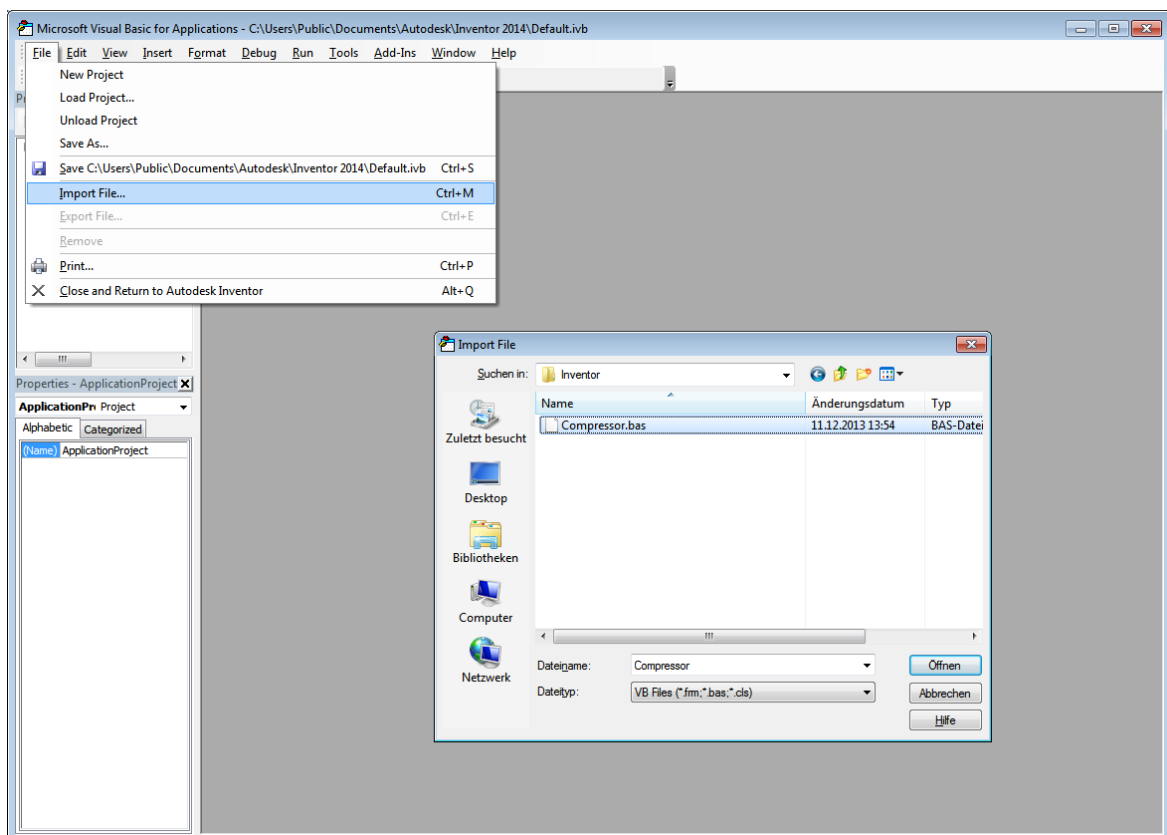
The data-import is realized by a macro that is created for each geometry individually by CFturbo. The macro is loaded and executed in Inventor.

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

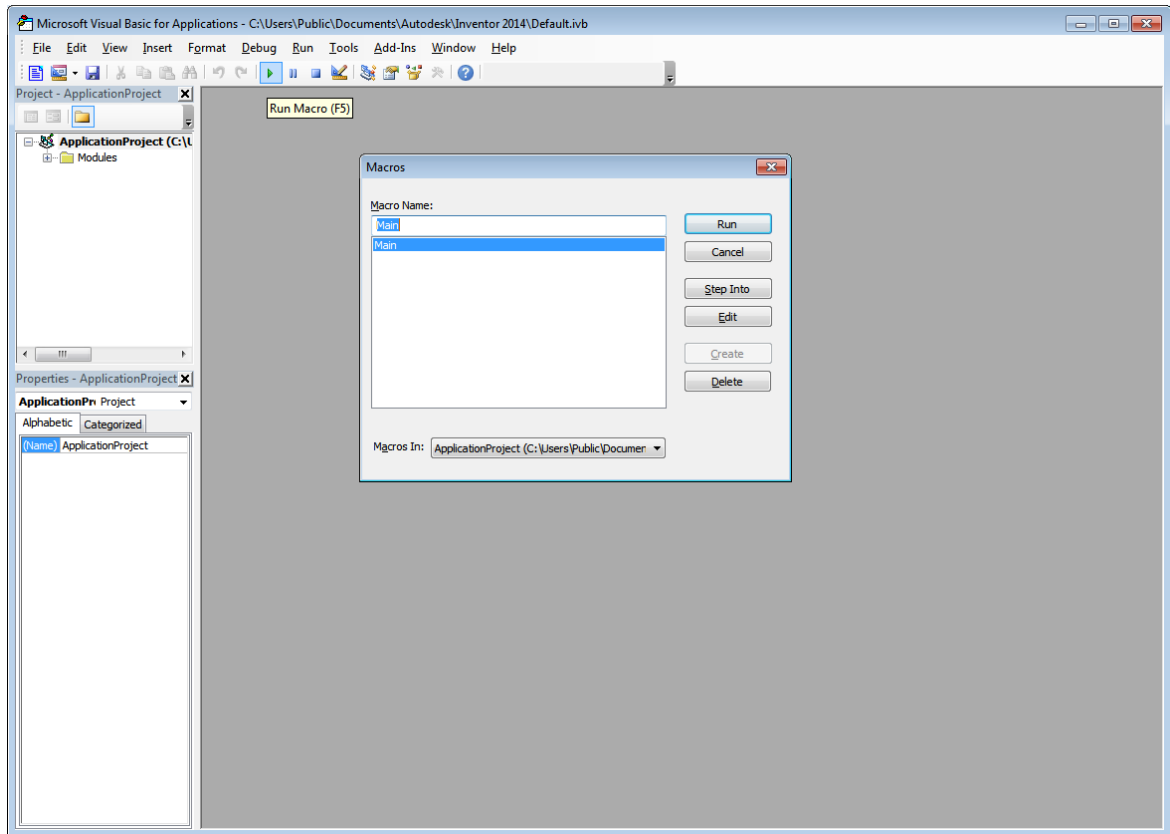
- Tools | VBA Editor



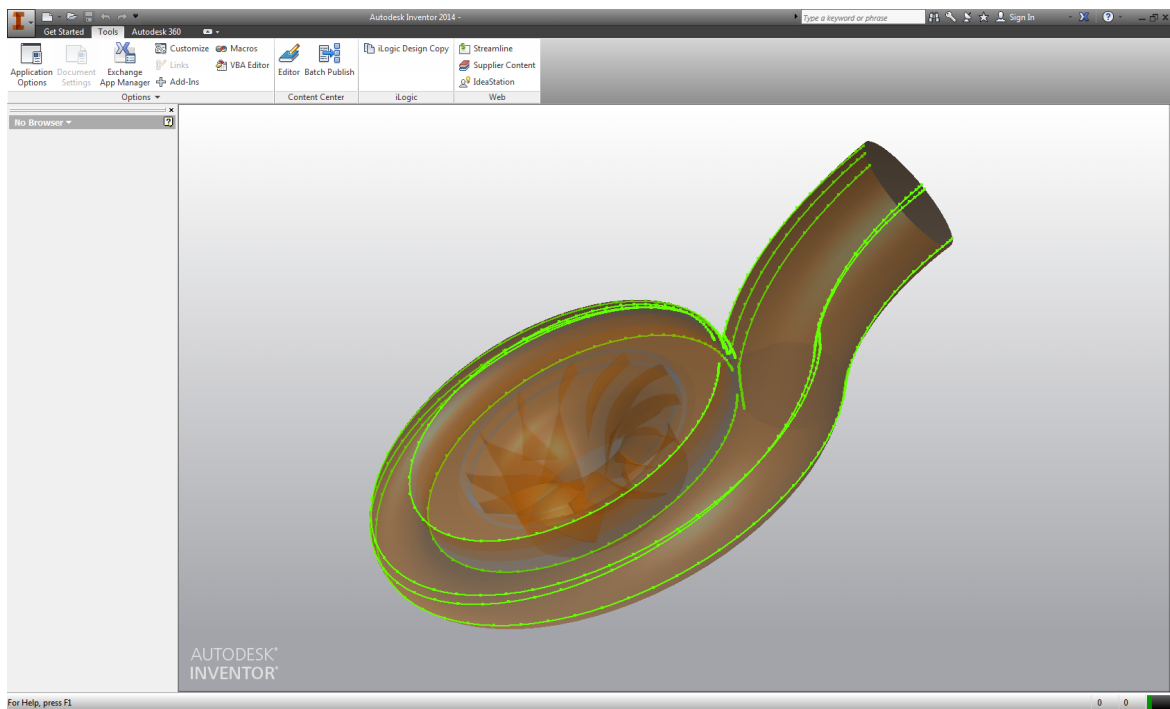
- Open file-open-dialog by *File | Import File...* and select *.bas macro-file, possibly a new project has to be created *File | New Project*



- 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](#)⁴⁸⁶ 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.

5.2.2.1.4.8 SpaceClaim (ANSYS)

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

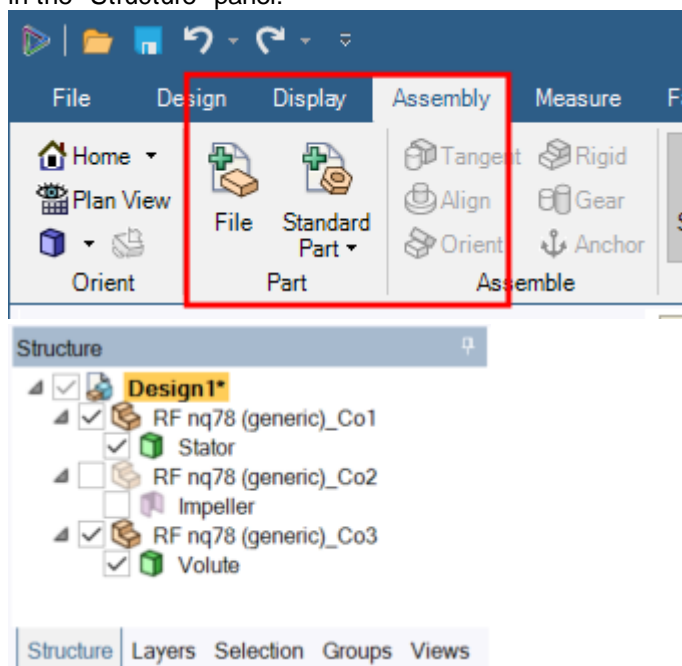
1. Using the "CAD, CAM/ SpaceClaim" export interface

When using the 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 SpaceClaim project by "Assembly/ File" and will be displayed in the "Structure" panel:

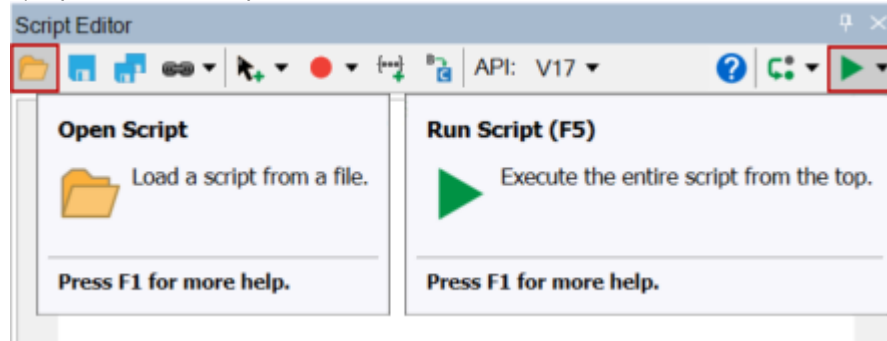


- One **Python** script *.py

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 SpaceClaim.

To run the script:

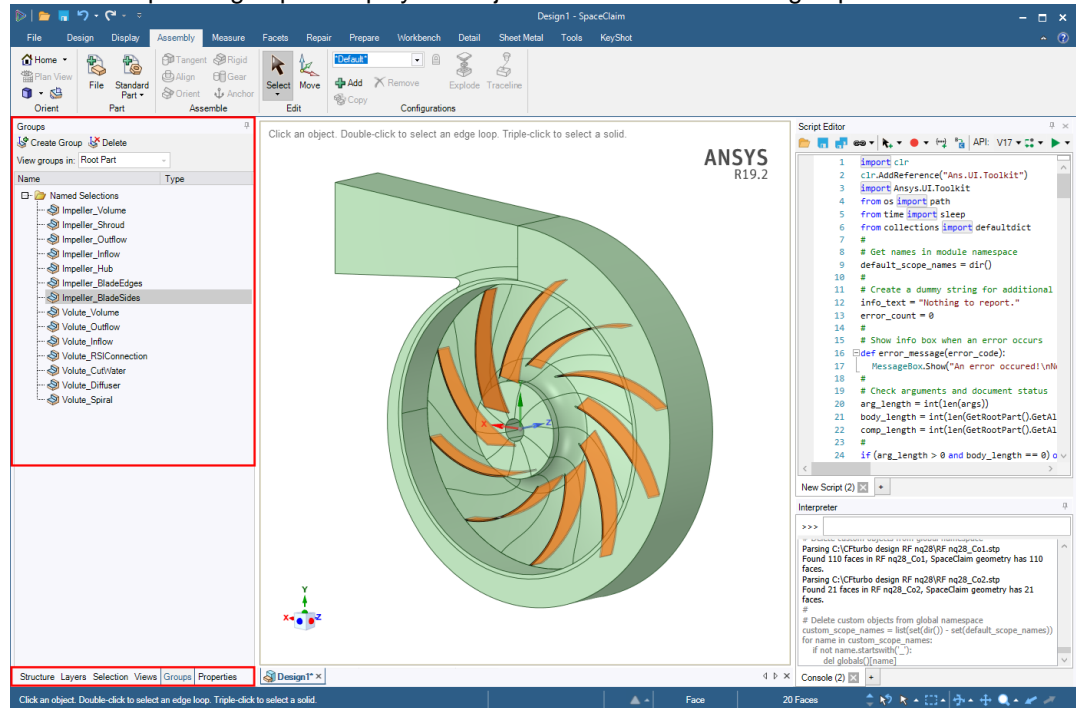
- 1) Open the "Script Editor" by "File/ New/ Script"
- 2) Open and run script:



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

The detected names are visible in SpaceClaim on the panel "Groups". 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.

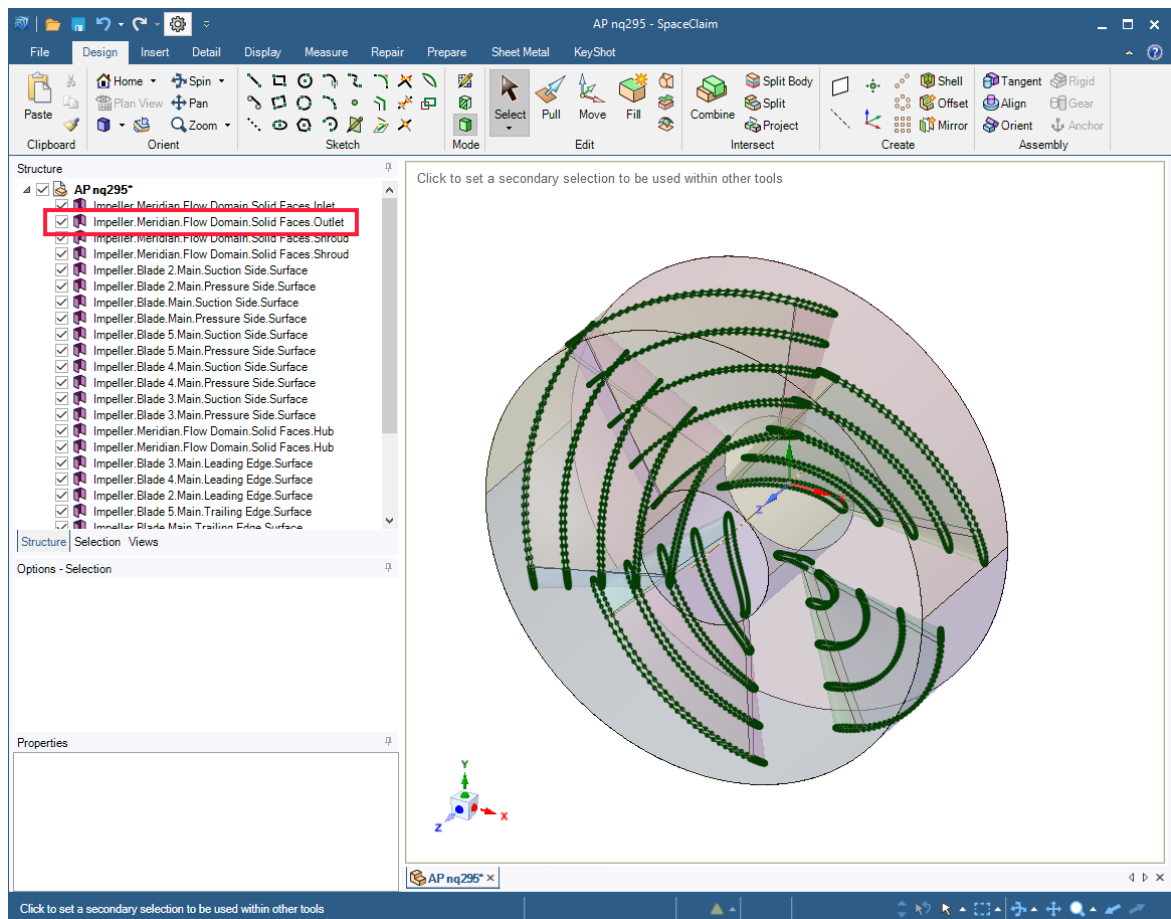


The described procedure is carried out automatically when the option "Open exported files" is selected in the CFturbo export dialog.

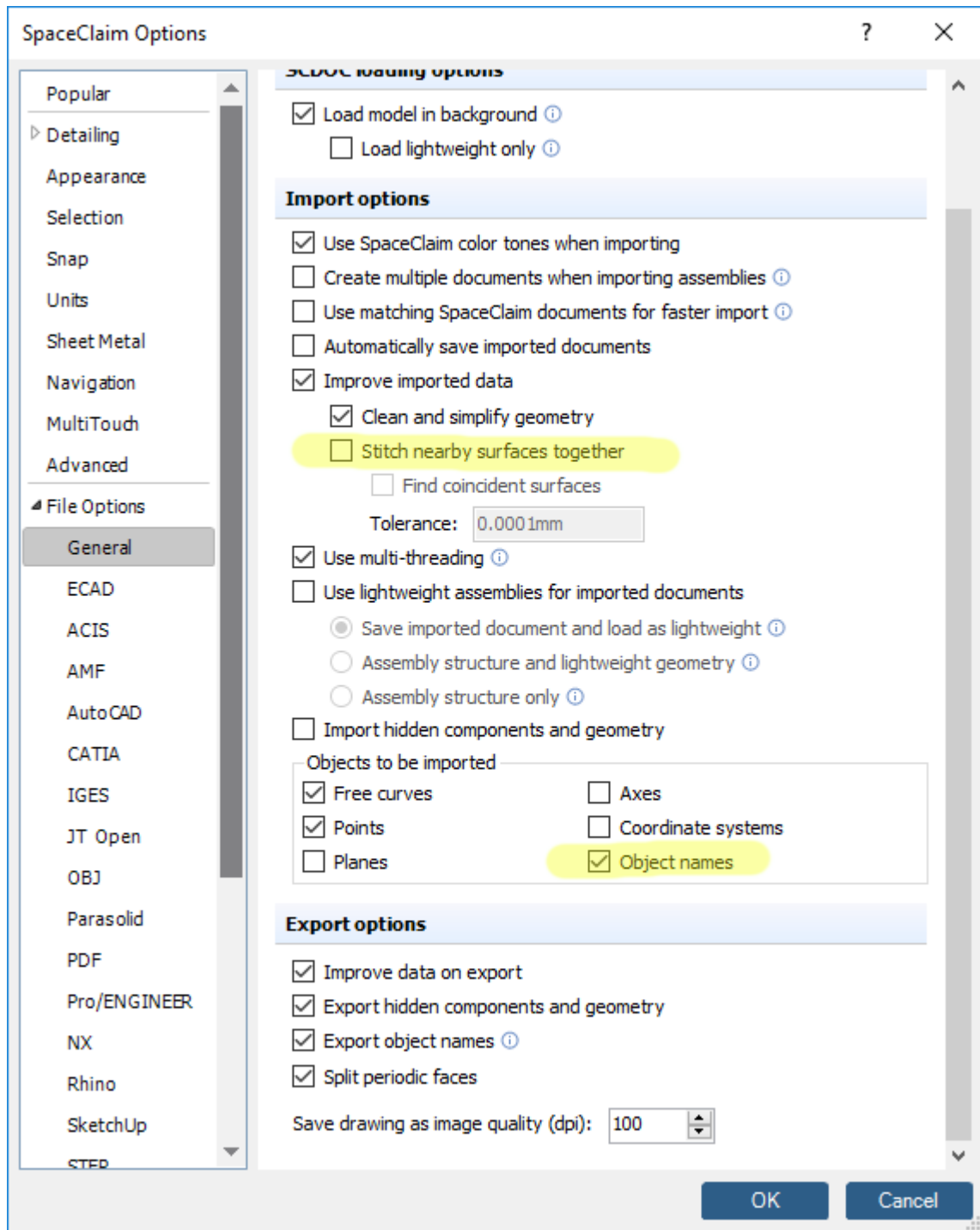
2. Using the "Basic/STEP" export interface

When using the generic STEP export interface all parts currently displayed in the CFturbo [3D view](#)²²⁵ are exported.

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

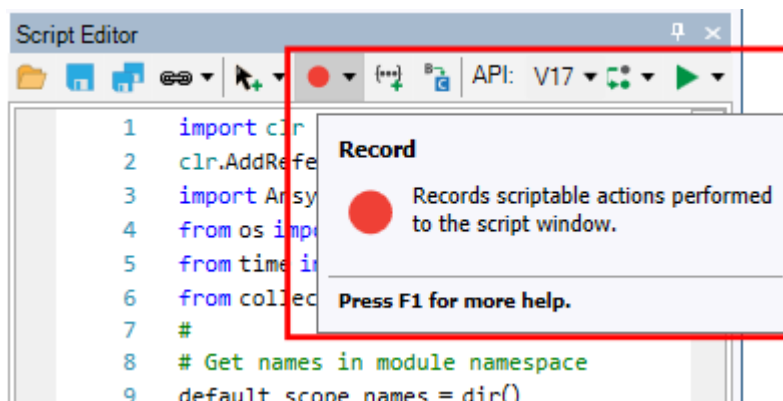


Following settings should be checked under SpaceClaim Options in order to allow the recognition of names for each imported surface:

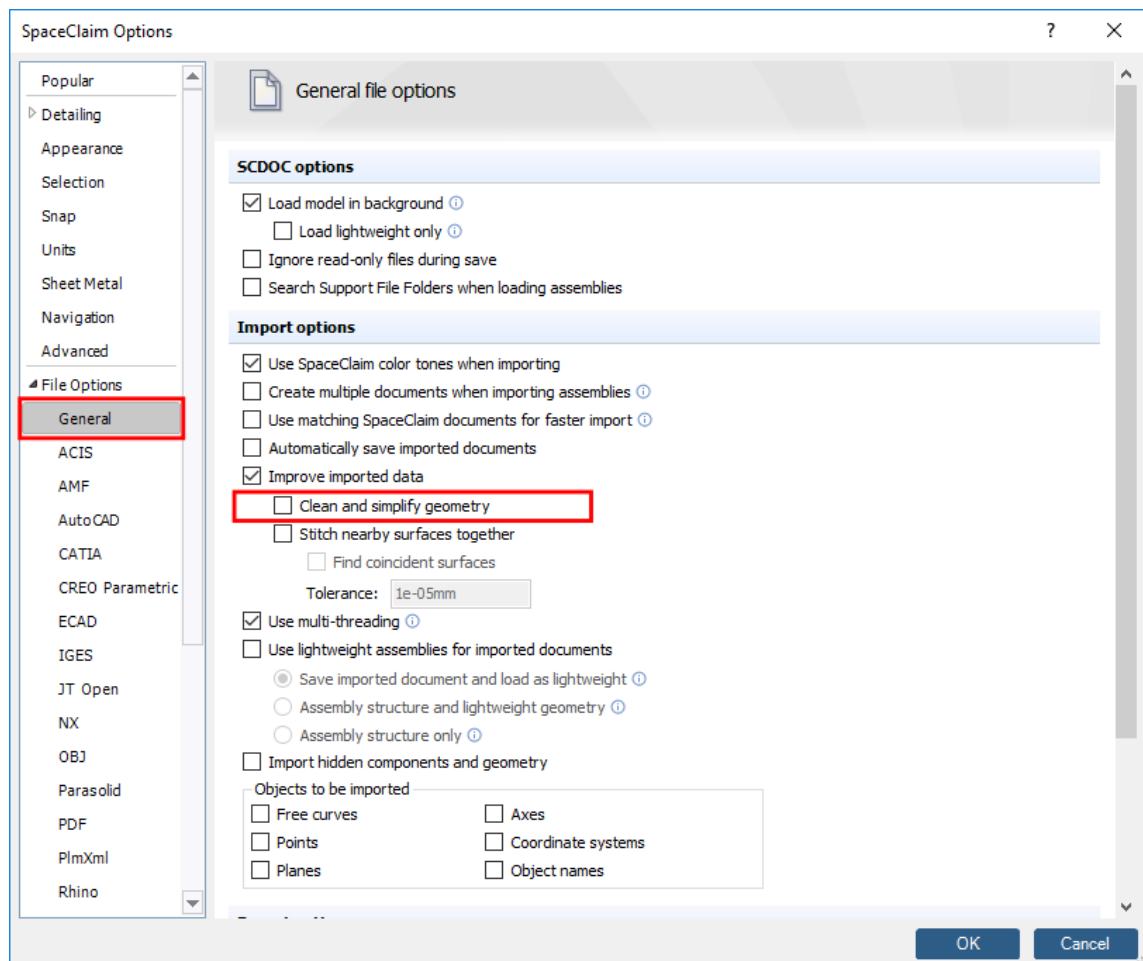


- When importing the geometry files and the Python script together, there is a chance that SpaceClaim will automatically activate the “record” function in the script editor. It is strongly advised to check and deactivate this feature, to ensure the proper function of the pre-built Python

script.



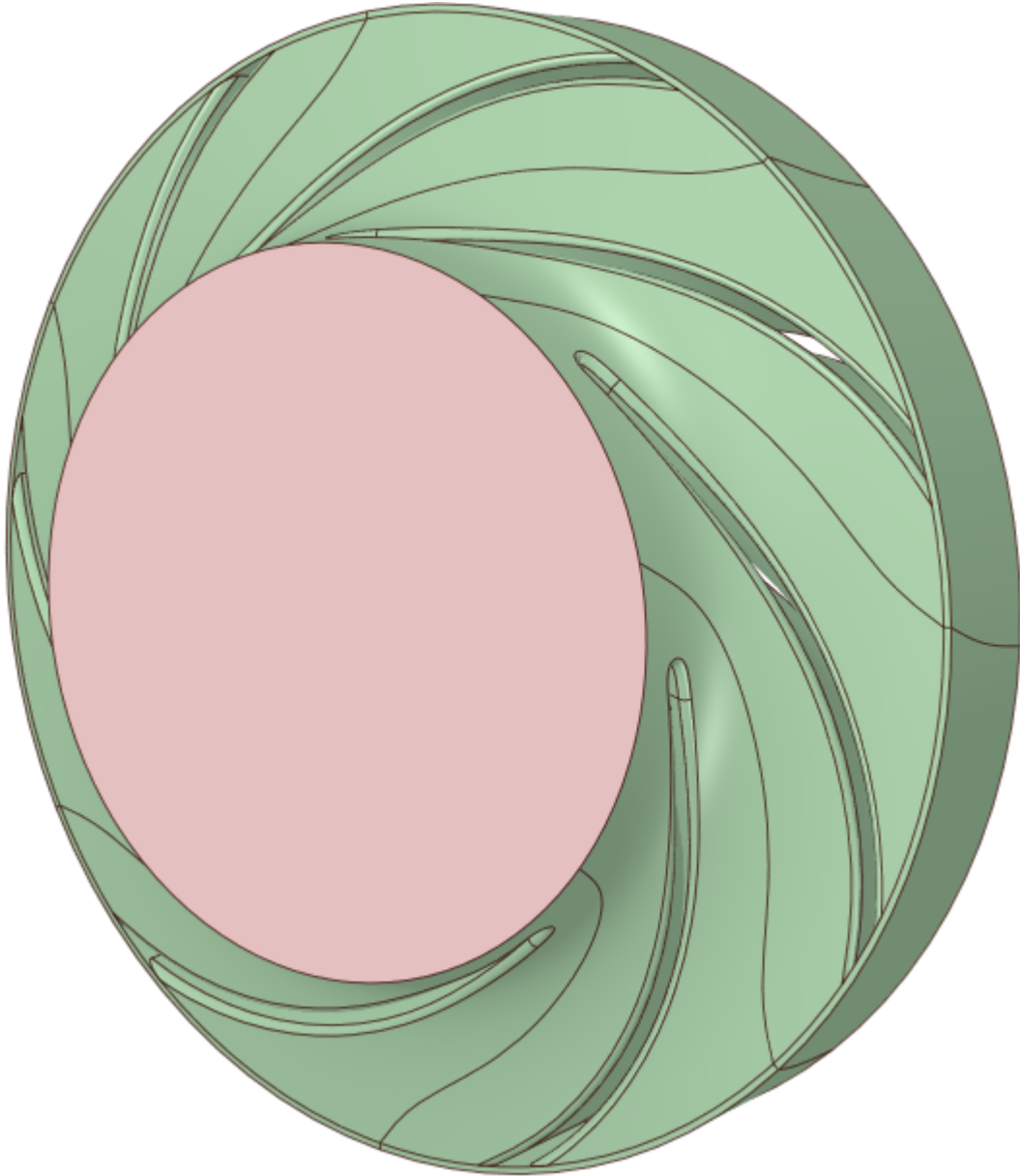
- Depending on the complexity of the geometry, SpaceClaim may try to simplify the geometry imports. Any simplification will prevent the Python script from functioning properly. It is therefore advised to deactivate the “Clean and simplify geometry” option in the SpaceClaim Options under the “General” 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.



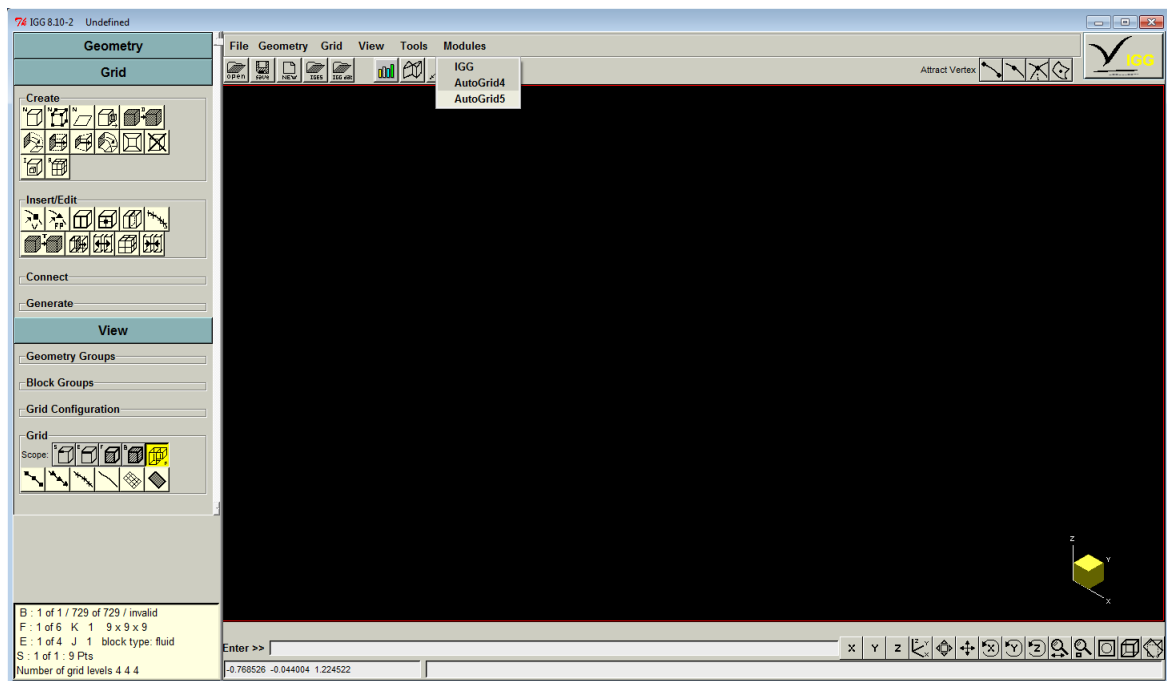
- Under certain circumstances some parts of the geometry are missing when forwarding the geometry from SpaceClaim to ANSYS meshing. The critical area is mainly the splitter edge of double volutes. In SpaceClaim the geometry looks fine. This is a known problem in SpaceClaim since version 19.2, Defect Number: 130684.

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

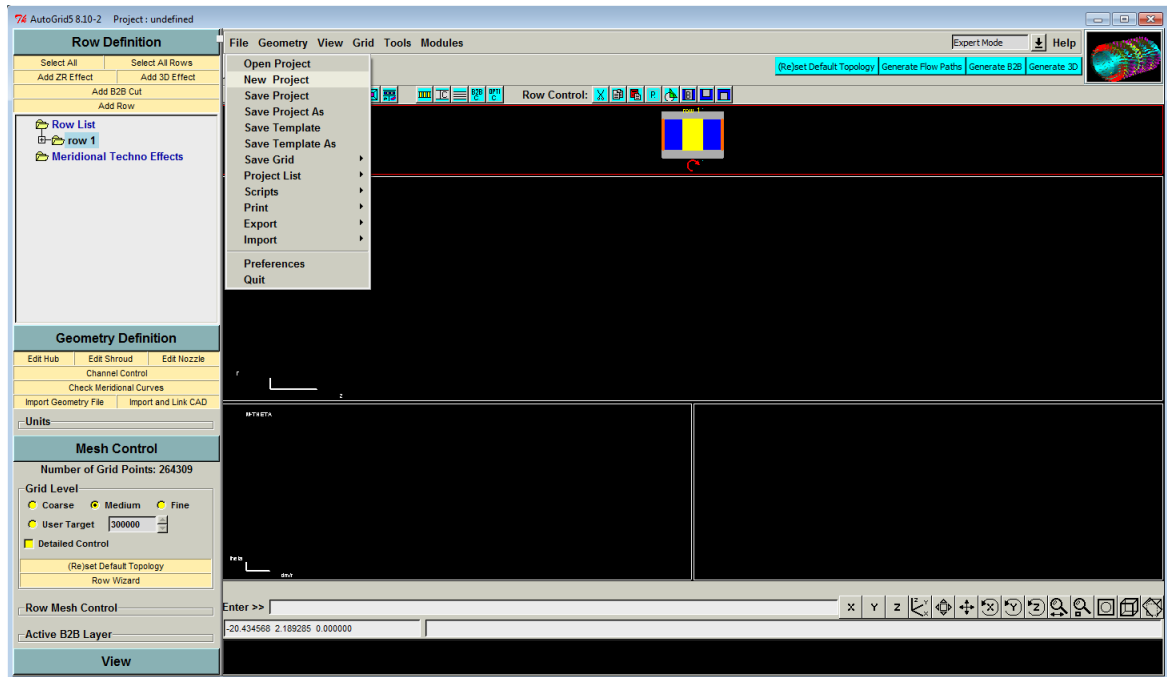
5.2.2.1.4.9 AutoGrid (NUMECA)

The geometry data for impeller is exported by CFturbo to „geomTurbo“-files which can be loaded by AutoGrid5.

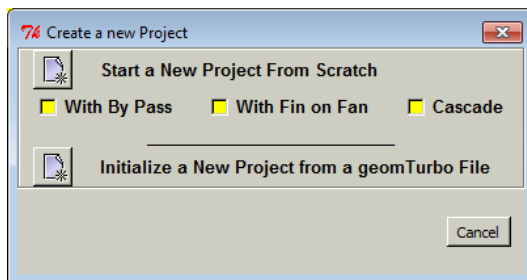
- Start IGG
- Change to AutoGrid5-mode: *Modules | AutoGrid5*



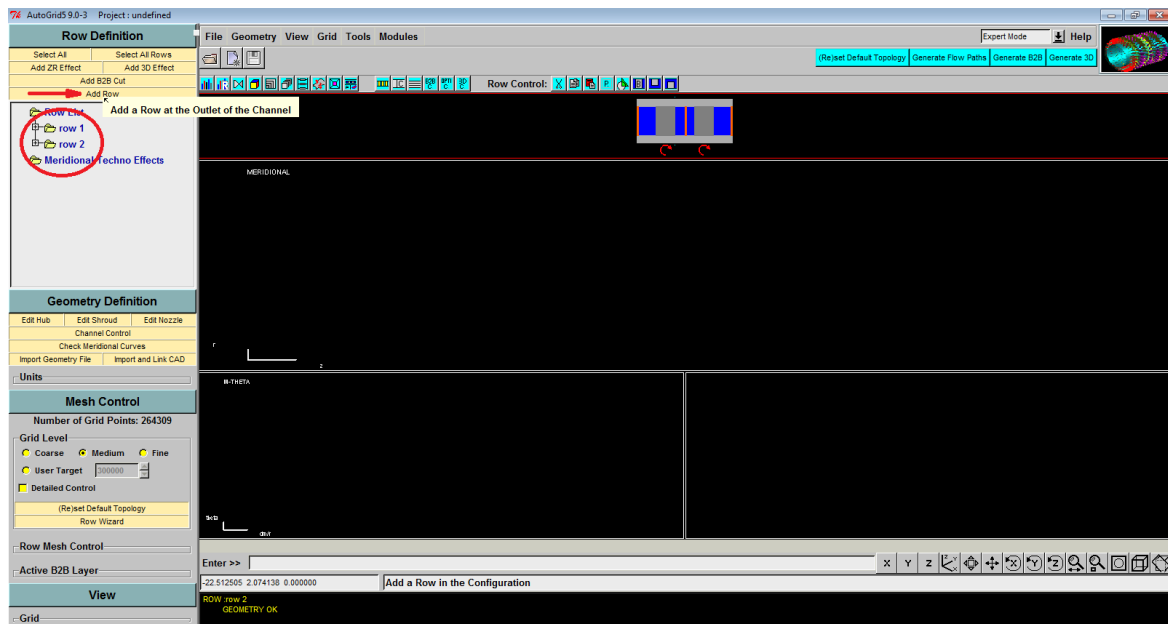
- Open a new project: *File | New Project*



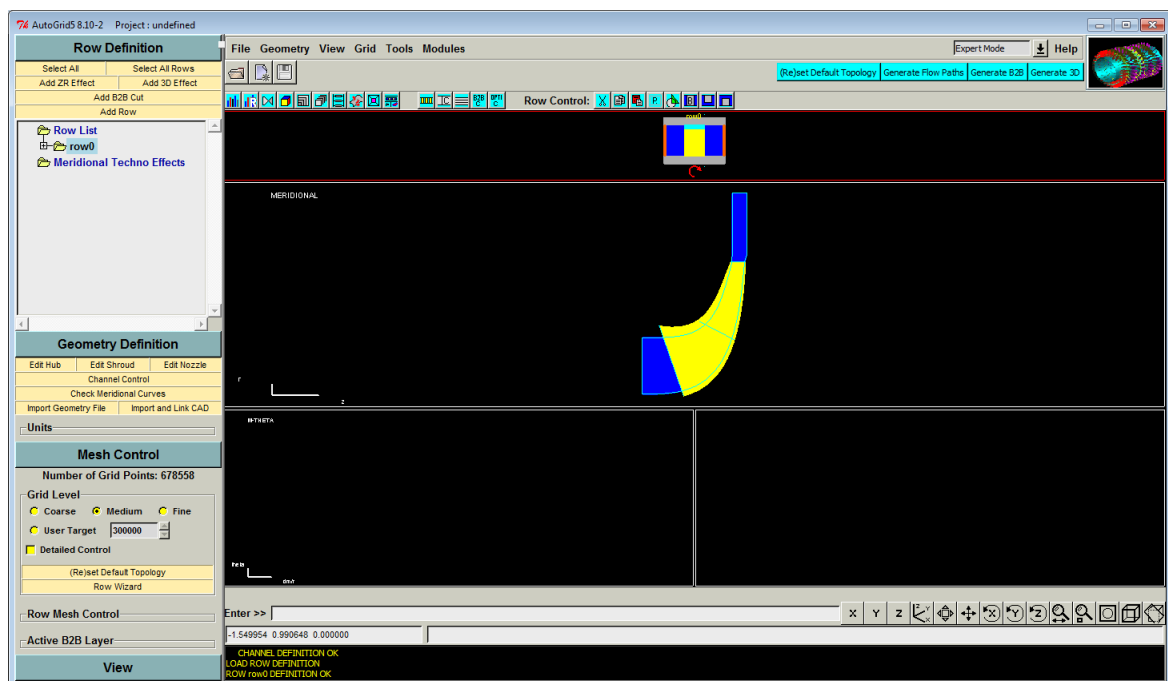
- Close dialog by *Initialize a New Project from a geomTurbo File*



- 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.

The export dialog "Export ICEM-CFD" and the CFturbo2ICEM scripts are only available with the corresponding license.

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. A detailed description of the parameters can be found in the [available documentation](#).

The **Set parameters...** button opens the "Export ICEM-CFD" dialog for defining the meshing parameters. These settings are saved in the *.tinXML file.

ANSYS ICEM CFD

General

Topology
☒ Full (360°)
☐ Single blade passage

Mesh method
☐ Robust (Octree)
☒ Quick (Delaunay)

Miscellaneous
 Component caption: Radial_Impeller
 Triangulation tolerance: 1E-6

Mesh

Tetrahedra
☒ Coarse ☒ Middle ☒ Fine
 Global element scale factor: 1
 Global element seed size: 4
 Min. size limit: 0.5
 Edge criterion: 0.15

Prism layers
☒ Coarse ☒ Middle ☒ Fine
 Number of layers: 7
 Growth law: Linear
 Height ratio: 1.2

Part	Max. size	Max. deviation	Layers	Height	Height ratio	End height	Total height	Height factor
Hub	4	1.1214	<input checked="" type="checkbox"/>	7	0.022321	1.2	0.049107	0.25
Shroud	4	1.1214	<input checked="" type="checkbox"/>	7	0.022321	1.2	0.049107	0.25
Blade	2	1.1214	<input checked="" type="checkbox"/>	7	0.022321	1.2	0.049107	0.25
Blade LE	1	0.74763	<input checked="" type="checkbox"/>	7	0.022321	1.2	0.049107	0.25
Blade TE	1	0.74763	<input checked="" type="checkbox"/>	7	0.022321	1.2	0.049107	0.25
Blade Fillets	0.5	-	<input checked="" type="checkbox"/>	7	0.022321	1.2	0.049107	0.25
Inflow	4	-	<input type="checkbox"/>	7	0.022321	1.2	0.049107	0.25
Outflow	2	-	<input type="checkbox"/>	7	0.022321	1.2	0.049107	0.25

Information: Max. size at interfaces of adjacent components should have similar values.

OK Cancel ? Help

No messages

Global settings

Tetrahedra
☒ Coarse ☒ Middle ☒ Fine
 Global element scale factor: 1
 Global element seed size: 4
 Min. size limit: 0.5
 Edge criterion: 0.15

Prism layers
☒ Coarse ☒ Middle ☒ Fine
 Number of layers: 7
 Growth law: Linear
 Height ratio: 1.2

Local settings

Part	Max. size	Max. deviation	Layers	Height	Height ratio	End height	Total height	Height factor
Hub	4	1.1214	<input checked="" type="checkbox"/> 7	0.022321	1.2	0.049107	0.25	0.0625
Shroud	4	1.1214	<input checked="" type="checkbox"/> 7	0.022321	1.2	0.049107	0.25	0.0625
Blade	2	1.1214	<input checked="" type="checkbox"/> 7	0.022321	1.2	0.049107	0.25	0.125
Blade LE	1	0.74763	<input checked="" type="checkbox"/> 7	0.022321	1.2	0.049107	0.25	0.25
Blade TE	1	0.74763	<input checked="" type="checkbox"/> 7	0.022321	1.2	0.049107	0.25	0.25
Blade Fillets	0.5	-	<input checked="" type="checkbox"/> 7	0.022321	1.2	0.049107	0.25	0.5
Inflow	4	-	<input type="checkbox"/> 7	0.022321	1.2	0.049107	0.25	0.0625
Outflow	2	-	<input type="checkbox"/> 7	0.022321	1.2	0.049107	0.25	0.125

Possible warnings:



Max. size at interfaces of adjacent components should have similar values.

Outlet extension is recommended due to high mesh quality near the trailing edge.



Meridional extension behind trailing edge is missing. See "Extension-RSI" in "CFD setup".

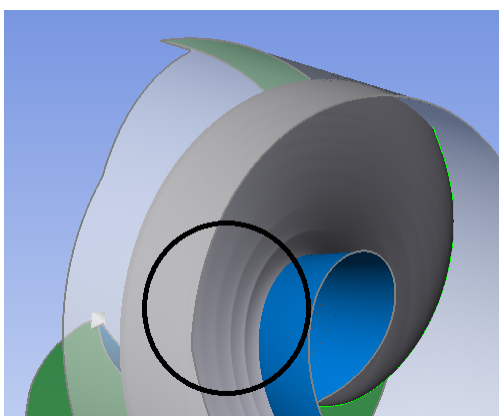
Cell size at the interface between neighboring components should be similar.

For more information about using CFturbo2ICEM please see the [available documentation](#).

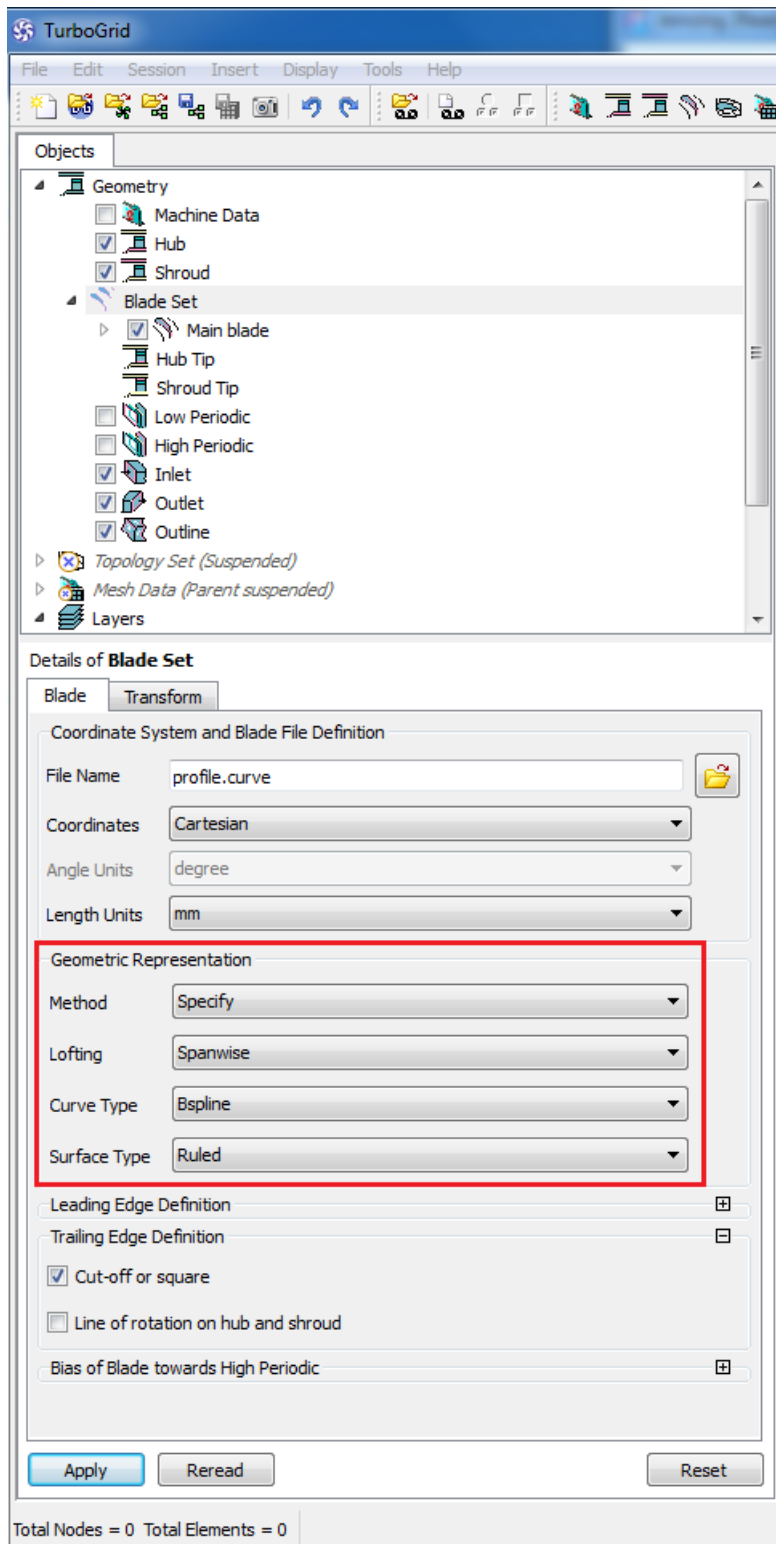
5.2.2.1.4.11 TurboGrid (ANSYS)

Troubleshooting

- Surfaces can be described in 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 TurboGrid object tree.



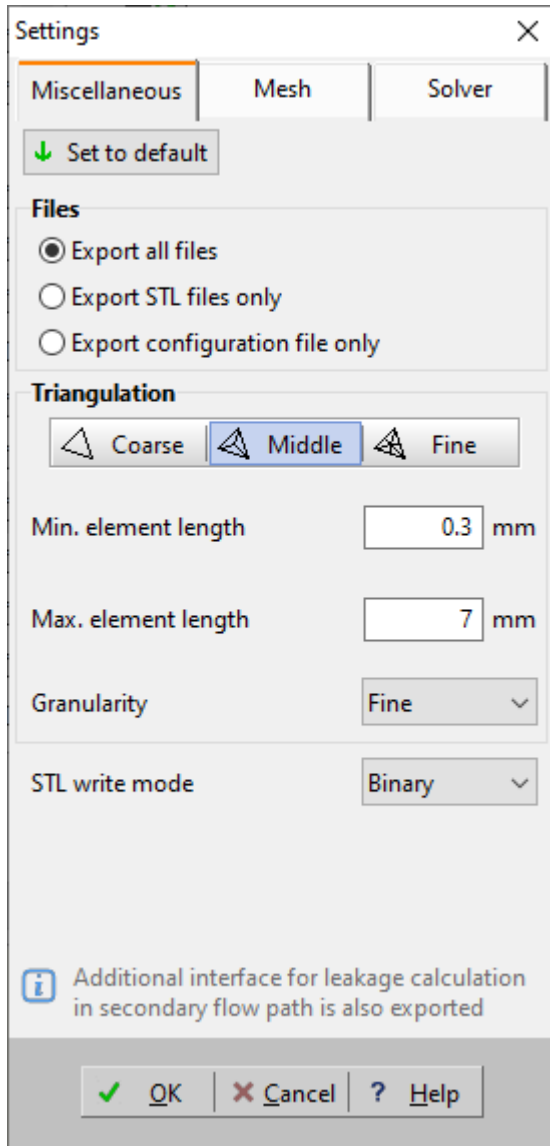
- For open impellers and stators, a small region between leading/ trailing edge and meridional inlet/ outlet could result in the following error message while importing in TurboGrid:
"Error extending the shroud tip line. Try reducing the "Tip expansion factor" value."

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](#)^[356]. 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](#)^[479]. This option does not incur a geometrical modification of the component but of the neighboring one if exists.

5.2.2.1.4.12 SimericsMP/ SimericsMP+ (Simerics)

Geometrical parameters



In addition to the [STL-Parameters](#)^{122]}, 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 dialog.

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 parameters

Settings

Miscellaneous | **Mesh** | Solver

↓ Set to default

Coarse | **Middle** | Fine

Size specif. Rel. to selected CAD surfaces ▾

☒ Refine cells next to boundary cells

Relative global cell sizes

Min. 0.0001 Max. 0.02

Cell size on surfaces 0.01

Rot. symmetric | **Volutes** | Refinement

☐ Use local cell size on surfaces

Inlet 0.01 Hub 0.01

Outlet 0.01 Shroud 0.01

2ndary flow path

Hub 0.01 Blade sides 0.01

Shroud 0.01 Blade LE 0.01

Blade TE 0.01

☐ Refine casing

OK Cancel Help

Mesh parameters are set globally for all selected CFturbo components to be exported.

The following global mesh parameters are available:

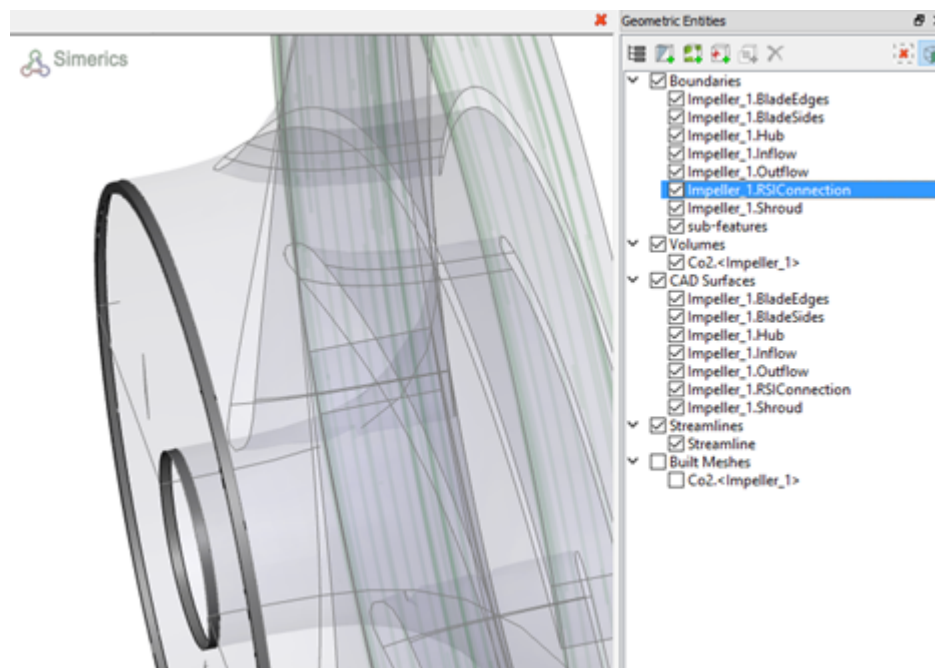
- Min. cell size
- Max. cell size
- Cell size on surfaces

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:

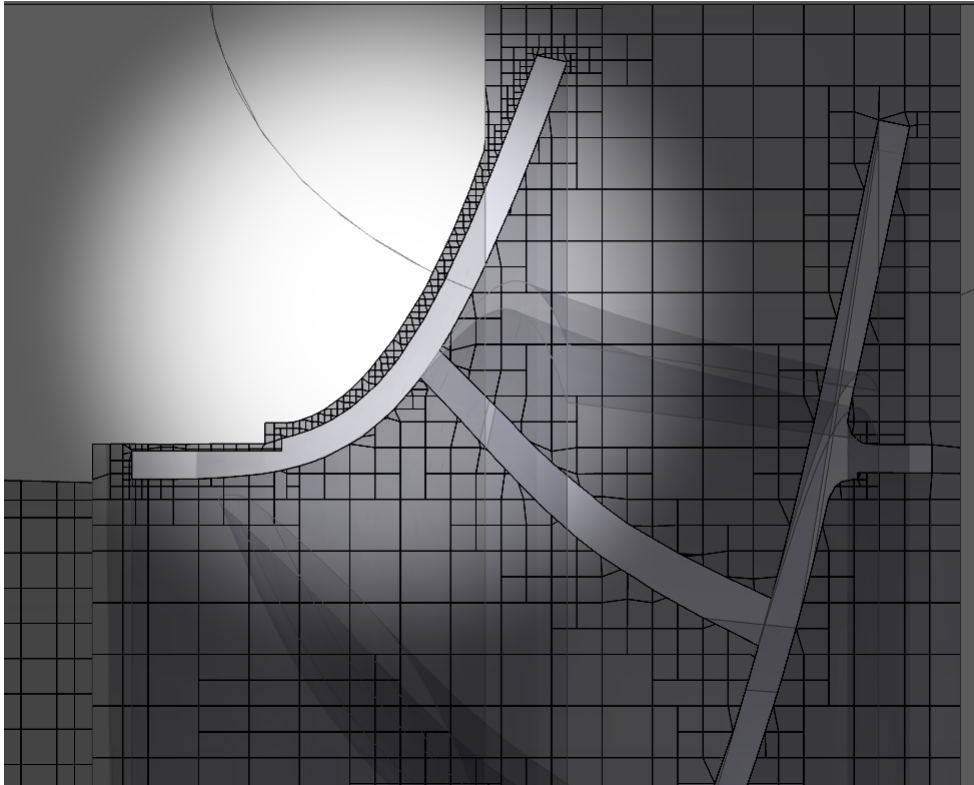
Rotational symmetric components	
Region	Description
Hub	-
Shroud	-
Blade sides	Blades suction and pressure sides
Blades LE / TE	Blades leading and trailing edges
Secondary flow path hub	Hub material domain surfaces
Secondary flow path shroud	Shroud material domain surfaces
Inlet	-
Outlet	-

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



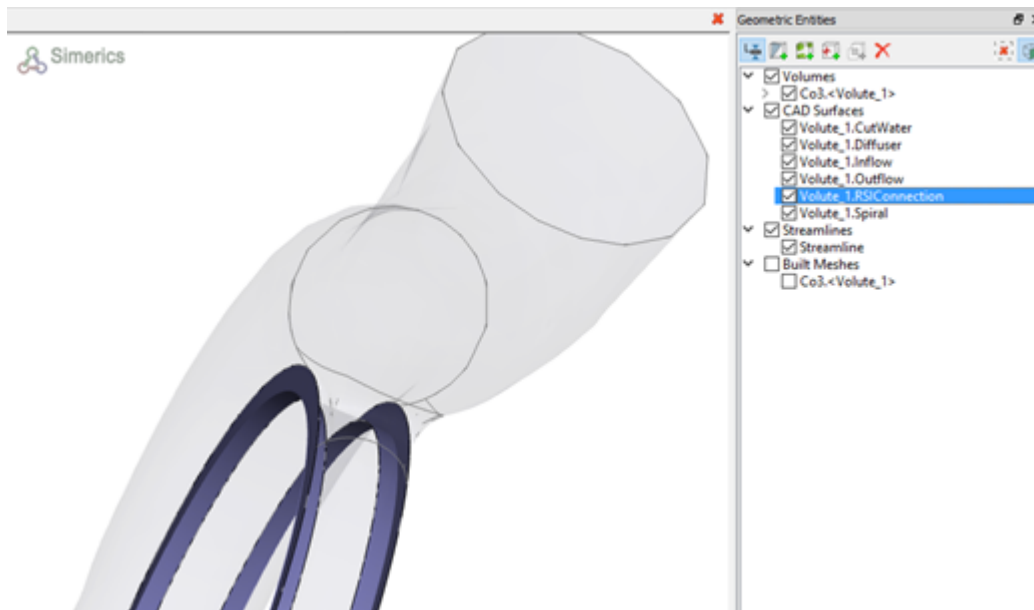
Secondary flow path mesh parameters are useful if a fine meshing is necessary in gap zones between solid bodies and casing.

Enabling the option "Refine casing", local cell sizes set for hub and shroud are also used for the casing surfaces.



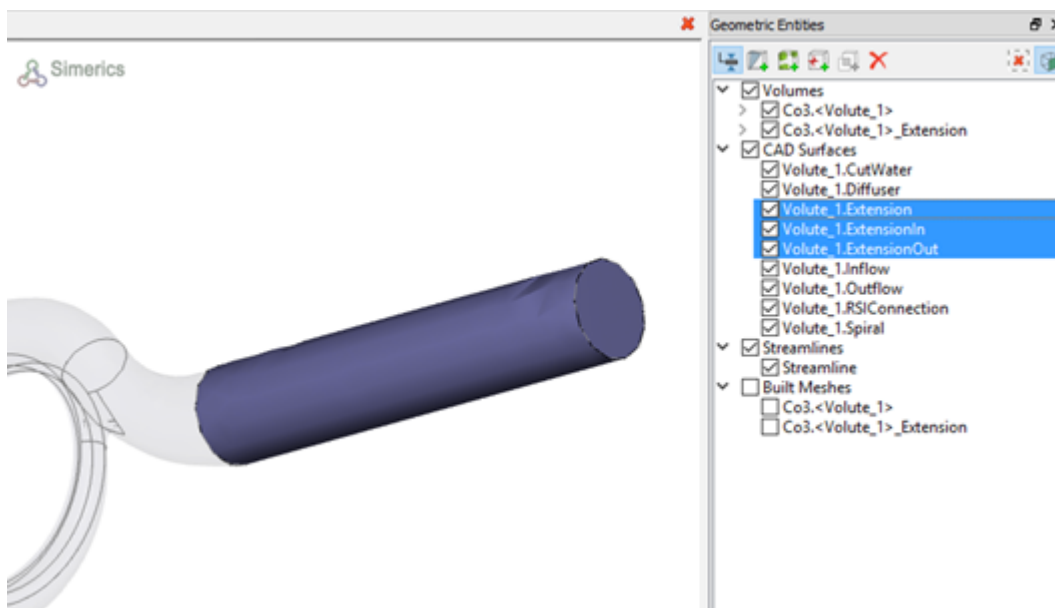
Volute	
Region	Description
Spiral	-
Diffuser	-
Cut-water	-
Splitter	Volute splitter (double volutes)
Inlet	-
Outlet	-

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



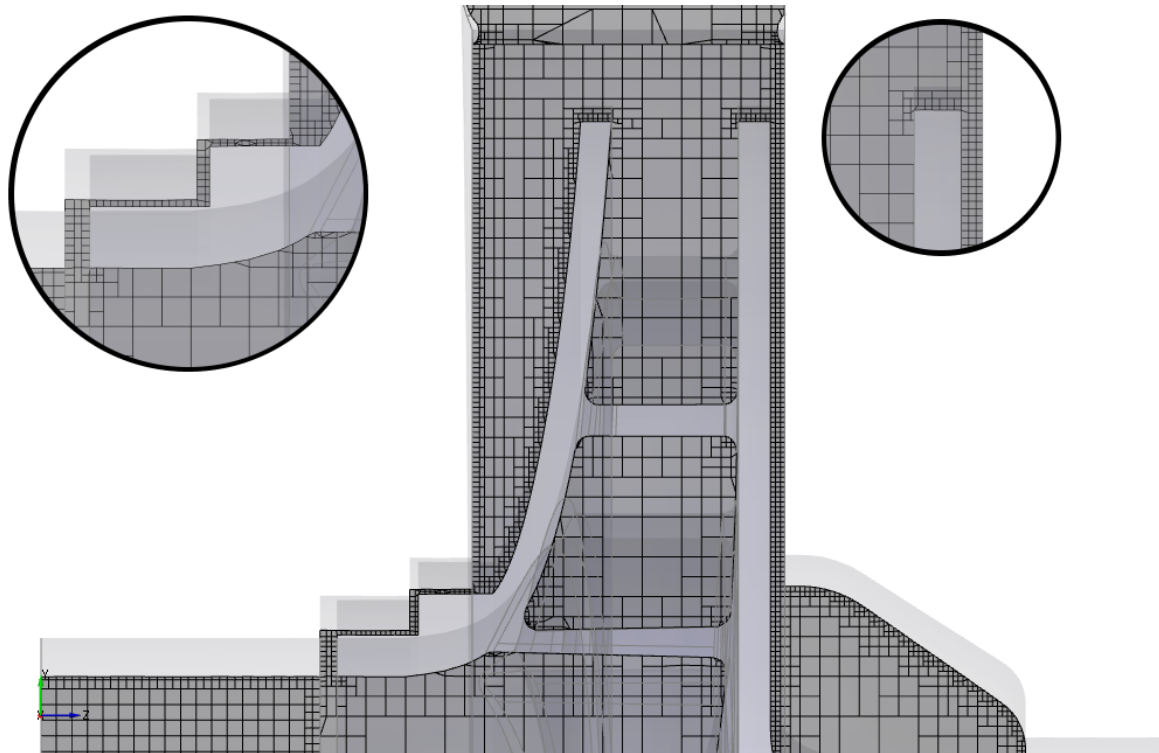
In case the volute extension is exported, “Diffuser” mesh parameters are used for the extension walls.

“Inlet” and “Outlet” mesh parameters are used for the extension inlet and outlet:

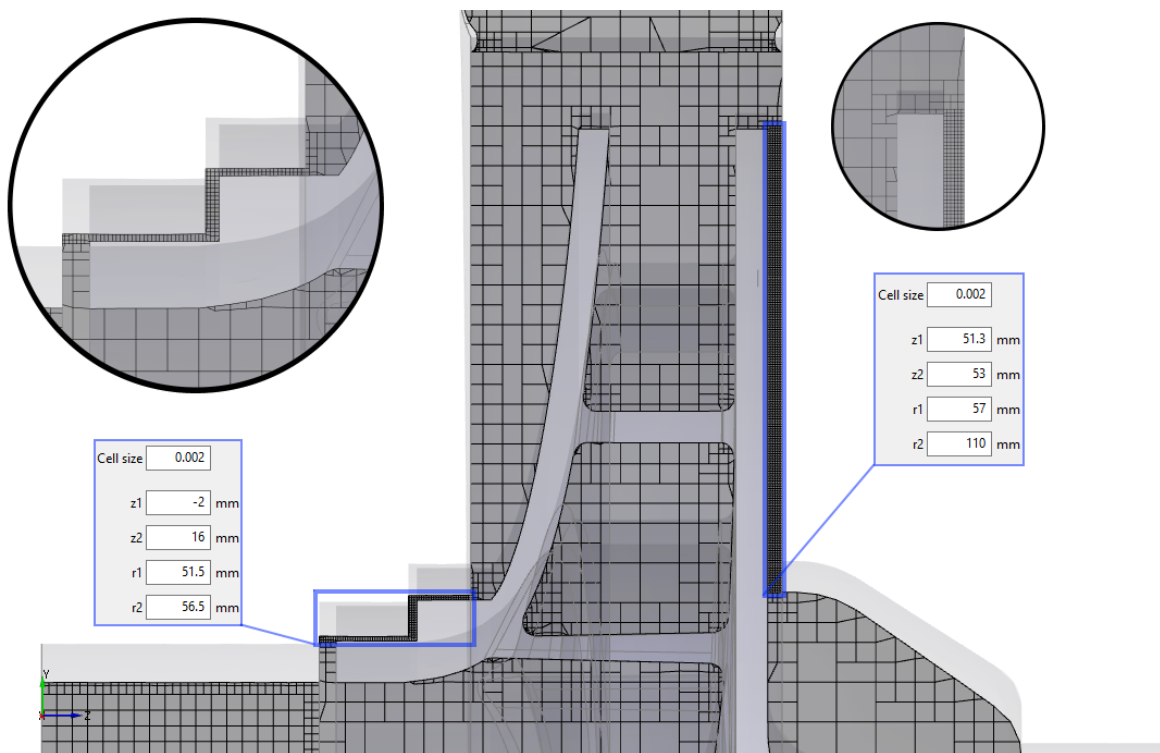


Refinement zones

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:

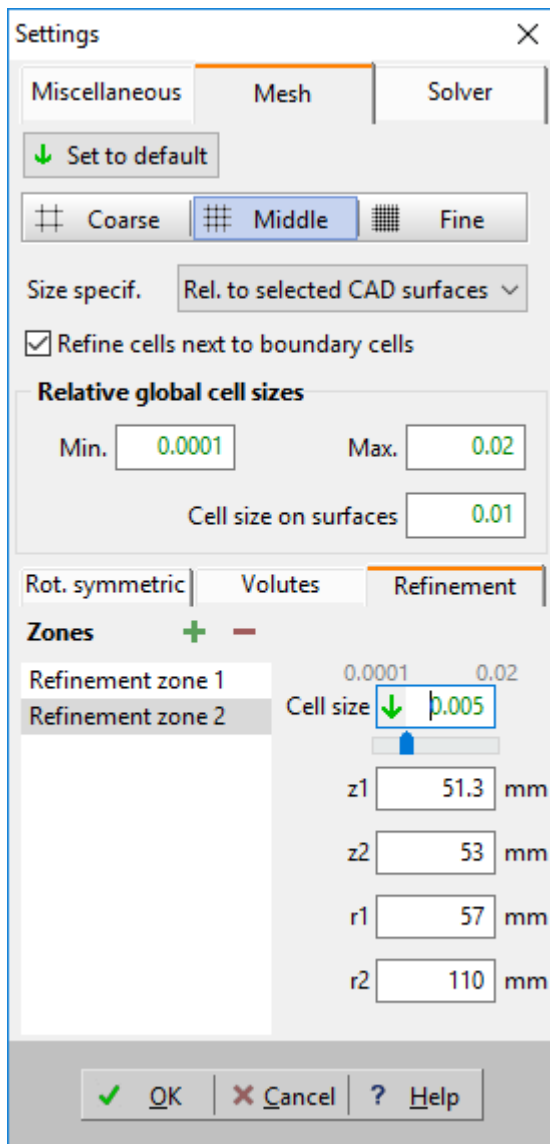


Mesh resulting from setting local mesh parameters on hub and shroud as well as casing surfaces



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 parameters

The following parameters are available:

Settings

Miscellaneous Mesh **Solver**

↓ Set to default

Simulation

SimericsMP Transient

Number of iterations 25

Number of revolutions 3

Rotation angle per step 3.0 °

Result saving frequency 120

Converge criterion 0.1

Numeric Scheme / Relaxation

Velocity 2nd Order Upwind 0

Pressure Upwind 0

☒ Activate cavitation module

☐ Reversible pump turbine

☒ Export turbomachinery expressions

OK Cancel Help

- **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.

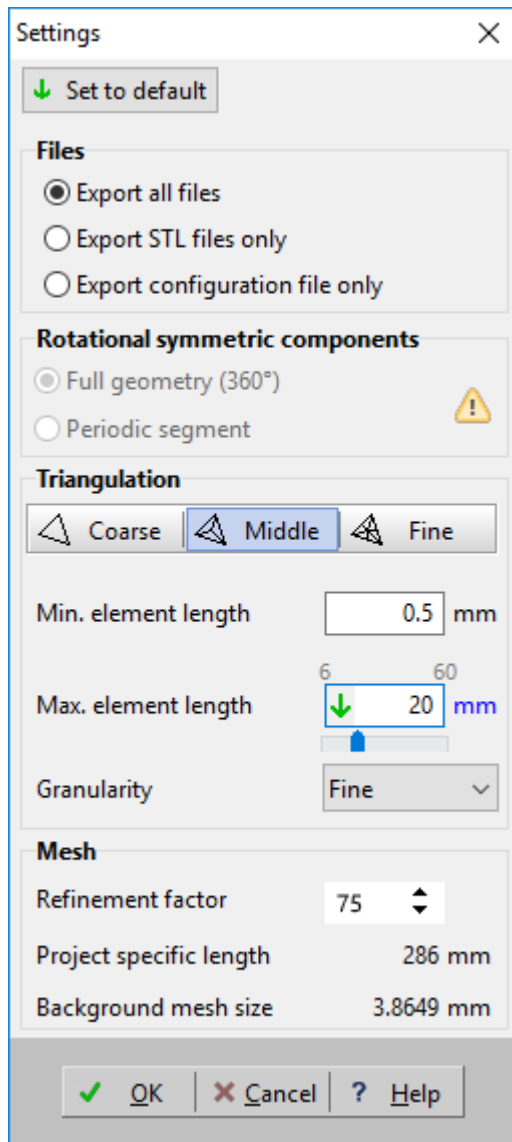
- A global value for **converge criterion** can be set. This value is used for all active modules. In case the default Simerics value (0.001 for steady simulations and 0.1 for transient simulations) is used, values are not written in the SPRO-file allowing Simerics to change the values automatically 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, ventilators)
- **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.

5.2.2.1.4.13 TCFD (CFD Support)

In addition to the [Triangulation-Parameters](#)^[122], the following parameters are available:



Full geometry or periodic segment

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

Files parameters

Allows to define which files should be exported.

Export all files: Configuration file (*.tcf) 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 dialog.

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.

Not supported characters

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

5.2.2.1.5 Data 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 / Test license

Export functionality can be restricted when using CFturbo with a Demo/Test license.

Data export can then be disabled for all individually designed components. To demonstrate the performance of the CAD/CFD interfaces, the data export is enabled for CFturbo default examples only. These default examples you can find in the CFturbo installation directory, sub-directory **Examples**.

5.2.2.2 Batch mode/ Optimization

? PROJECT | 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](#)³².

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.

Batch mode / optimization

Working directory: C:\daten\Software\CFturbo\Examples\PUMP - Radial, Mixed-flow\

Load Save Save as...

Parameters Export actions

File name of modified cfturbo project: RP nq49 volute (generic_modified.cft)

Expand to level: 1 2 3 all Select nodes: ☒ All ☐ None Print Copy

NAME	DESCRIPTION	SYMBOL	UNIT	VALUE	CONSTRAINTS	RANGE
PROJECT: RP nq49 volute (generic)						
Global setup						
Design Point						
Volume flow	Volume flow	Q	[m³/h]	454		
Revolutions	Rotational speed	n	[/min]	1770		
Head	Head	H	[m]	30		
1: Pipe in						
Main dimensions						
Setup						
With blades	Vaned/ vaneless stator			-		
Extent						
Hub extent	Hub extent $\Delta z, \Delta r$ (inlet → outlet)		[mm; ...]	[230; 0]		
Shroud extent	Shroud extent $\Delta z, \Delta r$ (inlet → ou...)		[mm; ...]	[230; 0]		
Inlet						
Hub point	End point position on hub z, r		[mm; ...]	[-230; 0]		
Shr point	End point position on shroud z, r		[mm; ...]	[-230; 94]		
Outlet						
Width	Distance between hub and shro...	b	[mm]	94		
Angle	Angle to axial direction	γ	[°]	90.0		
Offset center	Offset Center line $\Delta z, \Delta r$		[mm; ...]	[0; 0]		
Meridional contour						

Close Help No messages

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)

Min ... Max

Values

Min. value

260

...

Max. value

320

mm

Resolution

☒ All values

☐ Steps

10

OK Cancel Help

Min ... Max

Values

1

260

2

280

3

300

4

5

6

-

OK Cancel Help

or

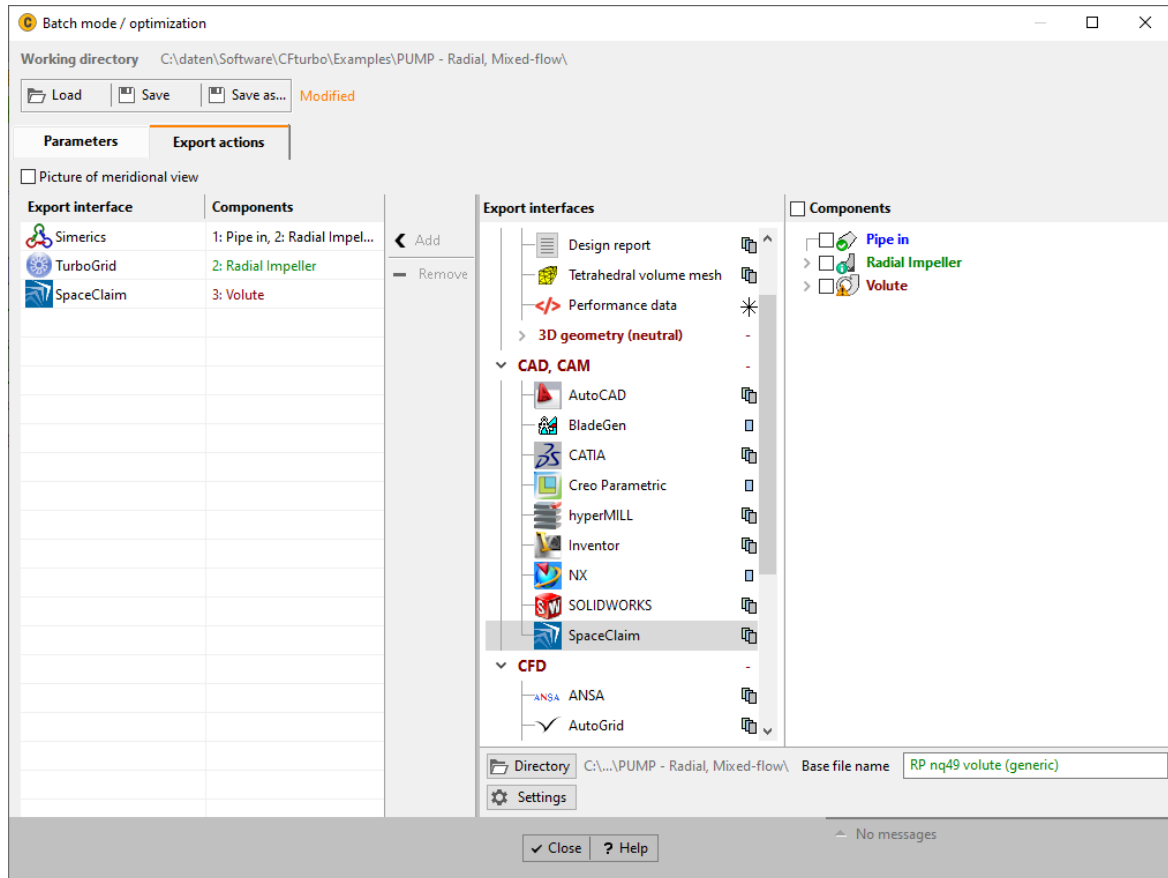
- using the following syntax:

Resolution	Range
Continuous	<Min. value>:<Max. value>
Discrete	<Min. value>:<Max. value>:Count> or <Value1>,<Value2>,<Value3>...

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](#) ¹⁰³ dialog) and pressing the **Add** button.



Possible warnings

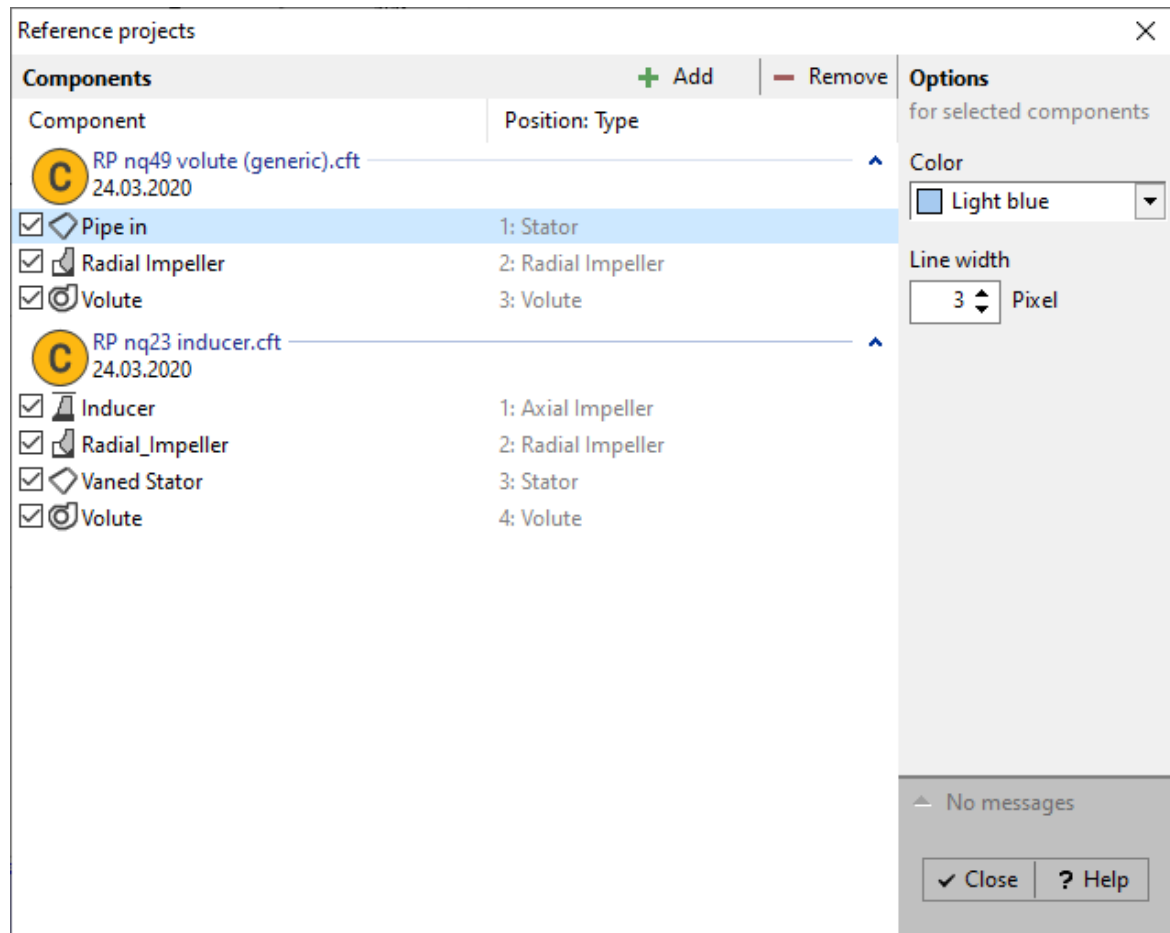
Problem	Possible solution
Export requirements are not fulfilled.	
Some of selected components no longer fulfill the export requirements.	Remove or replace the affected export actions.
Component selection already used.	
Applicable only in ANSYS Workbench mode. At least one selected component was already used for the same export interface in previous export actions. The critical export item is highlighted in red.	Remove affected export actions.

5.2.2.3 Reference components

? PROJECT | Reference components



This functionality can be used for simultaneous display of various designs to compare each other and for purposeful modification.



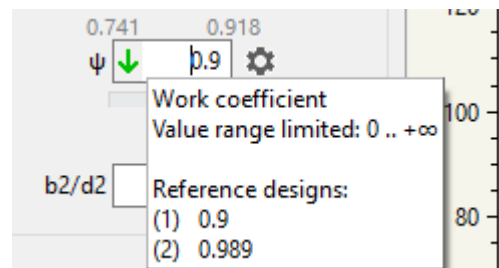
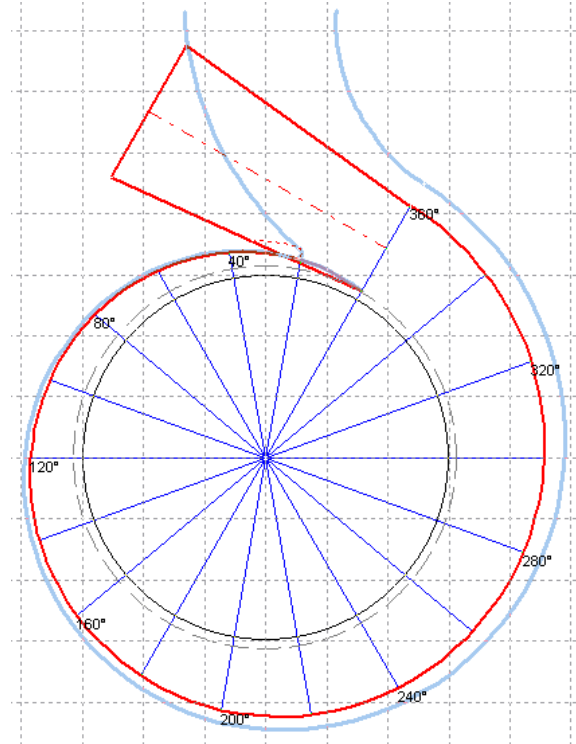
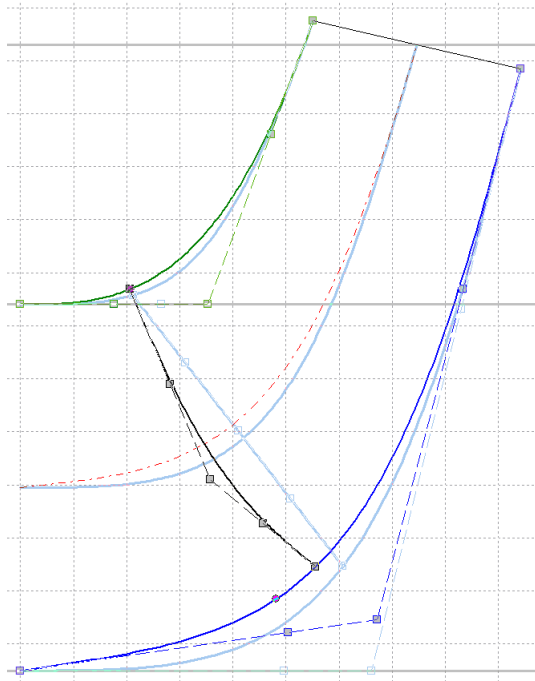
Using the **Add**-button any reference project (*.CFT- file) can be added. All components of the reference project are grouped under the selected file name.

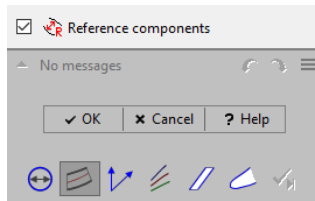
Each component has its own color and line width (panel **Options**). Multiple components can be selected using <Shift> and <Ctrl> keys. Clicking on the group header area selects all components of the corresponding project, <Ctrl> <A> selects all components.

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.





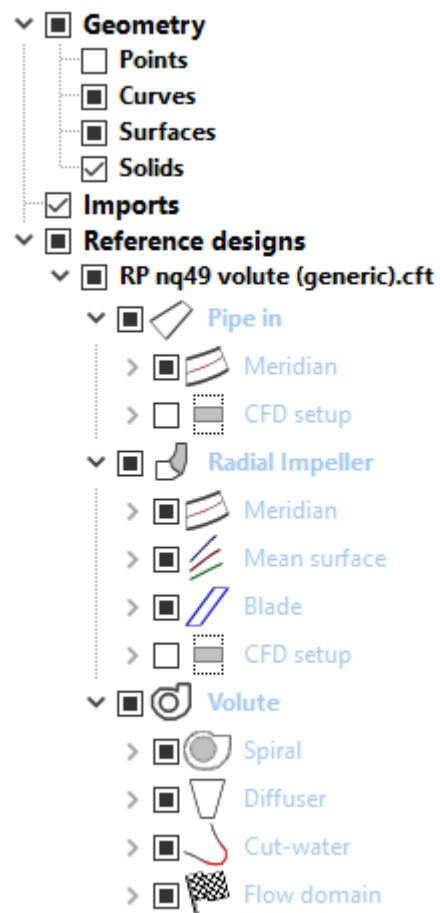
Down right in the design step dialog windows you could completely switch off the display of reference geometries and start the configuration dialog.

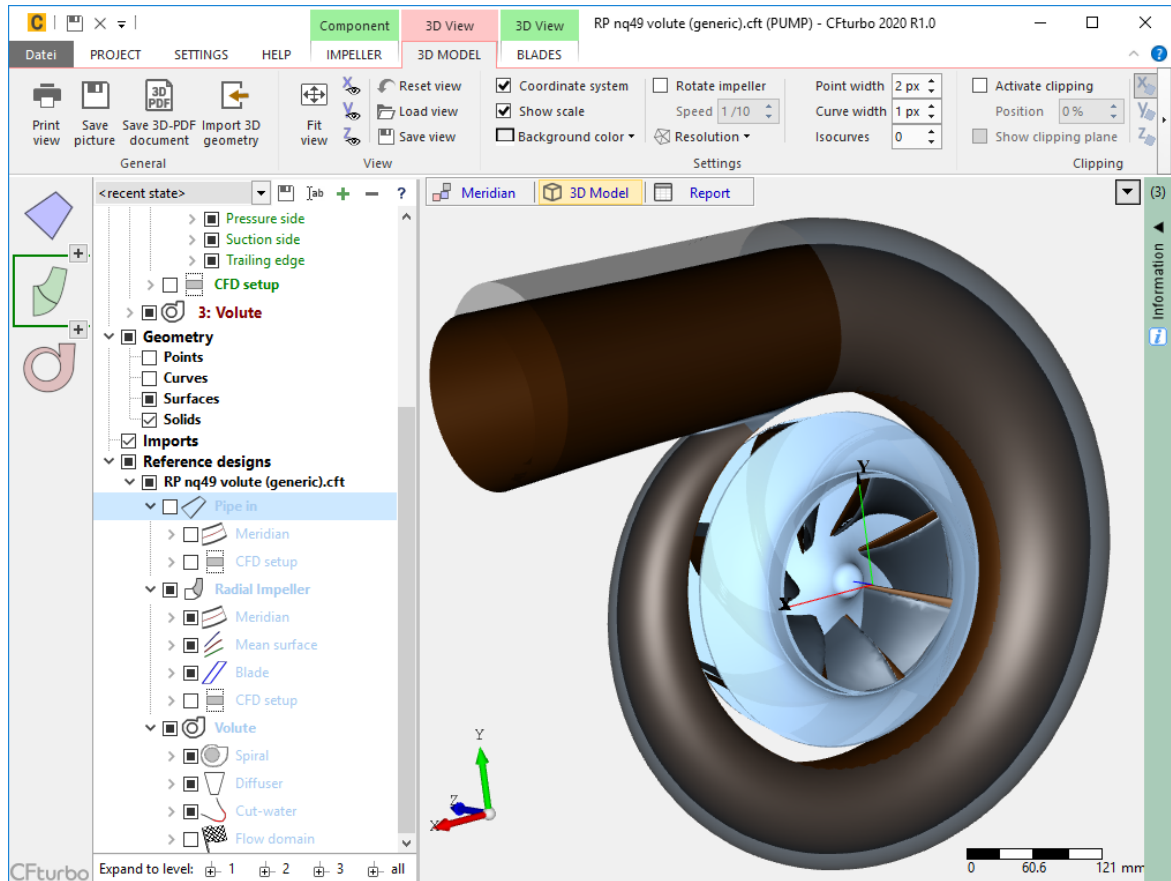
Please note: If you add reference designs in a design step dialog the imported geometry could be invisible initially if it's far away from the currently designed geometry. There is no automatic scaling of the diagram.

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

? PROJECT | Model finishing

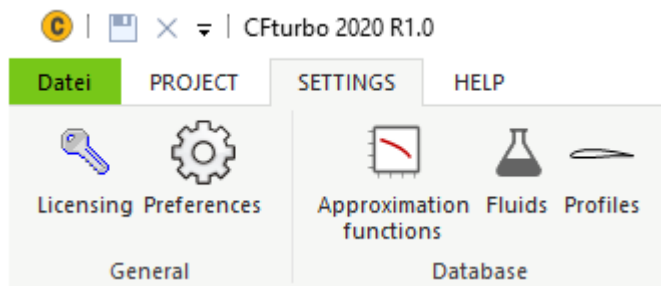
Model finishing is started globally for all components of the project.

Each vanned component has their own model finishing configuration, see [Model finishing for impeller](#) ⁴⁸⁷.

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](#) ¹⁸⁵

→ [Preferences](#) ¹⁸⁵

→ [Approximation functions](#) ¹⁹⁸

→ [Fluids](#) ²⁰¹

→ [Profiles](#) ²¹⁰

5.3.1 Licensing

? **SETTINGS | General | Licensing**



See [General/ Licensing](#) ¹⁸⁵

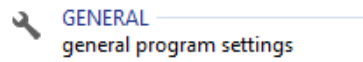
5.3.2 Preferences

? **SETTINGS | Preferences**



This menu item is used for global program options. Alternatively it can be accessed by the [File](#) ⁷⁵ menu.

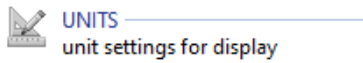
The dialog contains several topics on the left side and the corresponding settings on the right side.



Global settings

Mouse handling

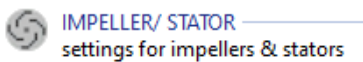
Program updates



Basic units

Specific speed

Other units



Progression diagrams

Initial default settings

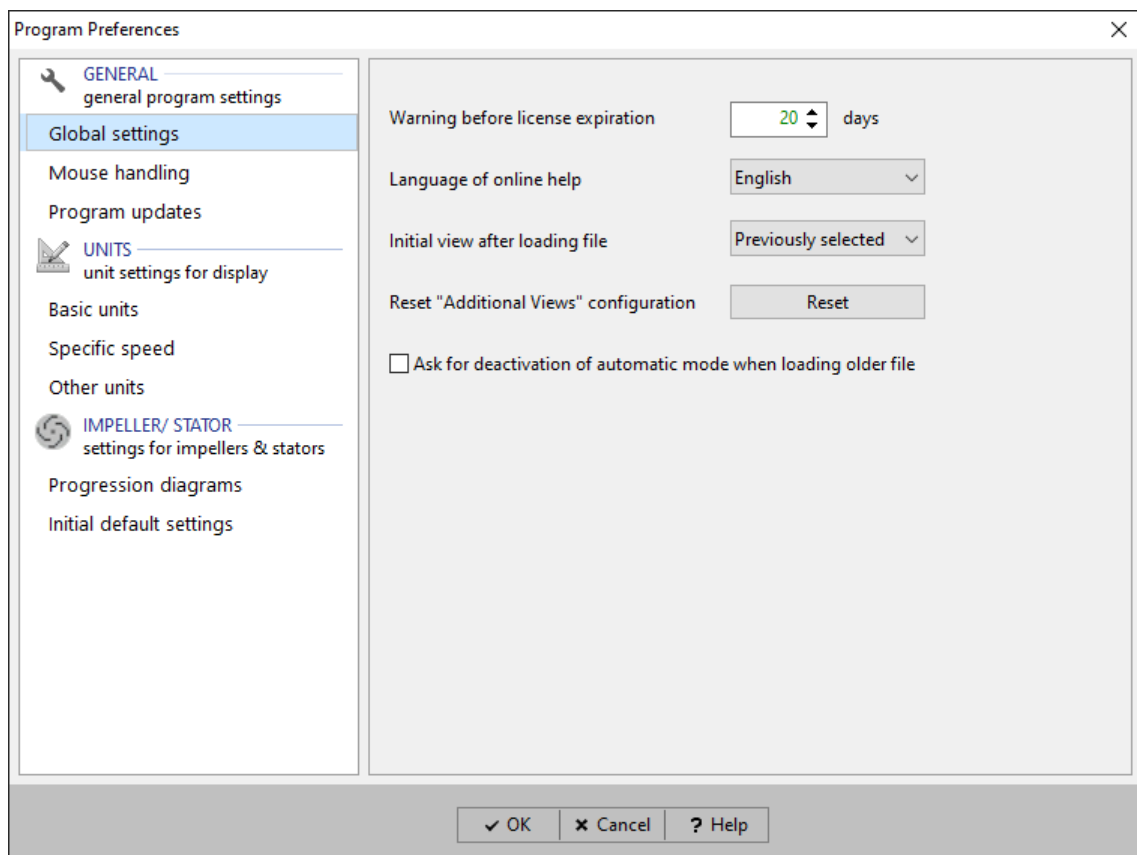
There are 3 main groups of topics available on the left side:

- [General](#)¹⁸⁶
- [Units](#)¹⁹⁰
- [Impeller/ Stator](#)¹⁹⁶

5.3.2.1 General

? SETTINGS | Preferences | General

Topic **GENERAL** is used for global program options.



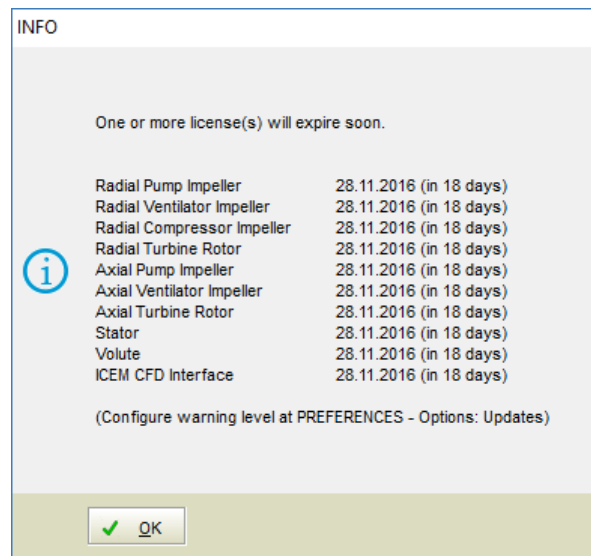
Language of online help

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

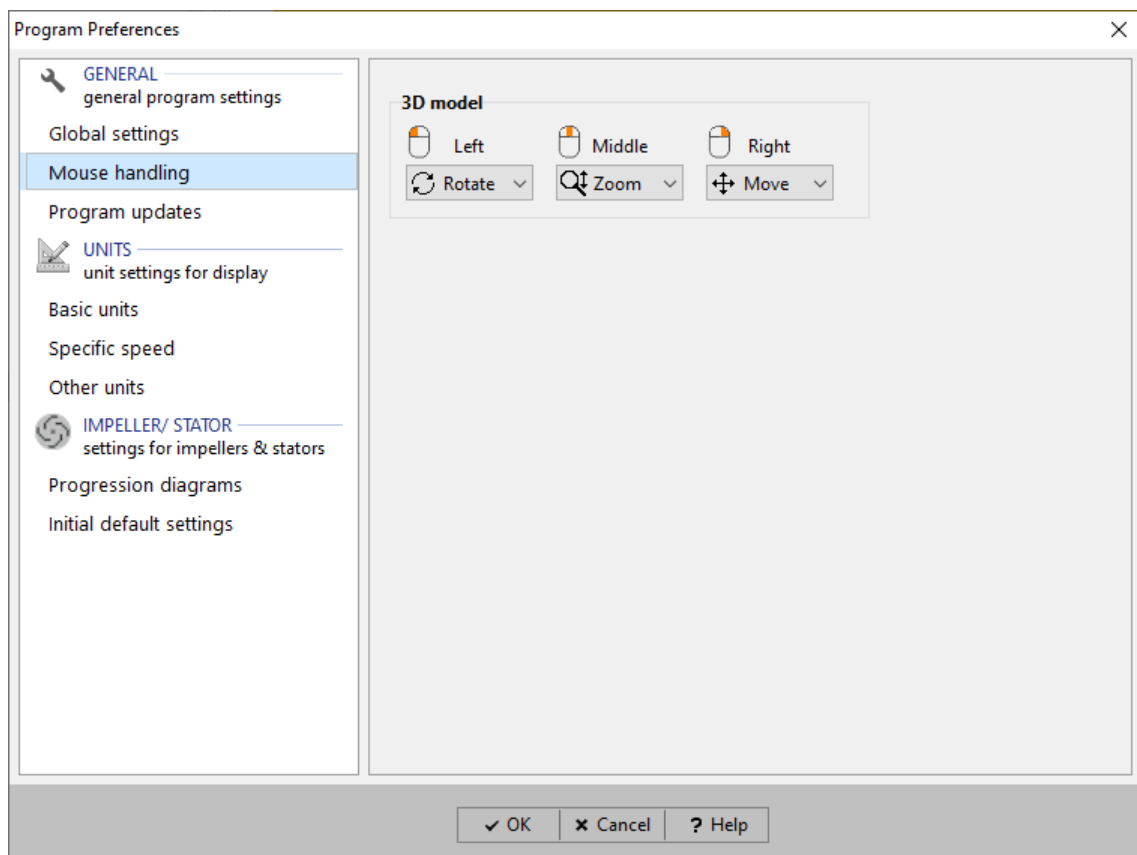
Select which view should be displayed after file loading. Choosing the *3D Model* will increase the time needed for loading, because the model gets updated first.

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](#) ⁴⁴.

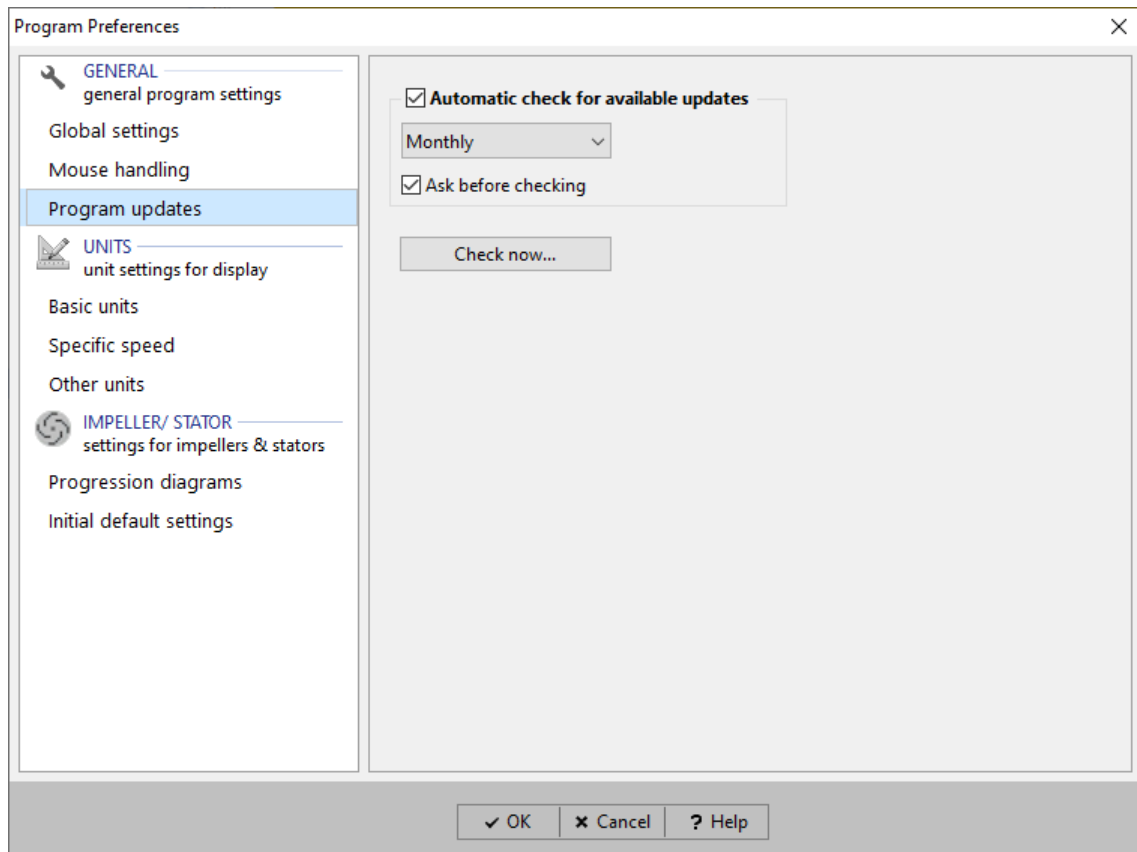


3D model mouse handling

Here you can assign functions (Rotate, Zoom, Move) to the mouse buttons (Left, Middle, Right) for handling the [3D model](#)²²⁵.

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](#)^[216]). 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.

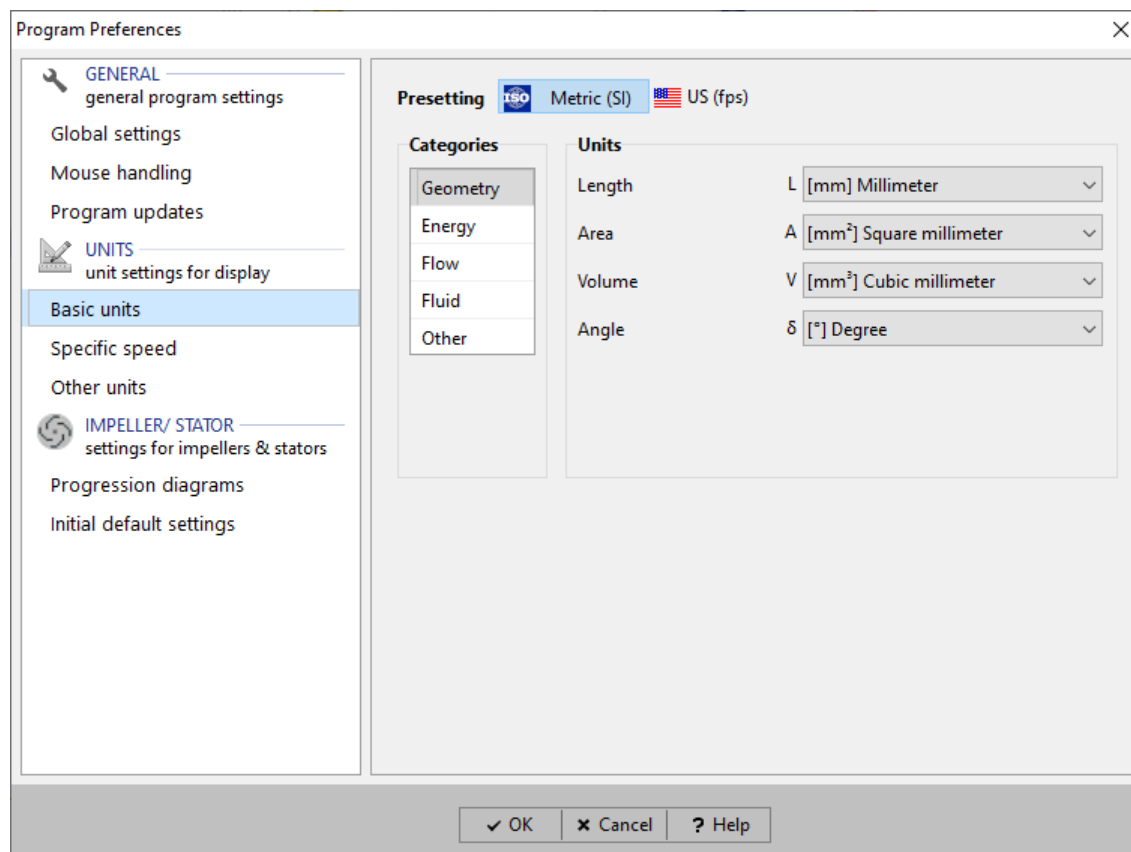
It's divided in 3 parts:

- [General](#)^[191]: general unit selection
- [Specific speed](#)^[192]: selecting a suitable specific speed definition

→ [Other](#)¹⁹⁵: some additional unit settings, like flow/blade angle and n_{ss} definition

5.3.2.2.1 General

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



Following grouped units are available:

Category	Unit	Symbol	available
Geometry	Length	L	mm, in, m
	Area	A	mm ² , m ² , in ²
	Volume	V	mm ³ , m ³ , in ³
	Angle		°, - (radian)

Energy	Specific energy, Enthalpy	Y, h	m ² /s ² , J/kg, Nm/kg, Ws/kg, ft ² /s ² , BTU/lb
	Head	H	m, ft
	Pressure	p	MPa, PSI, bar, Pa, mm H2O, mm Hg, in H2O, ft H2O
	Stress		MPa, PSI
	Temperature	T	°C, K, °F
	Power	P	W, kW, hp
Flow	Volume flow	Q	m ³ /h, m ³ /min, m ³ /s, l/min, l/s, ft ³ /min, ft ³ /s, gpm, gps
	Mass flow	m ³	kg/s, lb/s
	Velocity	v	m/s, ft/s
Fluid	Density	ρ	kg/m ³ , lb/ft ³
	Dynamic viscosity	μ	Pa·s, cP
	Kinematic viscosity	ν	m ² /s, ft ² /s
Other	Ratio	x/y	%, - (absolute)
	Revolutions	n	/min, /s
	Mass	m	kg, lb
	Time	t	s, m, h

You can simultaneously change all units to SI or US system by pressing the buttons above.

5.3.2.2.2 Specific speed

Here the specific speed definition can be selected. This definition is mainly used for the [Approximation functions](#) ^[198].

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

Program Preferences

GENERAL
general program settings

Global settings

Mouse handling

Program updates

UNITS
unit settings for display

Basic units

Specific speed

Other units

IMPELLER/ STATOR
settings for impellers & stators

Progression diagrams

Initial default settings

Definition

$$n_q = n \frac{Q^{1/2}}{([g]H)^{3/4}}$$

ISO n_q^* ISO ω_s ISO σ

EU n_q US N_s ASIA n_q'

Revolutions n [/min] per minute

Flow rate Q [m³/s] Cubic meter/sec.

Head H [m] Meter

Information

Factor on dimensionless value: 332.59

Typical range

Radial	Mixed-flow	Axial
10...40	40...140	140...400

✓ OK ✗ Cancel ? Help

Following definitions are available:

- General specific speed n_q^* (dimensionless)

$$n_q^* = n \frac{Q^{1/2}}{Y^{3/4}}$$

- Type number n_s (dimensionless)

$$\omega_s = n_s = 2\pi n \frac{Q^{1/2}}{Y^{3/4}}$$

- Speed coefficient

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

- European definition n_q

- US definition N_s

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

- Asian definition n_q'

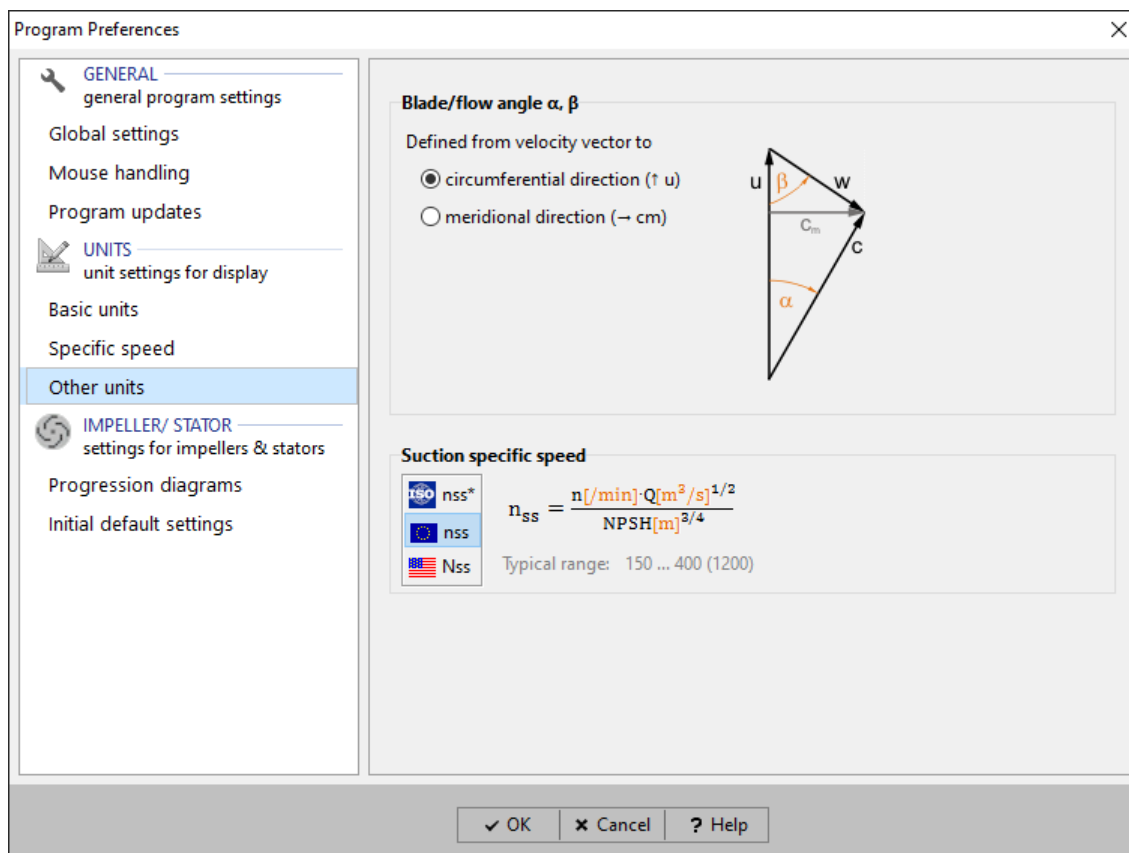
$$n_q = n[\text{min}^{-1}] \frac{Q[\text{m}^3/\text{min}]^{1/2}}{H[\text{m}]^{3/4}}$$

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

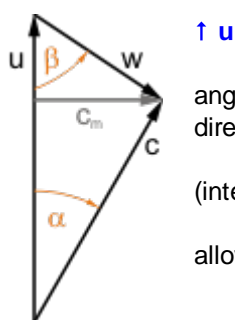
On the bottom side some information for the currently selected specific speed definition is displayed. The **Factor on dimensionless value** is the factor used to convert the General specific speed n_q^* to the currently selected definition. Furthermore the **Typical range** of the specific speed definition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

5.3.2.2.3 Other

Here some additional unit settings can be selected.



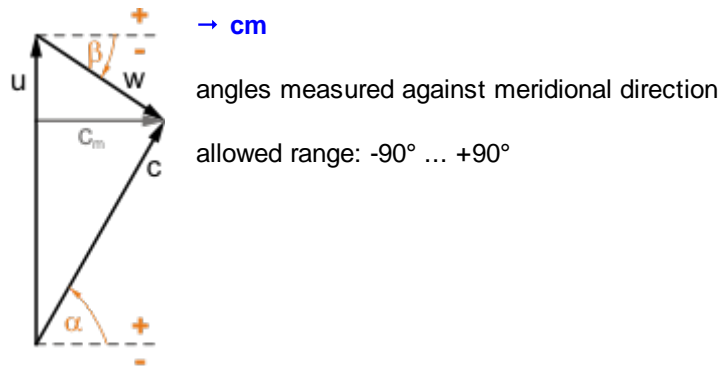
Blade/flow angle α ,



angles measured against circumferential direction

(internal angles of the velocity triangle)

allowed range: 0° ... 180°



Suction specific speed

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

- SI definition (dimensionless) n_{ss}^*

$$n_{ss}^* = n \frac{Q^{1/2}}{(g \cdot NPSH)^{3/4}}$$

- European definition n_{ss}

$$n_{ss} = n [\text{min}] \frac{Q [\text{m}^3/\text{s}]^2}{NPSH [\text{m}]^{3/4}}$$

- US definition N_{ss}

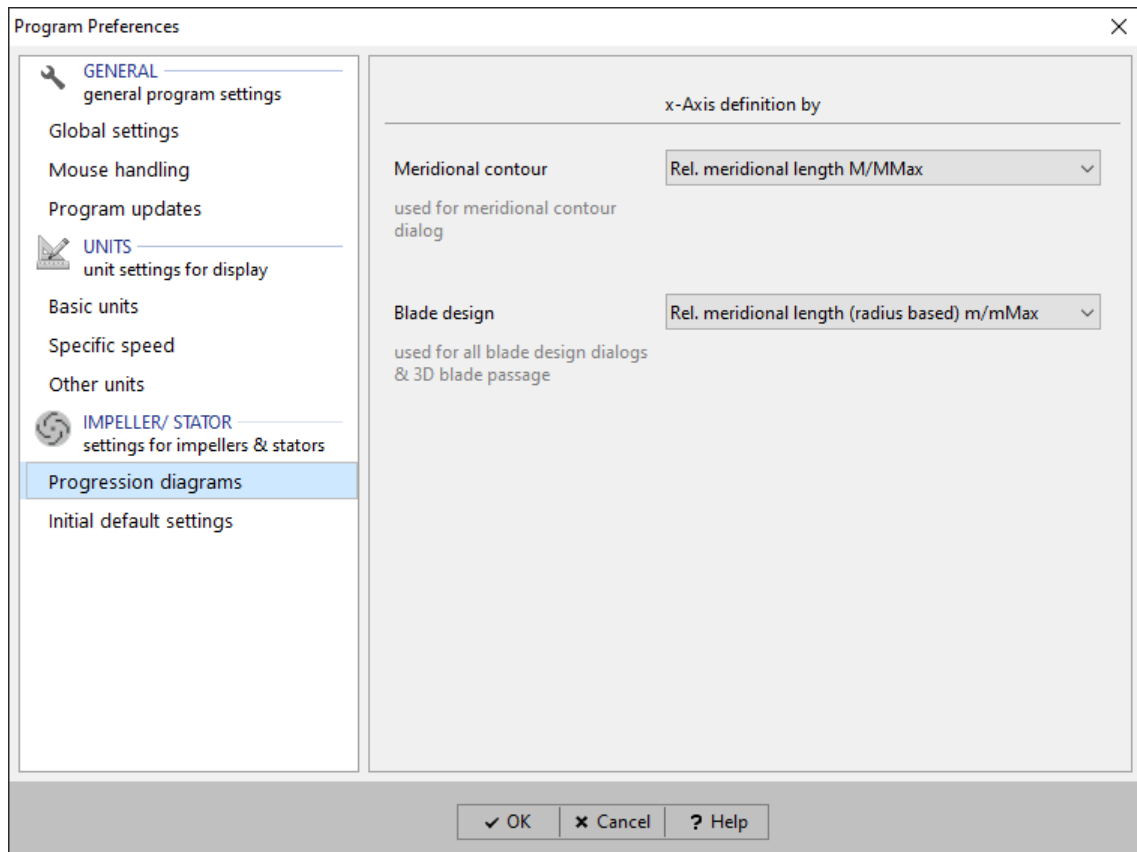
$$N_{ss} = n [\text{rpm}] \frac{Q [\text{gpm}]^2}{NPSH [\text{ft}]^{3/4}}$$

5.3.2.3 Impeller/ Stator

? SETTINGS | Preferences | Impeller/ Stator

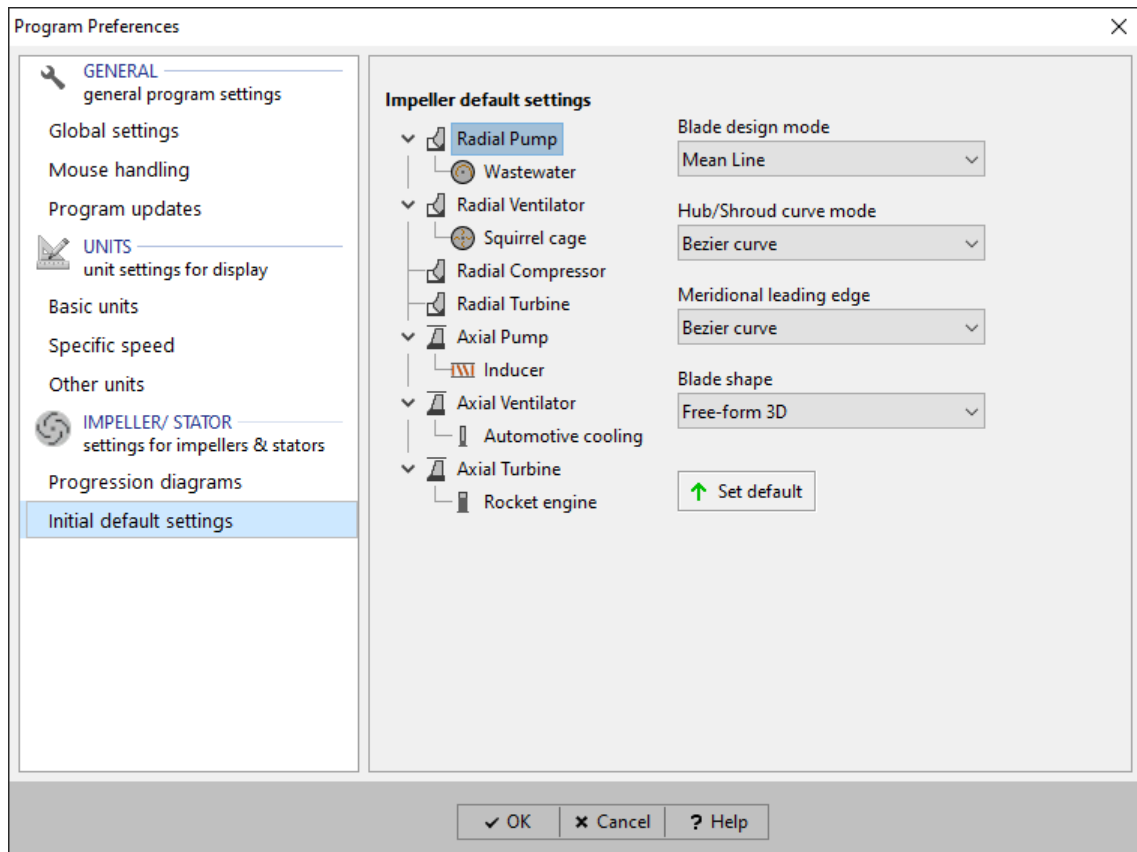


Topic **IMPELLER/ STATOR** is used for global default definition for impellers and stators. These settings are set at the initial opening of each dialog.



At **Progression diagrams** one can specify, which parameter should be used for the x-axis of the progression diagrams in the [Meridional contour](#)^[338] and Blading dialogs ([Meanline](#)^[405], [Profiles](#)^[444], [Edges](#)^[447], [3D cross section](#)^[232]). Some constellations may yield undefined x-values due to reference (e.g. r_{Max} , z_{Max}) values that are zero. Those constellations will be marked in the diagrams. One should use another option in such a case.

- abs. meridional length M
- rel. meridional length M/M_{Max}
- abs. radius based meridional length m
- rel. radius based meridional length m/m_{Max}
- abs. radius r
- rel. radius r/r_{Max}
- abs. axial length z
- rel. axial length z/z_{Max}



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, Ventilator, Compressor, Turbine), separately for each impeller subtype.

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

5.3.3 Approximation functions

? SETTINGS | Database | 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 subtypes:

- Axial Pump Impeller
 - Standard
 - Inducer
- Axial Turbine Rotor
 - Standard
 - Rocket Engine
- Axial Ventilator Impeller
 - Standard
 - Automotive Cooling
- Radial Compressor Impeller
- Radial Pump Impeller
 - Standard
 - Wastewater
- Radial Turbine Rotor
- Radial Ventilator Impeller
- Stator
- Volute

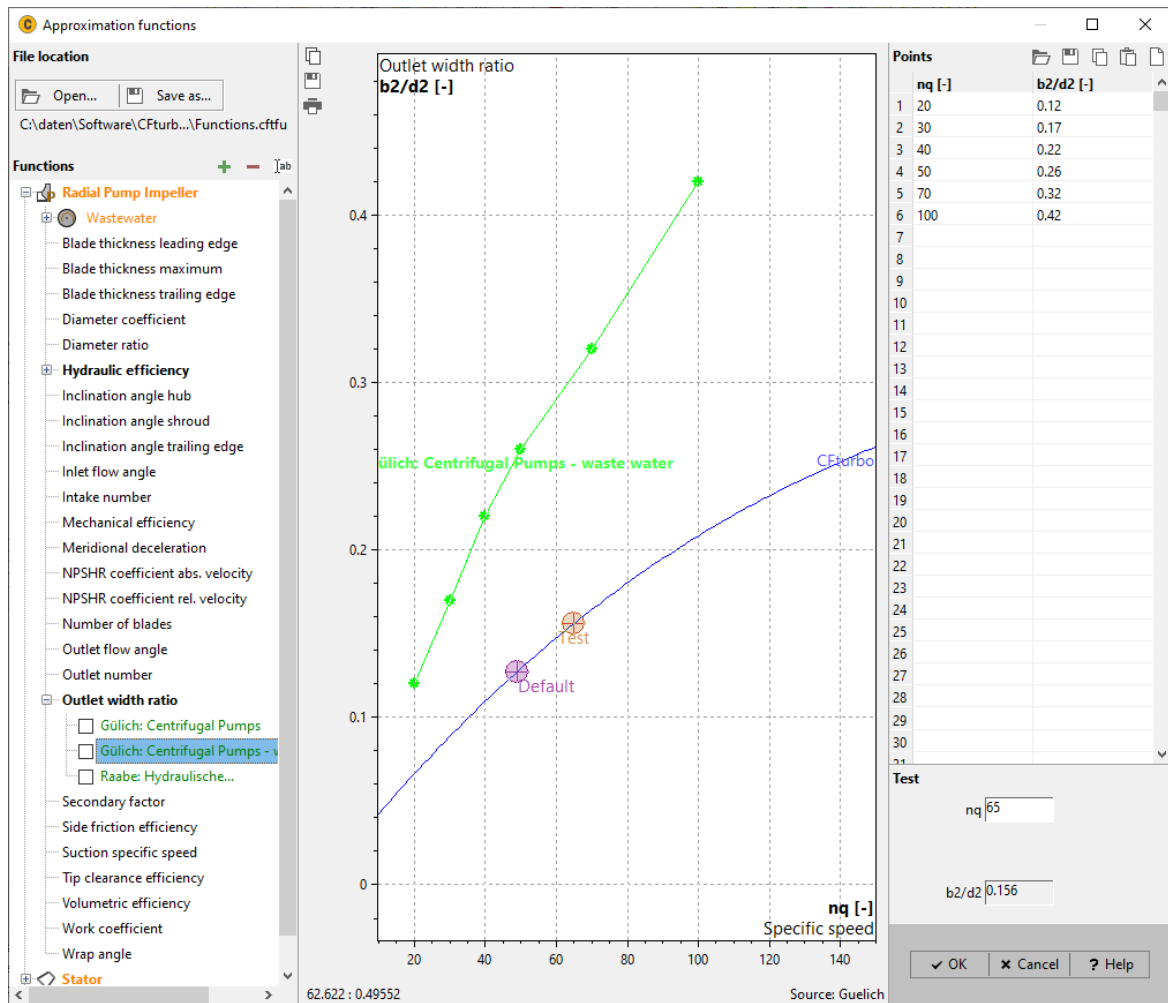
Each function has a hard coded default function. For each of these functions custom point wise defined curves can be added alternatively. These custom defined curves are saved in the file **Functions.cffu** that contains the custom defined functions only. The default functions are not saved in any external 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.cffu**, 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.

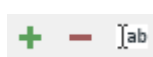
The function file will not be overwritten by any update. By default the functions file is located in the CFturbo installation directory. When you define any user-defined functions it's recommended to save the functions file not in the CFturbo installation directory but anywhere in the company network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



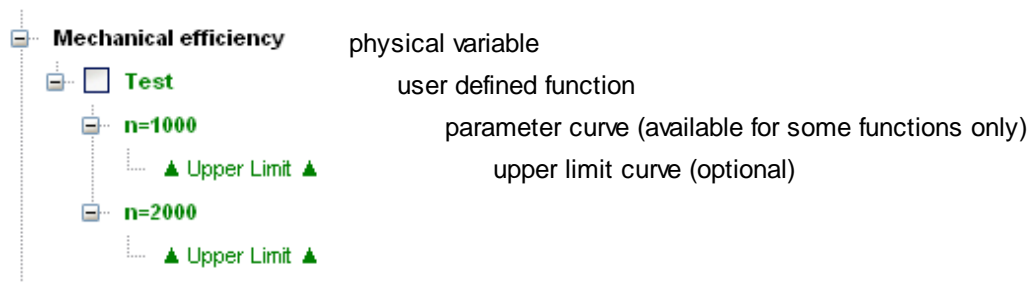
All available functions are listed in a tree structure in the panel **Functions** left from diagram, sorted by machine type.

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:



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:

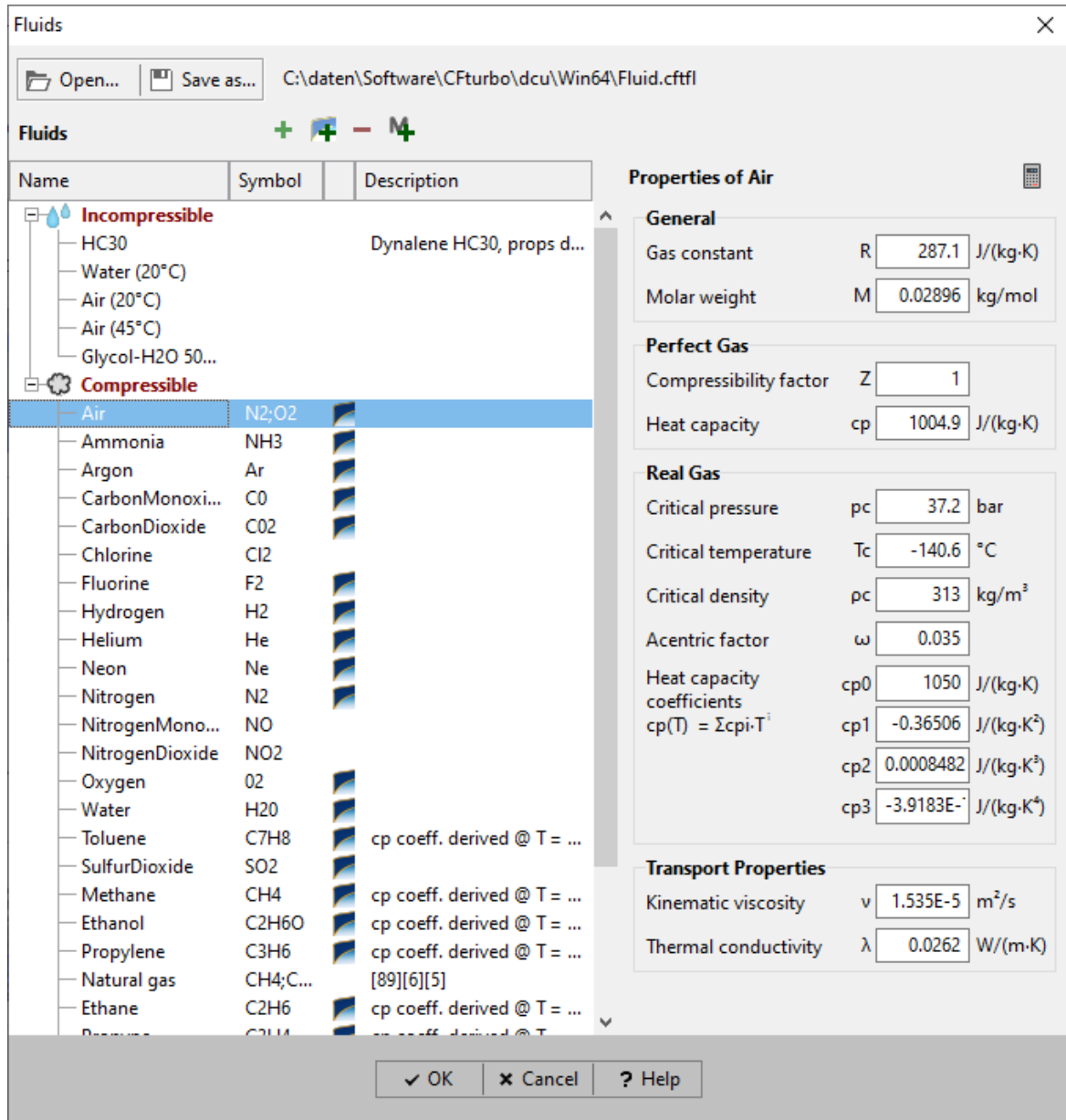
- import points from file (one point per line)
- export points to file
- copy all points to clipboard
- paste points from clipboard (e.g from Excel)
- clear the table

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](#)^[206] library. To this end the button  has to be used.

In the third column it is specified by  whether a [CoolProp](#) ^[206] definition of the fluid is available. This applies only for compressible fluids.

Incompressible fluid [for pumps, ventilators only]

Parameters are:

- density
- kinematic viscosity
- thermal conductivity
- 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

$$R = \frac{8.31446 \text{ J}/(\text{mol} \cdot \text{K})}{M}$$

- critical pressure p_{crit} , temperature T_{crit} and density ρ_{crit}
- acentric factor
- heat capacity coefficients c_{pi} (at zero pressure)
- compressibility factor Z
- kinematic viscosity
- thermal conductivity

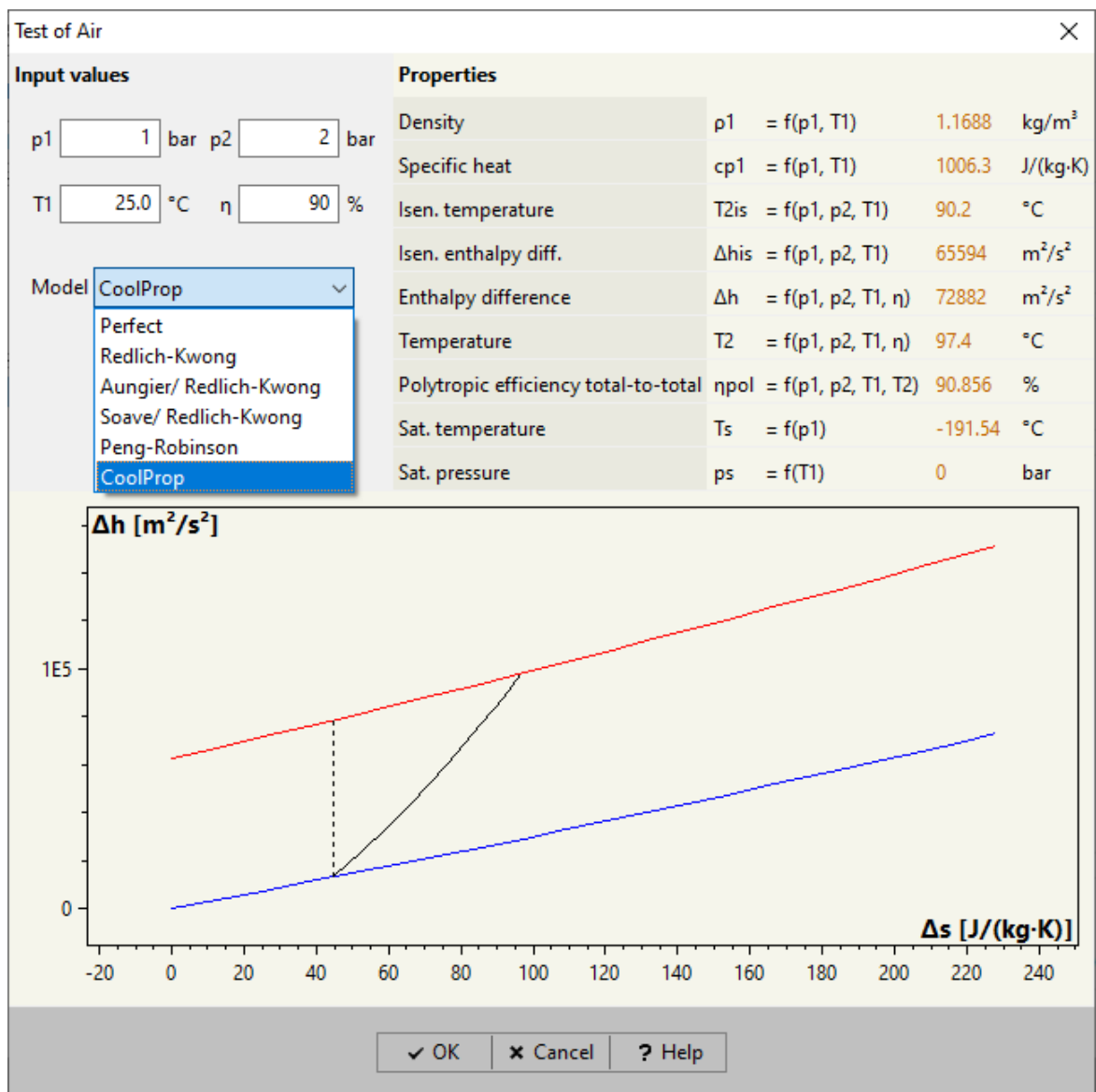
Currently the following gas models are implemented. They represent a relation between pressure, temperature and density (here given with its reciprocal the spec. volume v):

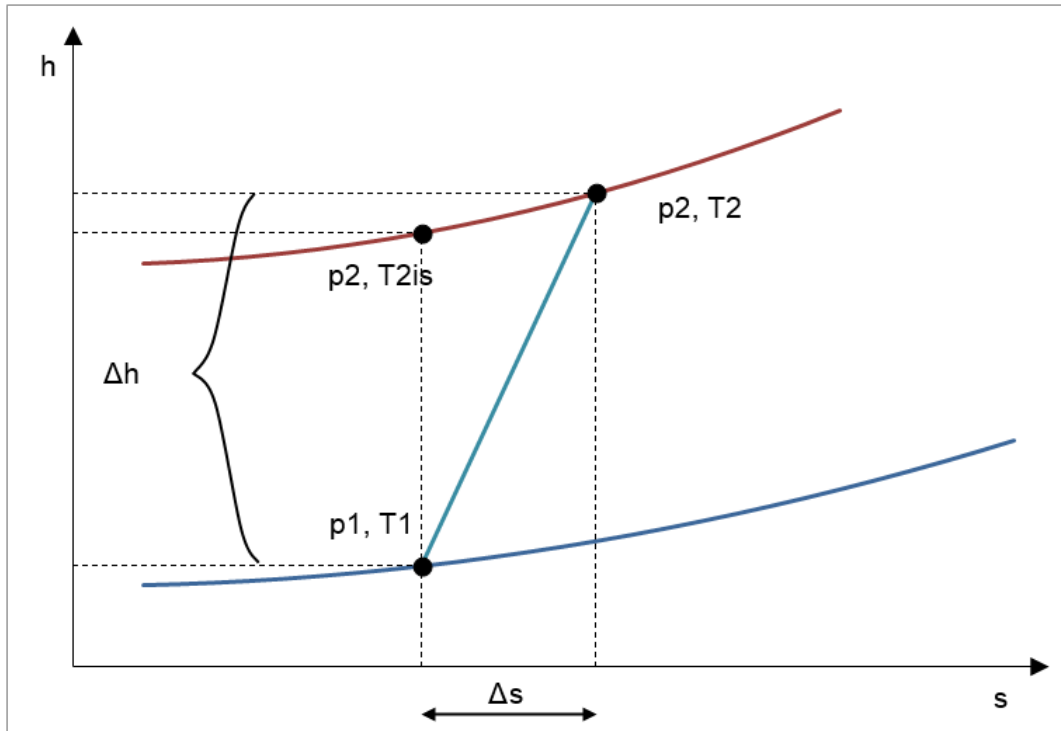
Gas model	Approach	Annotation	Reference (first published)
Perfect Gas	$p = \frac{R \cdot T \cdot Z}{v}$		
Redlich-Kwong	$p = \frac{R \cdot T}{v - b + c} - \frac{a(T)}{v(v + b)}$	Each approach has its own set of coefficients a, b and c.	Redlich, O., Kwong, J.N.S. ⁵⁶⁹
Aungier/Redlich-Kwong			Aungier, R.H. ⁵⁶⁹
Soave/Redlich-Kwong			Soave, G. ⁵⁶⁹
Peng-Robinson			Peng, D.Y., Robinson, D.B. ⁵⁶⁹
CoolProp	see reference	Currently available for fluid creation only but not within the project	www.CoolProp.org

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 ρ_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 η . The according enthalpy and entropy differences h and s resp. is given too, see h-s-diagram.





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, ventilators only]

The constant data needed for the design process will be derived at a temperature T_1 . Three different sets of fluids are available: pure fluids, water mixtures and 2 phase fluids. From the latter the liquid state shall be used.

Add CoolProp liquid

Name	Description
Pure liquids	
AS10	Aspen Temper -10, Potassiu...
AS20	Aspen Temper -20, Potassiu...
AS30	Aspen Temper -30, Potassiu...
AS40	Aspen Temper -40, Potassiu...
AS55	Aspen Temper -55, Potassiu...
DEB	Diethylbenzene mixture - D...
DowJ	DowthermJ
DowJ2	Dowtherm J, Diethylbenzen...
DowQ	DowthermQ
DowQ2	Dowtherm Q, Diphenyletha...
HC10	Dynalene HC10
HC20	Dynalene HC20
HC30	Dynalene HC30
HC40	Dynalene HC40
HC50	Dynalene HC50
HCB	Hydrocarbon blend - Dynal...
HCM	Hydrocarbon mixture - Gilo...
HFE	Hydrofluoroether - HFE-710...
HFE2	HFE-7100, Hydrofluoroether
HY20	HYCOOL 20, Potassium for...
HY30	HyCool 30, Potassium form...
HY40	HyCool 40, Potassium form...
HY45	HyCool 45, Potassium form...
HY50	HyCool 50, Potassium form...
NBS	NBS, Water
NaK	Nitrate salt, 0.6 NaNO3 and ...
PBB	Pirobloc HTF-BASIC
PCL	Paracryol, Aliphatic Hydroc...
PCR	Paratherm CR
PGLT	Paratherm GLT
PHE	Paratherm HE

Properties of DowJ

T1 °C T2 °C p1 bar

Mass fraction H2O x %

Density	ρ	= f(T1)	858.52	kg/m ³
Specific heat	c_p	= f(T1)	1853.6	J/(kg·K)
Kin. viscosity	ν	= f(T1)	9.9469E-7	m ² /s
Therm. conductivity	λ	= f(T1)	0.12731	W/(m·K)
Vapor pressure	p_v	= f(T1)	0	bar

✓ OK ✗ Cancel ? Help

Compressible fluid [for compressors, turbines only]

The heat capacity coefficients c_{pi} (at zero pressure) are derived at a temperatures T_1 and T_2 . A least square fitting algorithm is used for this purpose.

Add CoolProp gas

Name	Symbol
CoolProp gases	
1-Butene	C4H8
Acetone	C3H6O
Air	N2;O2;H2O;CO2
Ammonia	NH3
Argon	Ar
Benzene	C6H6
CarbonDioxide	CO2
CarbonMonoxide	CO
CarbonylSulfide	COS
cis-2-Butene	C4H8
CycloHexane	C6H12
Cyclopentane	C5H10
CycloPropane	C3H6
D4	D4
D5	D5
D6	D6
Deuterium	D2
Dichloroethane	C2H4Cl2
DiethylEther	C4H10O
DimethylCarbonate	C3H6O3
DimethylEther	C2H6O
Ethane	C2H6
Ethanol	C2H6O
EthylBenzene	C8H10
Ethylene	C2H4
EthyleneOxide	C2H4O
Fluorine	F
HeavyWater	D2O
Helium	He
HFE143m	C2H3F3O
Hydrogen	H2
HydrogenChloride	ClH

Properties of Ammonia @(p1, T1, T2)

p1 bar T1 °C T2 °C

Gas constant	R	488.22	J/(kg·K)
Specific heat	cp = f(T1)	2092.8	J/(kg·K)
Critical pressure	pcrit	113.33	bar
Critical temperature	Tcrit	132.25	°C
Critical density	pcrit	225	kg/m³
Acentric factor	ω	0.25601	
Kin. viscosity	ν = f(p1, T1)	1.4539E-5	m²/s
Therm. conductivity	λ = f(p1, T1)	0.024933	W/(m·K)
cp coefficient 0	cpc0 = f(T1..T2)	2.19E3	J/(kg·K)
cp coefficient 1	cpc1 = f(T1..T2)	-2.7	J/(kg·K²)
cp coefficient 2	cpc2 = f(T1..T2)	0.00997	J/(kg·K³)
cp coefficient 3	cpc3 = f(T1..T2)	-6.94E-6	J/(kg·K⁴)
Sat. temperature	Ts = f(p1)	-33.588	°C
Sat. pressure	ps = f(T1)	10.032	bar

Graph: cp [J/(kg·K)] vs T [°C]

OK Cancel Help

5.3.4.2 Gas mixtures

Compressible fluid [for compressors, turbines only]

New gas mixtures can be designed on the basis of gases already defined in the compressible branch of the [fluid manager](#)^[201].

All mixture parameters apart from kinematic viscosity and thermal conductivity are calculated with the help of the component's mass ratio w_i and parameter f_i :

$$f_{\text{mix}} = \sum_i w_i f_i.$$

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 $\varphi_{ij}(T)$. Mole fraction and mass ratio are connected by the molar weight M :

$$\sum_i x_i M_i = \frac{1}{\sum_i \frac{w_i}{M_i}}.$$

The correction factor $\varphi_{ij}(T)$ is determined from the component's kinematic viscosity ν_i and molar weight M_i according to [Mason & Saxena](#)^[569] by:

$$\phi_{ij}(T) = \frac{1}{2\sqrt{2}} \left(1 + \frac{M_i}{M_j} \right)^{-1/2} \left[1 + \left(\frac{v_i(T)}{v_j(T)} \right)^{1/2} \left(\frac{M_j}{M_i} \right)^{1/4} \right]^2.$$

The kinematic viscosity is determined by:

$$v(T) = \sum_i \frac{x_i \cdot v_i(T)}{\sum_j x_j \cdot \phi_{ij}(T)}.$$

The thermal conductivity is determined by:

$$\lambda(T) = \sum_i \frac{x_i \cdot \lambda_i(T)}{\sum_j x_j \cdot \phi_{ij}(T)}.$$

5.3.5 Profiles

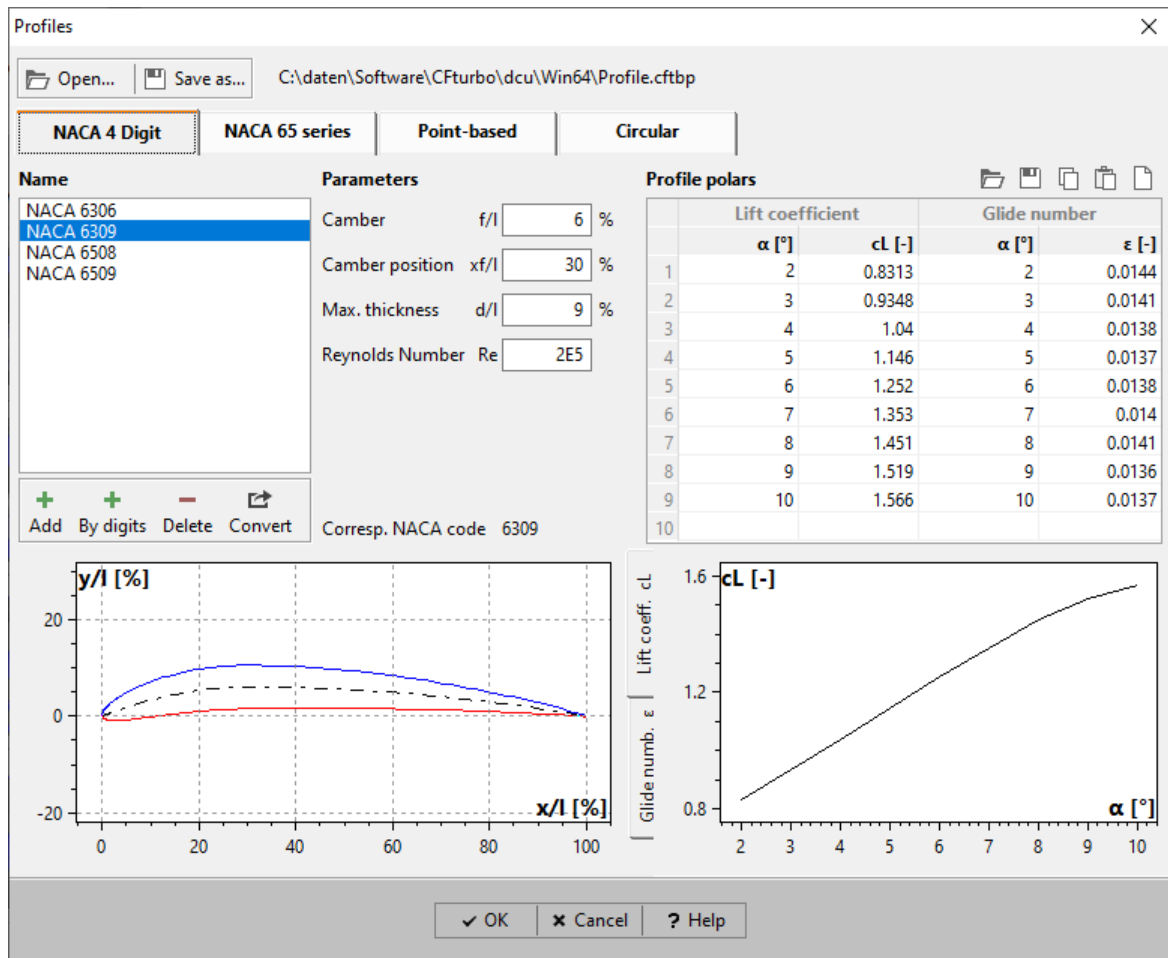
? SETTINGS | Database | Profiles

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](#)^[475].

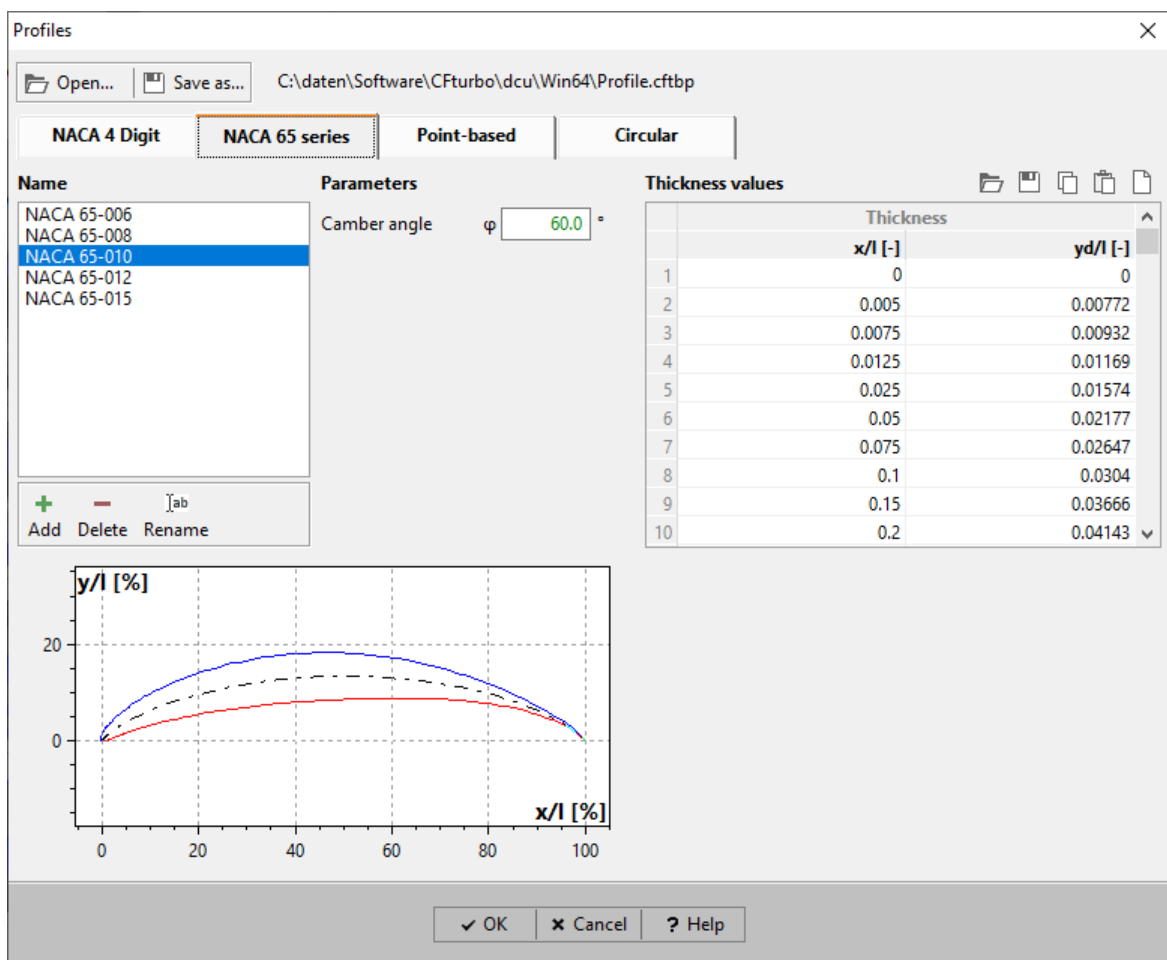
The meanline consists of two parabola arcs, whose transition point is their apex, respectively. The point is defined by the the first two digits.

$$y_s = \frac{f}{l} \frac{1}{\left(\frac{x_f}{l}\right)^2} \left[2 \frac{x_f}{l} \frac{x}{l} - \left(\frac{x}{l}\right)^2 \right] \quad \text{if } \frac{x}{l} \leq \frac{x_f}{l}$$

$$y_s = \frac{f}{l} \frac{1}{\left(\frac{x_f}{l}\right)^2} \left[1 - 2 \frac{x_f}{l} + 2 \frac{x_f}{l} \frac{x}{l} - \left(\frac{x}{l}\right)^2 \right] \quad \text{if } \frac{x}{l} > \frac{x_f}{l}$$

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.

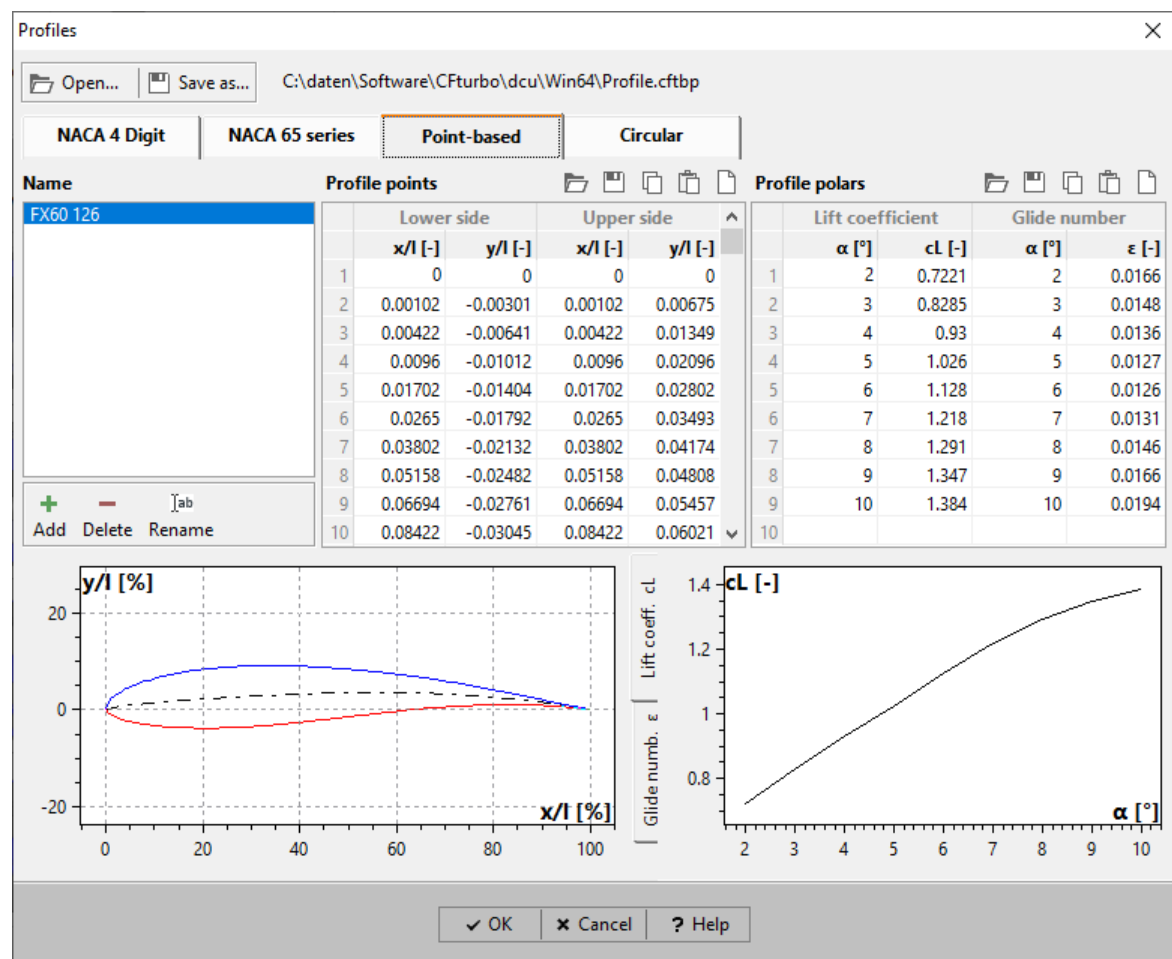
The meanline can be calculated from a theoretical lift coefficient that is calculated from a user-defined camber angle, see [Carolus](#)^[566] p. 54, (Eq. 3.11, 3.12):

$$c_{fl} = \frac{2\pi}{\ln(2)} \cdot \tan\left(\frac{\varphi}{4}\right) \text{ mit } \varphi = \beta_{B2} - \beta_{B1}$$

$$\frac{y_s}{l} = -\frac{c_{fl}}{4\pi} \left[\left(1 - \frac{x}{l}\right) \cdot \ln\left(1 - \frac{x}{l}\right) + \frac{x}{l} \cdot \ln\left(\frac{x}{l}\right) \right]$$

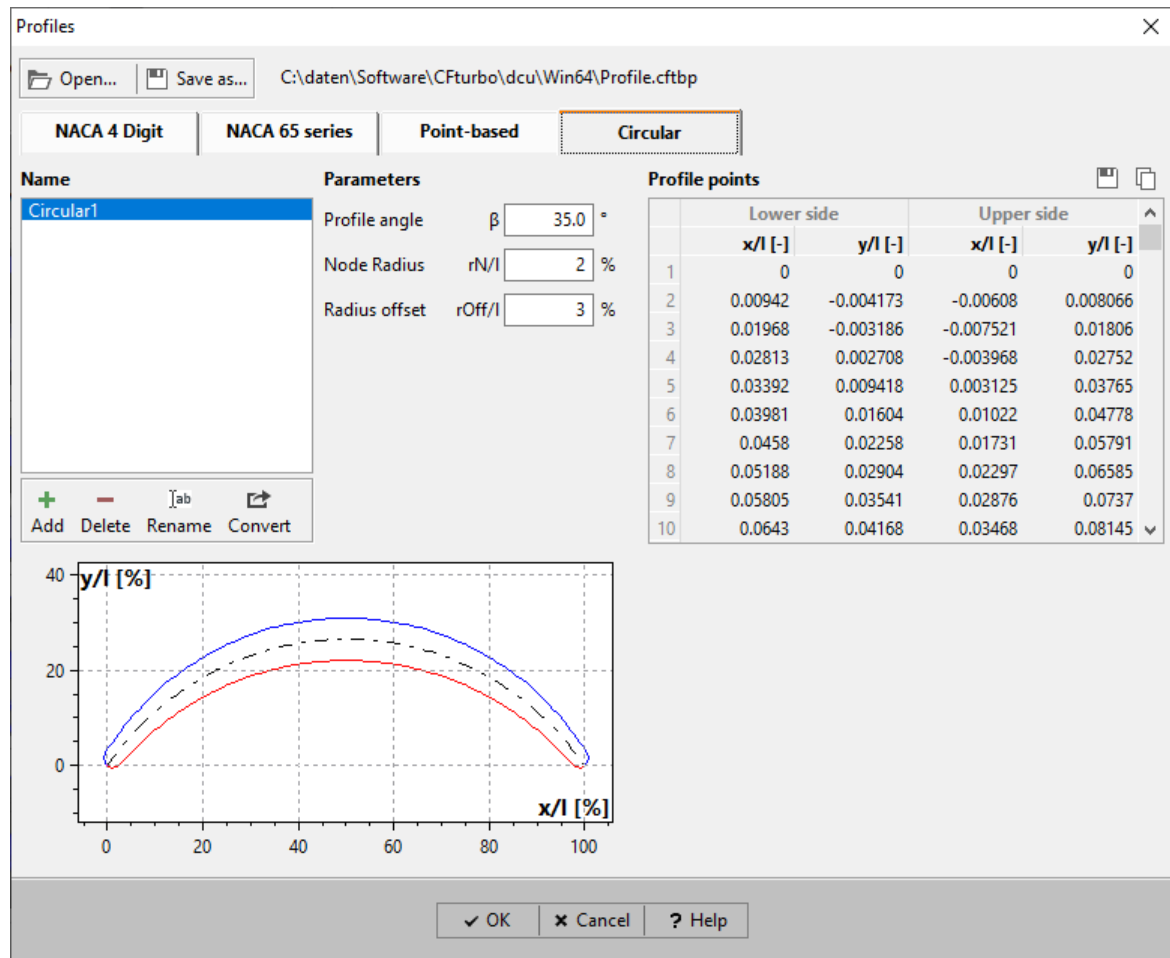
The nose radius can be modified, thickness values have to be changed in the table. One can type in a camber angle that will result in a cambered profile. This profile can be promoted to Point-based profiles by pressing convert. Please note that the skeleton design is done in the [appropriate design step](#)^[457]. Therefore the camber angle is not part of the profile description and is given here for informational and conversion reasons.

Point-based

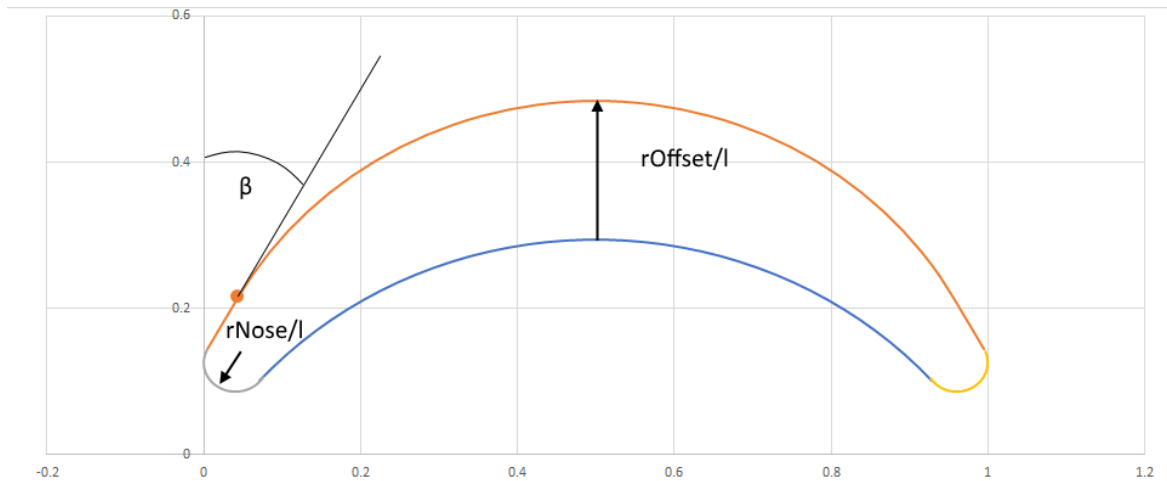


Besides NACA profiles also user-defined profiles are provided. Therefore the lower and upper side of the profile has to be known. Moreover lift coefficients and glide numbers need to be set with respect to the angle of attack.

Circular

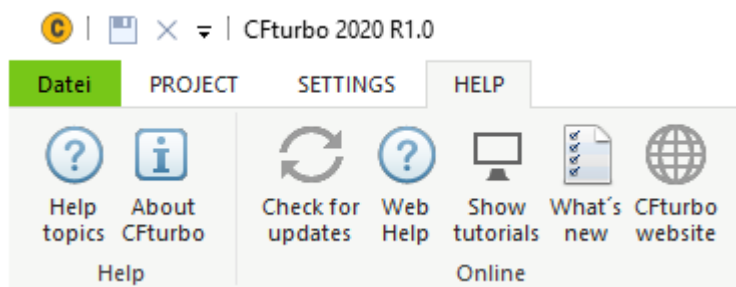


Circular profiles are constructed by 2 circular arcs, 1 linear piece and a nose with a certain nose radius. They are described by 3 parameters: profile angle β , nose radius rN/l and a radius offset. The construction details are displayed in the sketch below:








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**

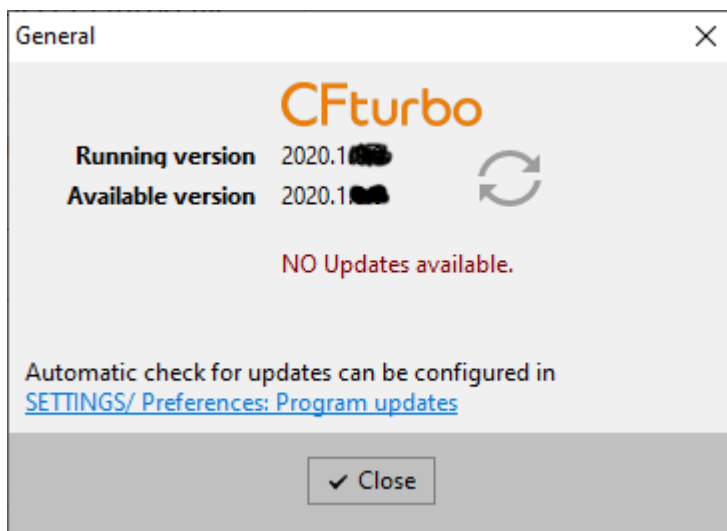
Open CFturbo website in browser


5.4.1 Check for Updates

? [Help](#) | [Online](#) | [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).

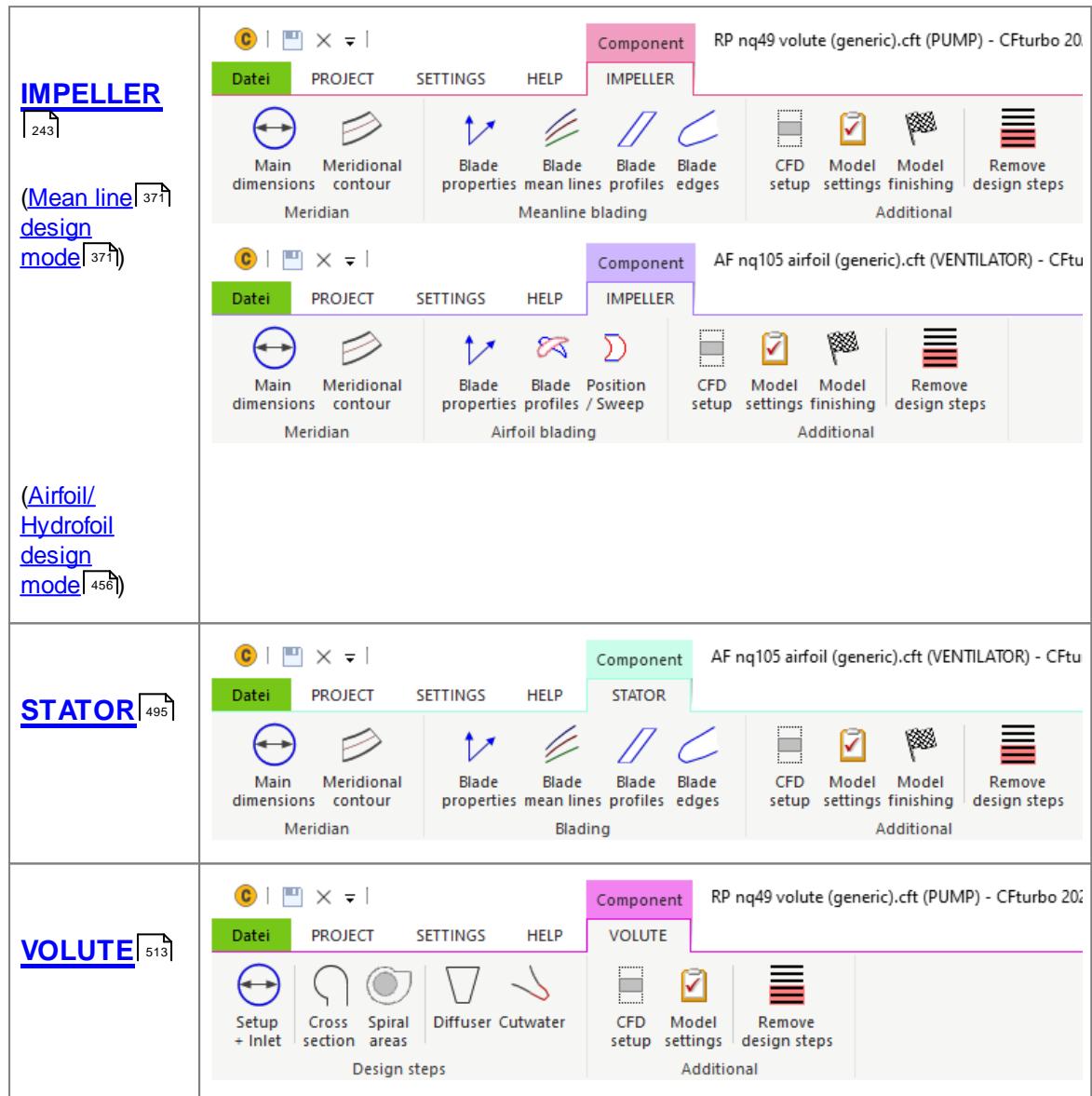


Update check can be executed automatically. This can be configured in the [Preferences](#) 

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:



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](#)^[65]).

For designing the complete geometry of a single component you have to run through all items of the appropriate menu step by step.

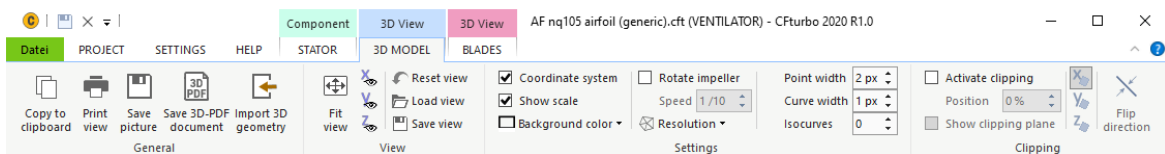
Alternatively all these menu items can be selected in the Meridian view using the toolbar directly on the selected component (see [Meridian](#) ^[222]).

5.6 3D View

This context sensitive menu is used for handling of the 3D model. It becomes visible if the [3D Model](#) ^[225] view is currently selected. Detailed description can be found in [Views/ 3D Model](#) ^[225].

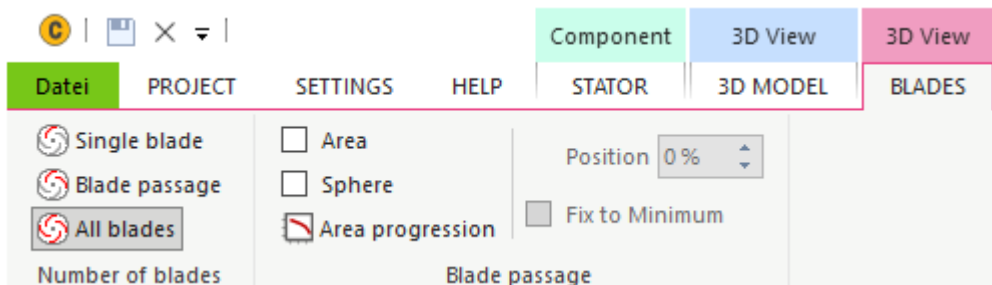
2 tabs are available:

3D MODEL



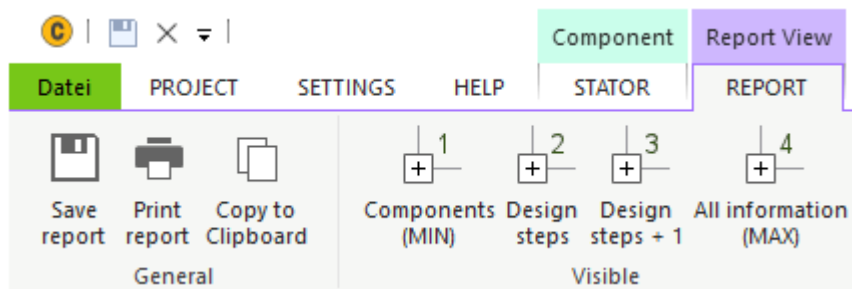
BLADES

This Menu is used for handling geometries with blades (impeller, vaned stator) in the 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.



Detailed description can be found in [Views/ Report](#)^[240].

Part

VI

6 Views

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](#) [74].

- [Meridian](#) [222]

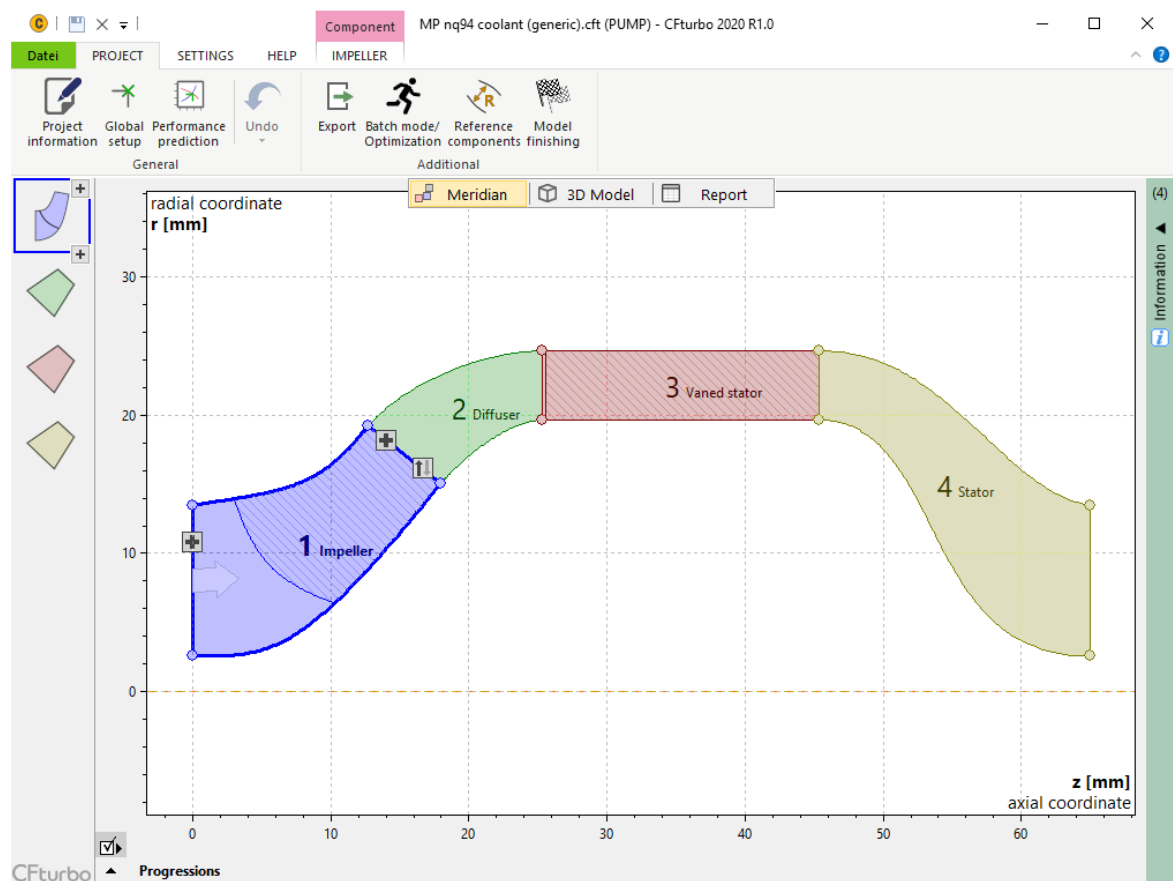
The diagram with the meridional view of the components gives an overview of the project and enables quick access to the components and the [Interfaces](#) [39] in between.

- [3D Model](#) [225]

Shows the whole project as a 3D model.

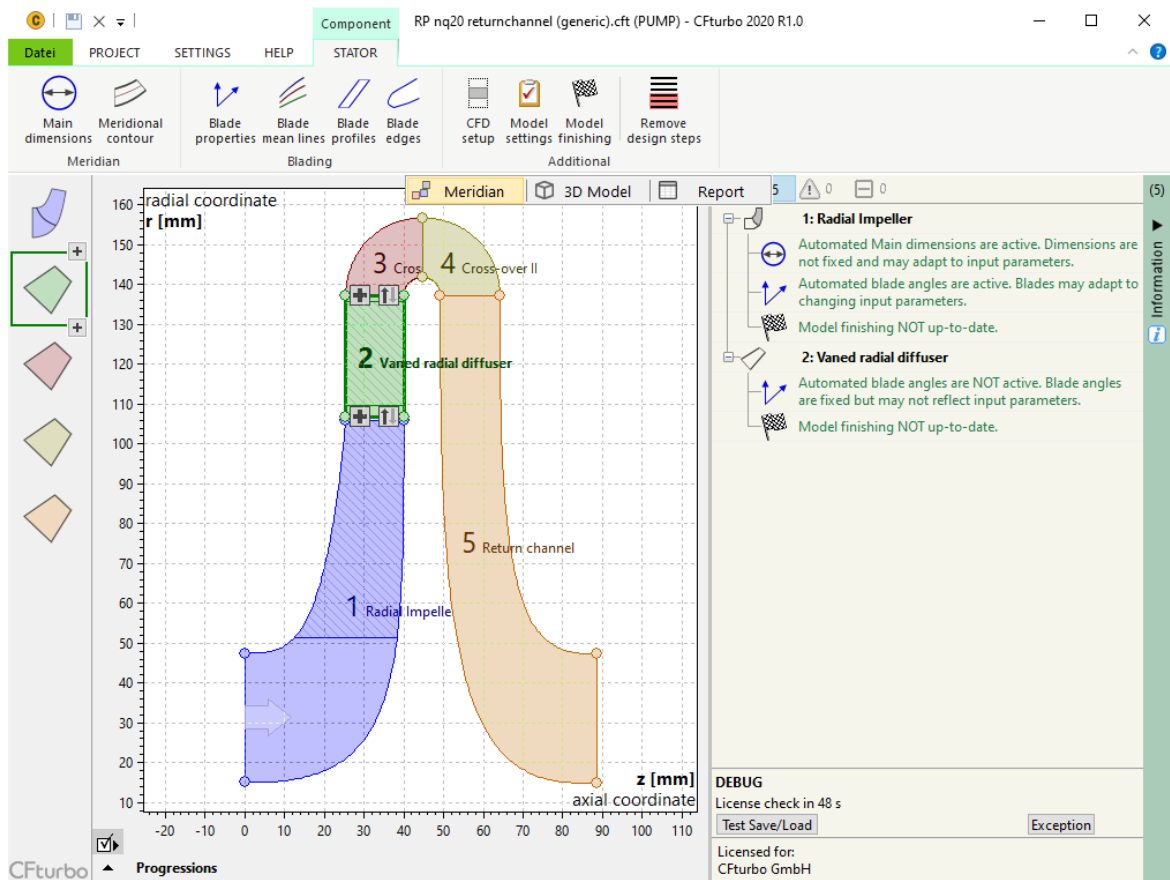
- [Report](#) [240]

Presents a tabular view on the project information and the parameters of the components down to design step level.



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.

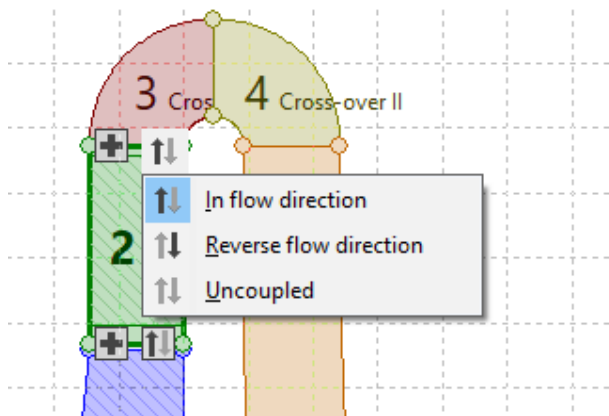


Alternatively you can use the corresponding ribbon menu (see [IMPELLER/ STATOR/ VOLUTE](#)²¹⁷).

Right clicking on the component opens its context menu, see [Activate/ Rename/ Delete components](#)^[63].



See [Add component](#)^[41].

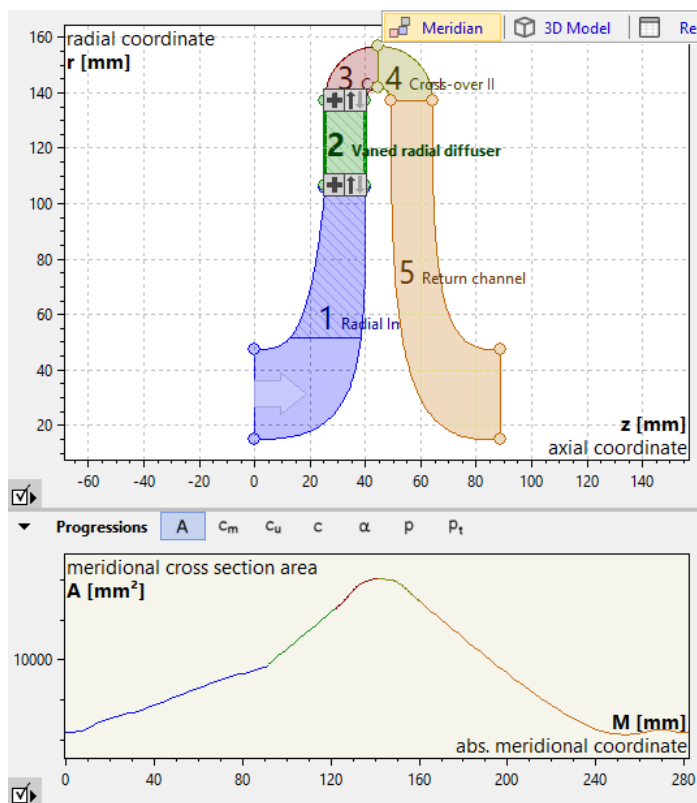


Geometric coupling

The direction of the [geometric coupling](#)^[39] 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



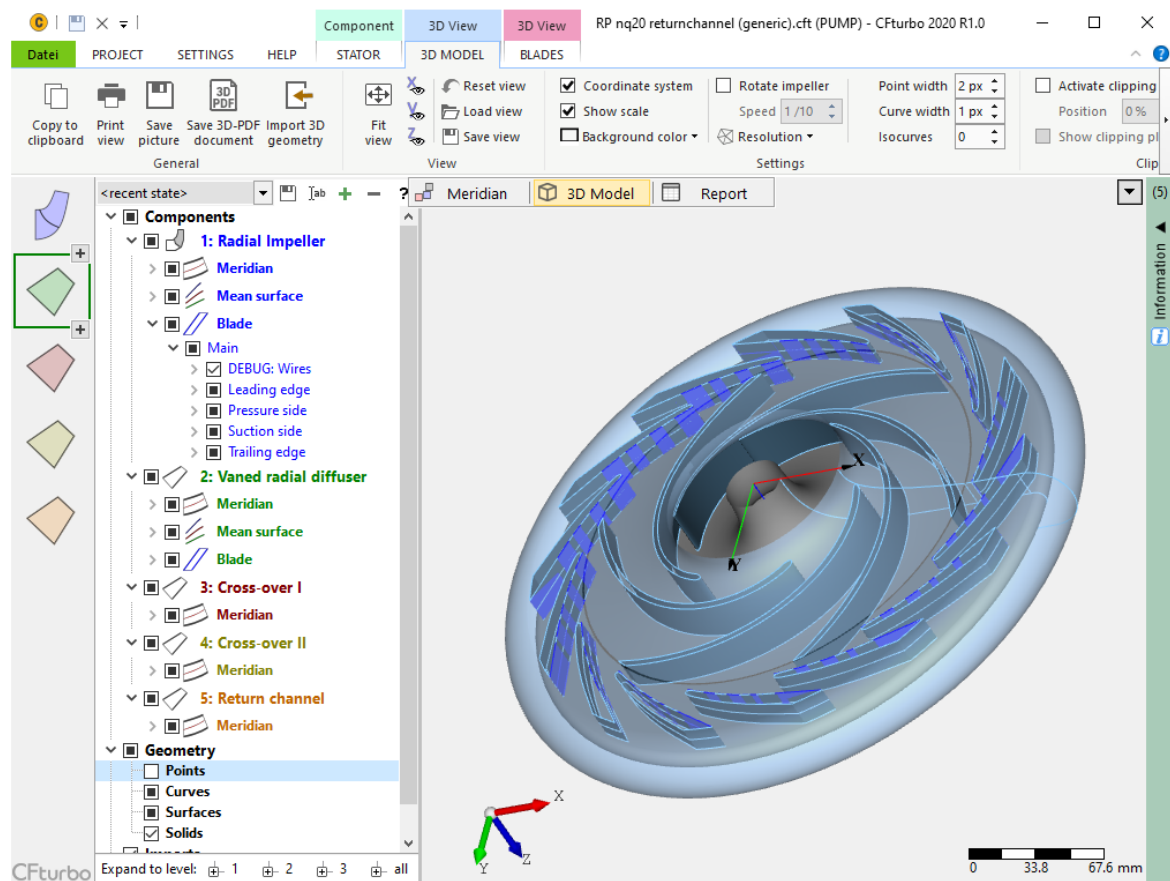
Below the meridional view, **progressions** of several physical quantities along the flow direction of all components can be displayed:

A	Cross section area
c_m	Meridional velocity
c_u	Circumferential velocity
c	Absolute velocity
	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](#)^[218].

The CAD model can be exported as IGES, STEP, STL, Parasolid or BREP - see [Export](#)^[103]. For export, only the currently visible geometrical elements are considered.



Navigation

The 3D display can be influenced by **mouse**:

 Rotate	Rotation around center of visualized geometry or clicked point on a 3D-object respectively  The rotation center is visualized by a 3D marker.
 Zoom	↑ Zoom (also mouse wheel) ↔ Rotation around z-axis
 Move	Move

The functions can be assigned to mouse buttons via [Preferences/ General](#) ¹⁸⁶.

Furthermore, the 3D-View is sensitive to mouse movement and mouse clicks in the following manner:

Mouse movement	Activates highlighting of 3D-object under the cursor. When hovering on the 3D-object, a hint with its name is displayed.
Left mouse-button	Selects and deselects the 3D-object under the cursor, respectively. When clicking in empty space, all 3D-objects are deselected.
<Ctrl> + left mouse-button	Multi-selection of 3D-objects.
Right mouse-button	Opens context menu with display properties for all selected 3D-objects (see Model tree (left) ²³⁴).

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 \(top\)](#) ²²⁷

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.

→ [Model tree \(left\)](#) ²³⁴

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](#) ²³⁸

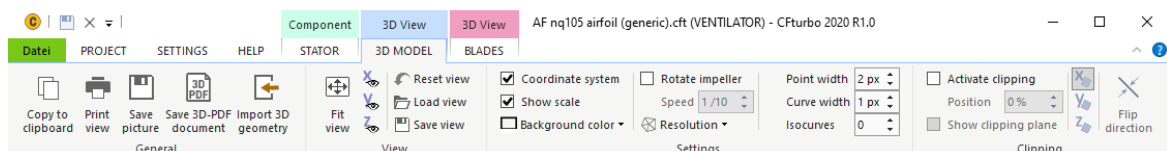
→ [Open/ Save design](#) ⁷⁸

→ [Data export](#) ¹⁰³



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

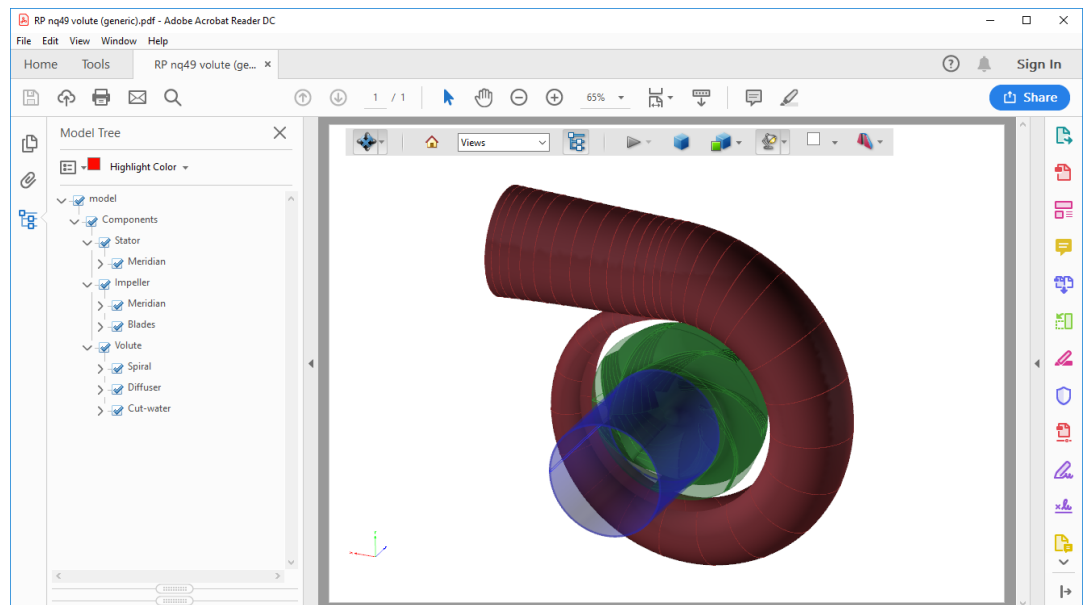
Visible 3D-model data is transferred to a PDF-document, enabling the user to distribute or archive a 3D-representation of CFturbo-models.

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](#)^[225] and can be [transformed](#)^[237] and exported.

If the import consumes a lot of time, a lower resolution can be selected (see "Settings" below).

View

	Fit view (zoom all geometry to visible region)
	Viewing direction in positive or negative (<↑>) x-axis direction
	Viewing direction in positive or negative (<↑>) y-axis direction
	Viewing direction in positive or negative (<↑>) z-axis direction
	Reset view (default position)
	Load view from file
	Save current view to file

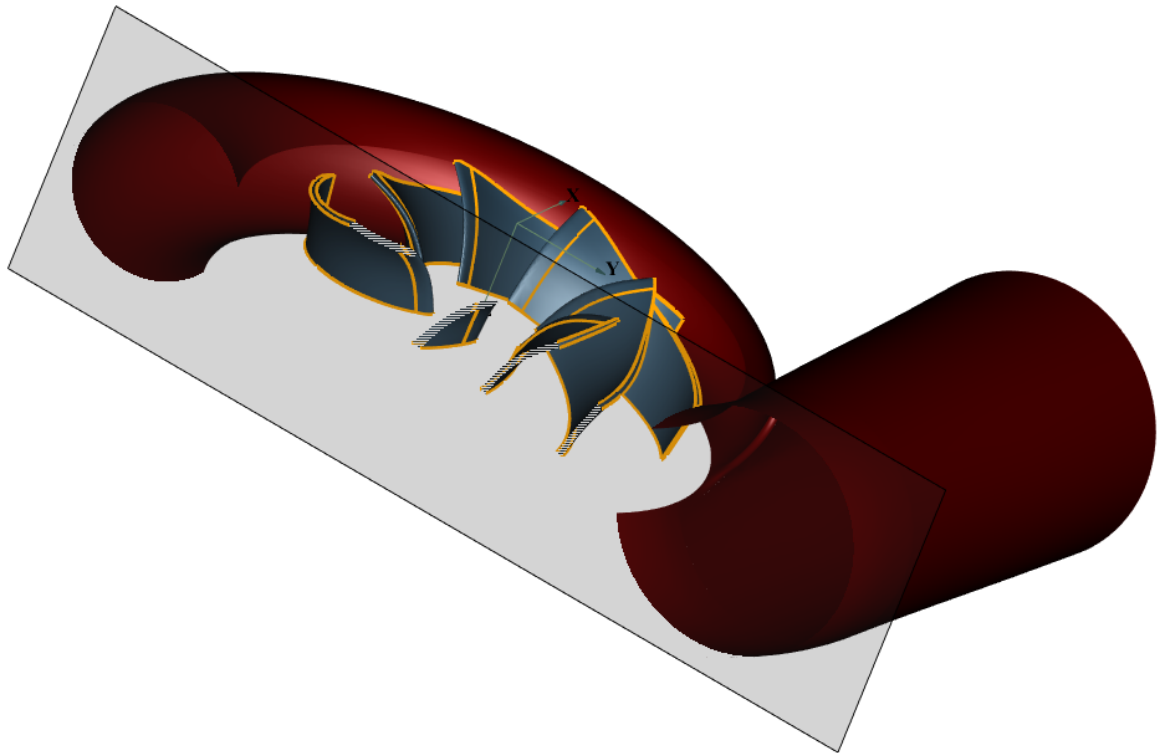
Settings

	Switch coordinate system on/off
	Switch scale system on/off
	Set background color
	A uniform rotation of the impeller around the z axis can be generated, whereby the velocity can be influenced by the track bar.
	Select resolution of curves and surfaces (affects display)
	<input type="checkbox"/> Coarse
	<input type="checkbox"/> Middle
	<input type="checkbox"/> Fine
	Define line width for points
	Define line width for curves
	Set number of surface isocurves

Clipping

A clipping plane for $x=\text{const.}$, $y=\text{const.}$ or $z=\text{const.}$ can be defined and optionally displayed. The position of the clipping plane can be adjusted.

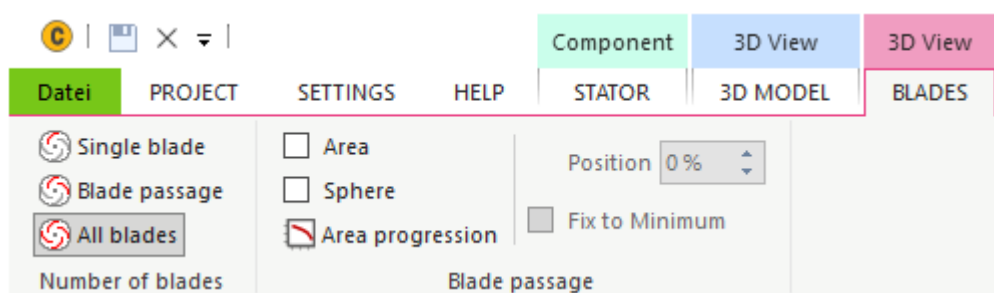
The direction of clipping (visible clipping side) can be switched.




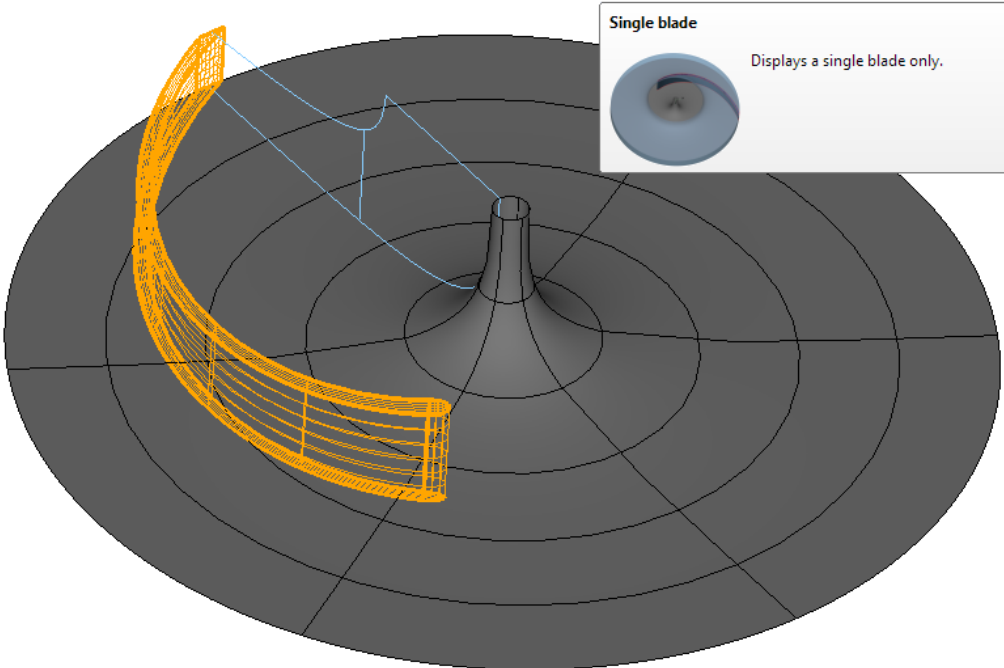

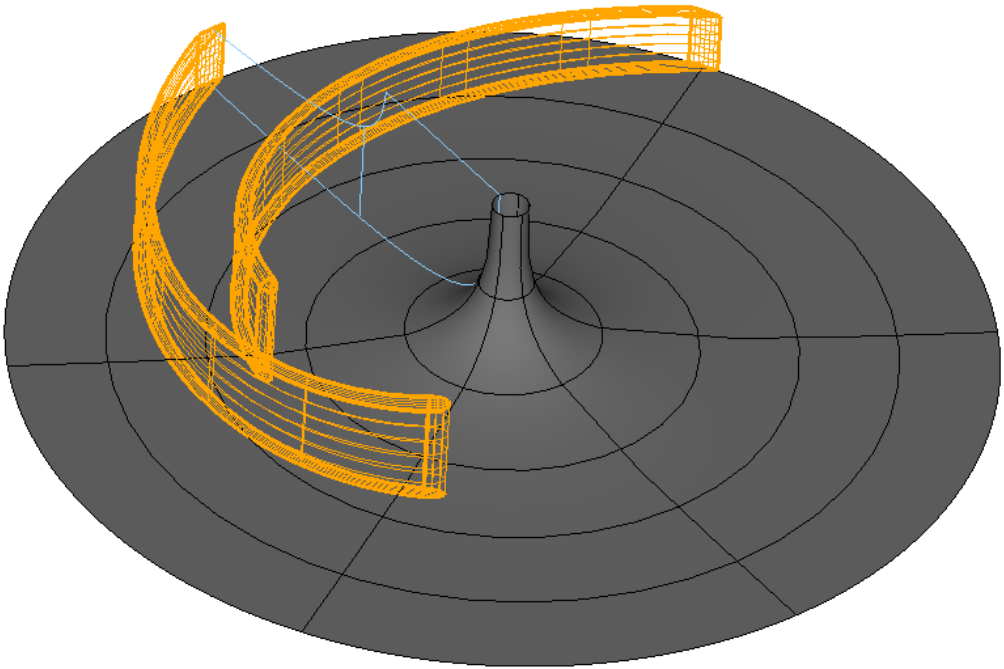

BLADES

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.

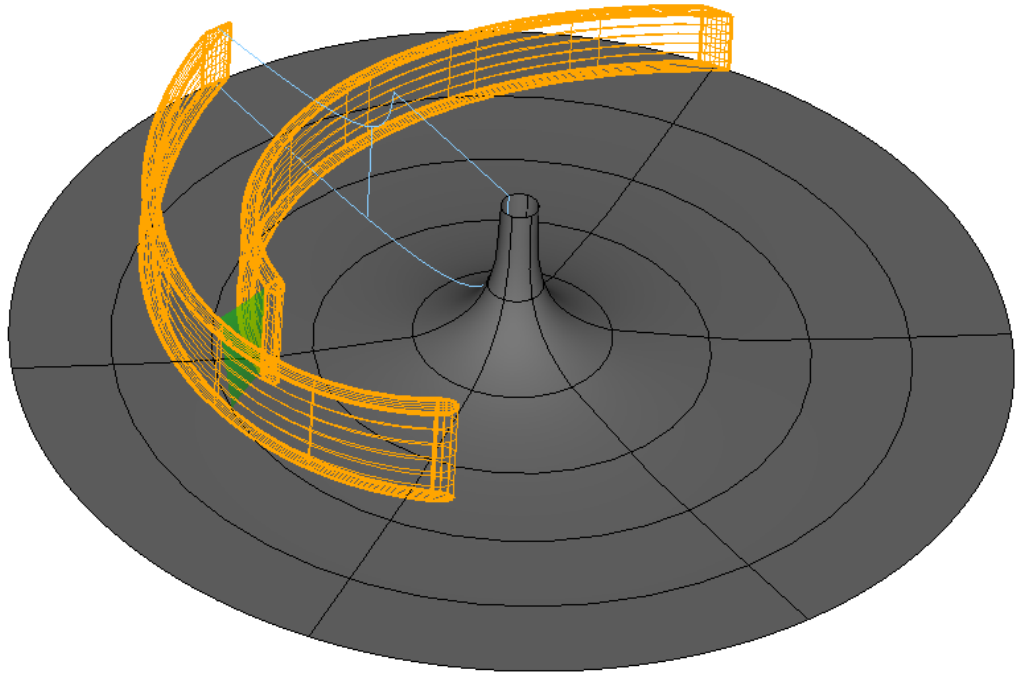


	Single blade
---	--------------

	 <p>Single blade Displays a single blade only.</p>
	<p>Blade passage</p> <p>Display a single blade passage bordered by 2 neighboring blades.</p> 
	<p>All blades</p>

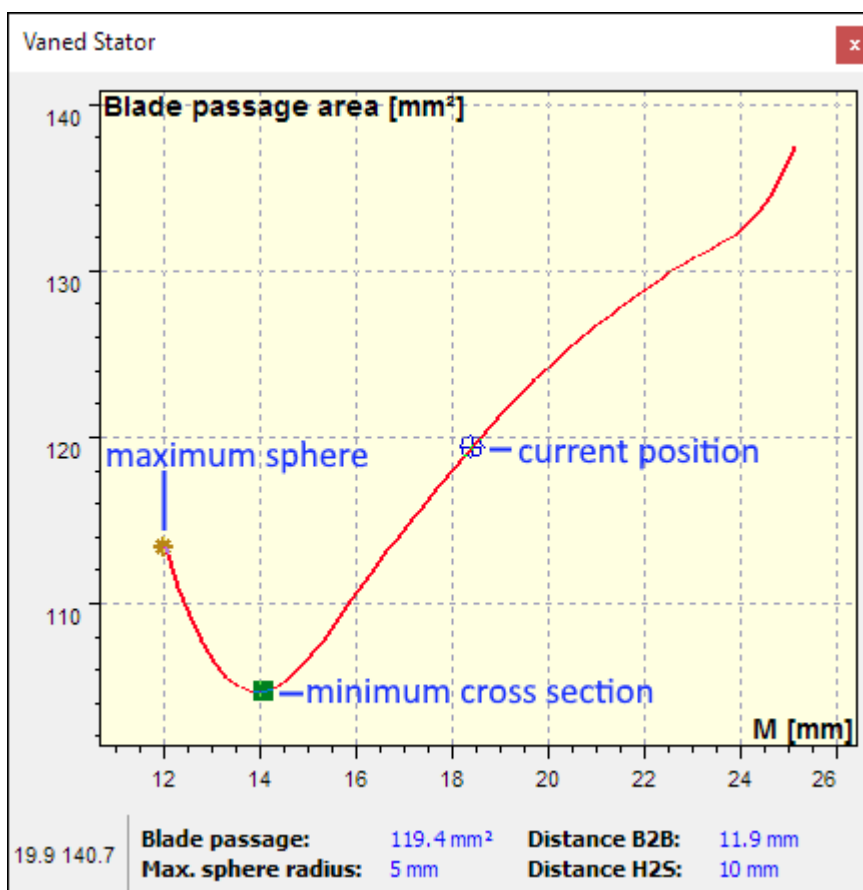
Display all blades of the selected impeller or vaned stator.

Area



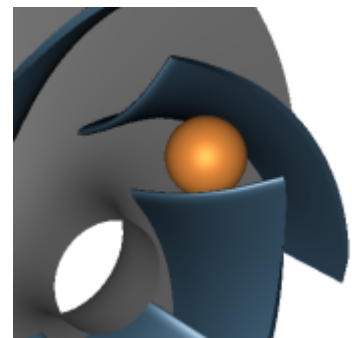
Display an approximately perpendicularly flown through area between hub, shroud and two neighboring blades for the currently selected component. The position of this area can optionally be fixed to the location of the throat area (**Fix to minimum**). Otherwise, it can be slid to any reasonable position 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](#)¹⁹⁶ 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.

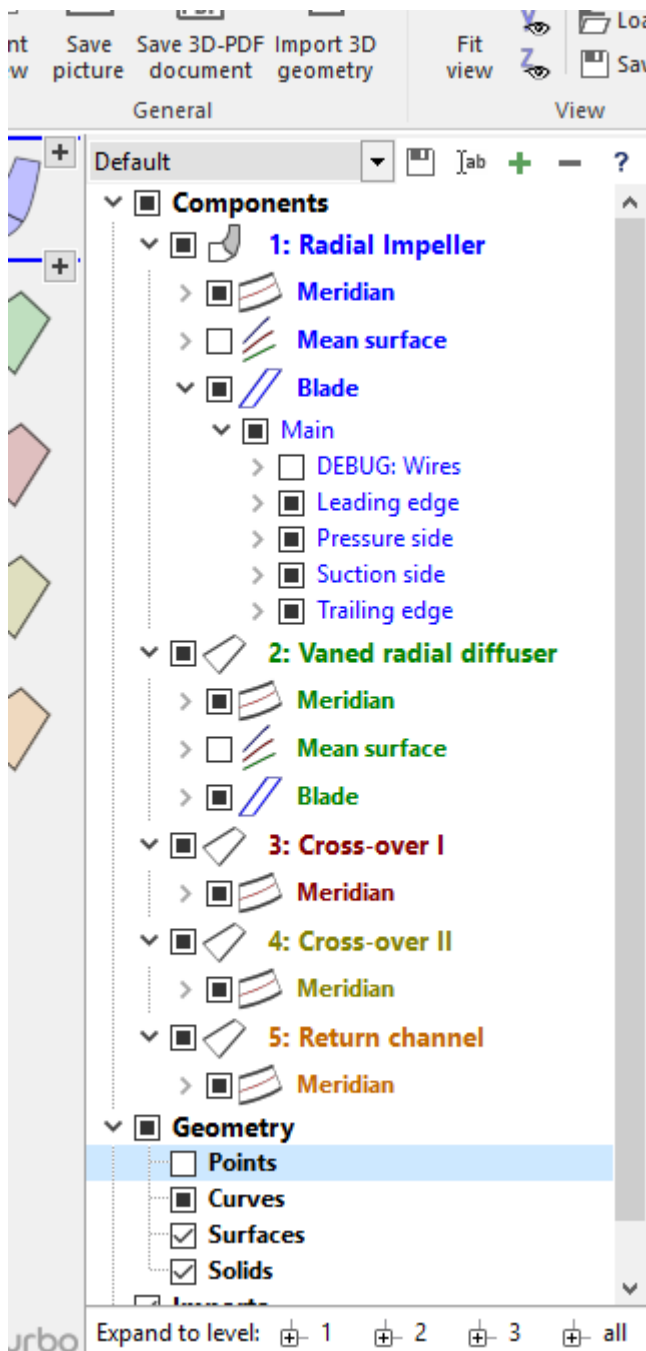


Sphere

The sphere represents a particle with the highest possible diameter that can be conveyed through the blade passage.



6.2.2 Model tree (left)

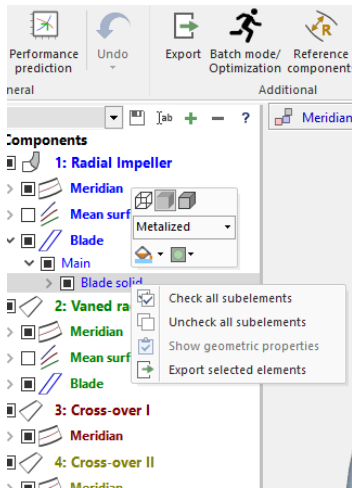


The **model tree** contains all available geometry parts listed in a tree structure, whereby their visibility can be switched on or off alternatively. All visible elements are exported, if the model is saved as IGES, STEP, STL, Parasolid or BREP - see [Export](#)^[103].

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.

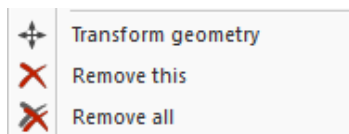
The attributes can be defined by right click:



- 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:



- applies user defined geometric transformations (see below)
- removes selected import from model tree and 3D view
- removes all imported elements from model tree and 3D view

Model tree structure

The model tree has 3 main sections:


1) Section **Components**



contains all components of the project with the following sub elements:

Impeller/Stator	Volute
Meridian	Spiral
Mean surface	Diffuser
Blade	Cut-water




[CFD setup](#)  478

[CFD setup](#)  562

If an element contains child elements, it can be expanded by clicking on the collapsed element symbol ().

Each single element without child elements can be selected () or unselected ().

Each single element with child elements can have 3 states:

 <input checked="" type="checkbox"/> Hub <input checked="" type="checkbox"/> Points <input checked="" type="checkbox"/> Curve <input checked="" type="checkbox"/> Surface	<input checked="" type="checkbox"/> The element and all child elements are selected.
 <input checked="" type="checkbox"/> Hub <input type="checkbox"/> Points <input checked="" type="checkbox"/> Curve <input checked="" type="checkbox"/> Surface	<input checked="" type="checkbox"/> The element and not all child elements are selected.
 <input type="checkbox"/> Hub <input type="checkbox"/> Points <input checked="" type="checkbox"/> Curve <input checked="" type="checkbox"/> Surface	<input type="checkbox"/> 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
- Solids

This allows:

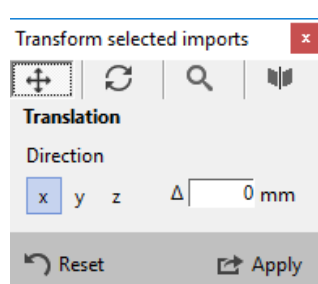
- to select *all* objects of a certain geometrical type. In the 3D view, only those elements become visible, whose parent elements are selected also.
- to modify the display properties of all *currently visible* objects of a certain geometrical type.

3) Section Imports

This section contains all imported geometric models including CFturbo components of [reference projects](#)^[187] or simply [imported 3D models](#)^[227].

Visibility and render properties for imported models can be modified in the same way as for components of section *Components*.

The option *Transform geometry* is intended to help align imported component models with the project model to make visual comparisons of the model shapes more convenient. To this end, any number of simple transformations can be applied via the dialog that opens when *Transform geometry* is selected.



The *Transform geometry* dialog allows the application of four different types of geometric transformations, accessible by clicking on the corresponding symbols (from left to right: translation, rotation, uniform scaling, mirroring).

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.



To apply a transformation to the selected parts, select a transformation type, set its parameters and click the *Apply* button (or hit Enter).



The model transformation can be *reset* to the state which it was imported with by clicking the reset button.

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 as, see above).

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.

Default ▼	Select existing model state
	Save model state
	Rename selected model state

	Add new model state
	Delete selected model state

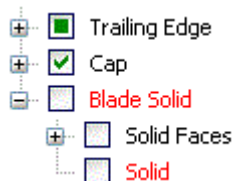
The following *predefined model states* cannot be modified:

- "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](#)²²²

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](#)⁶⁰.

Possible warnings

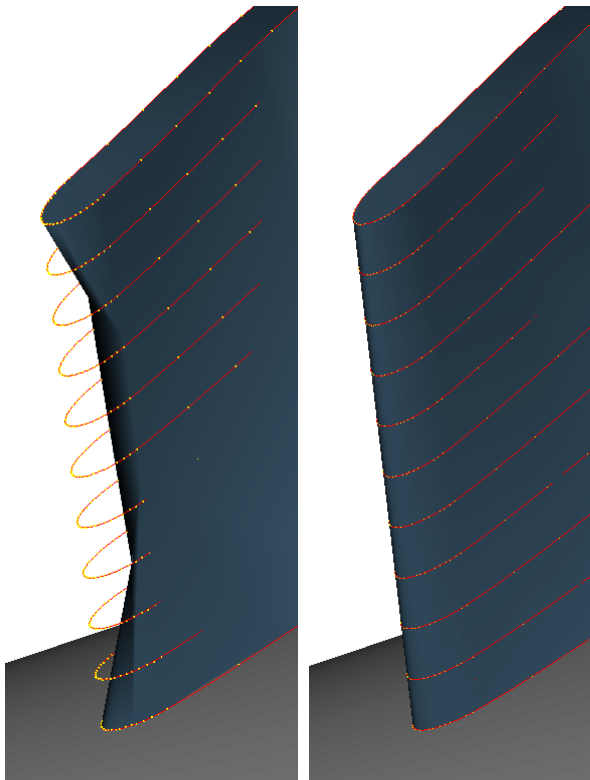
Problem	Possible solutions
3D-Error: Could not create solid.	

Problem	Possible solutions
Distance tolerance is too low or too high	Change the distance tolerance (see Model settings ⁴⁸⁶)
Number of data points is disadvantageous (seldom)	Change the number of data points for the 3D model (see Model settings ⁴⁸⁶)

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-](#) ⁴⁸⁶ or [Volute-Settings](#) ⁵⁶³)
- try a different display resolution (see [Model display \(top\)](#) ²²⁷)



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 \(top\)](#)^[227]).

Normally, choosing another resolution (see [Model display \(top\)](#)^[227]) 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

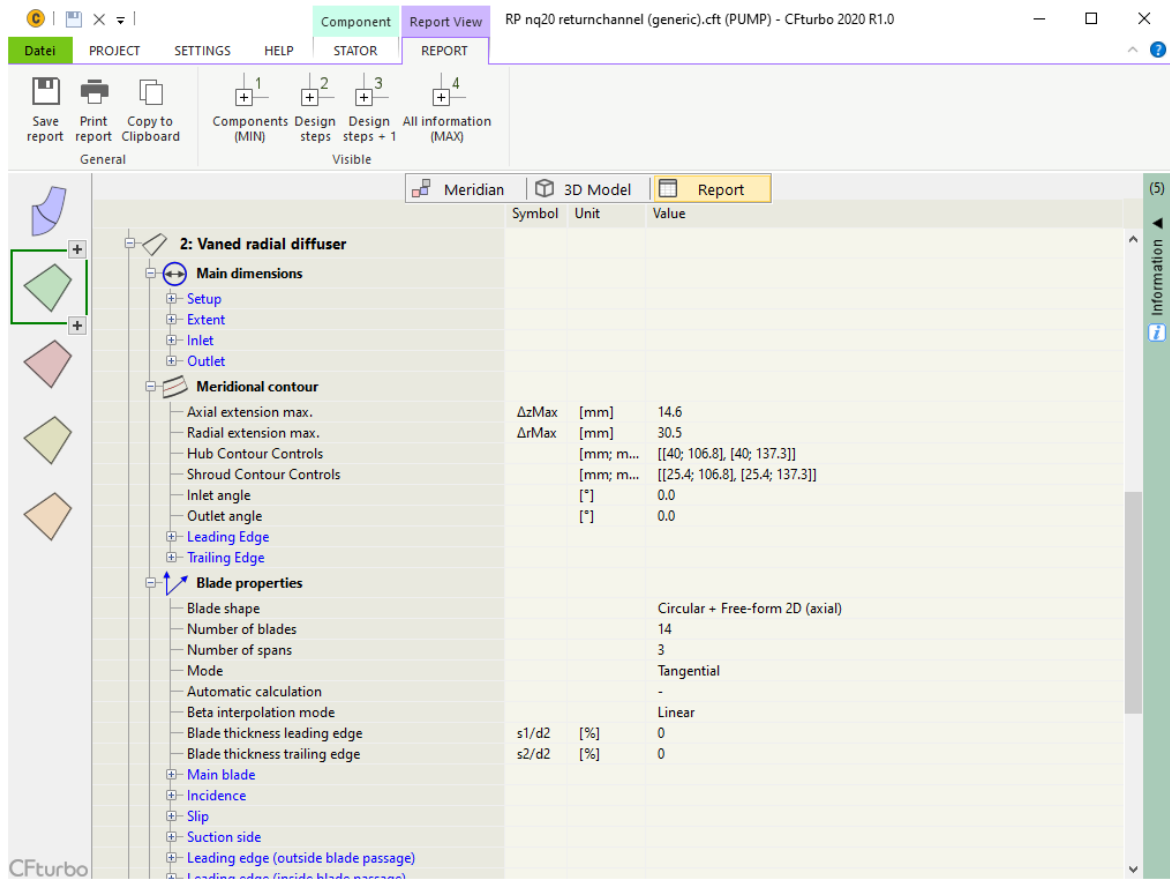
Visualization errors and artifacts can often be resolved by updating the graphic card driver.

See also: [Known problems](#)^[53]





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](#)^[219].

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

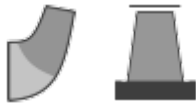
	Save report as HTML, RTF, CSV or TXT
	Print report
	Copy the content to the clipboard All marked rows are copied. If nothing is marked then all content is copied. Marking can be done by mouse, <Ctrl> <A> marks all. Content will be pasted in MS Word/Excel as table.
	Expand nodes to several levels

Part



7 Impeller

? Impeller



This chapter describes in detail the design process for all impeller 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](#) [244]
- [Meridional contour](#) [338]
- [Mean line design](#) [371]
 - [Blade properties](#) [371]
 - [Blade mean lines](#) [405]
 - [Blade profiles](#) [438]
 - [Blade edges](#) [447]
- [Airfoil/ Hydrofoil design](#) [456]
 - [Blade properties](#) [457]
 - [Blade profiles](#) [473]
 - [Blade sweep](#) [475]
- [CFD setup](#) [478]
- [Model settings](#) [486]
- [Model finishing](#) [487]
- [Remove design steps](#) [65]

Possible warnings





Problem	Possible solutions
Neighboring blades are intersecting each other.	
(see message)	<p>Numerous details of the design influence the blade shape. Some examples for possible solutions:</p> <ul style="list-style-type: none"> • Modify main dimensions • Reduce number of blades • Reduce blade wrap angle • Reduce blade thickness

7.1 Main dimensions

? IMPELLER | Main dimensions

The Main Dimensions menu item is used to define main dimensions of the impeller.

Details by impeller type

- [Radial/Mixed-flow Pump / Ventilator](#) 
- [Axial Pump / Ventilator](#) 
- [Centrifugal Compressor](#) 
- [Radial-inflow Turbine](#) 
- [Axial Turbine](#)

Possible warnings

Problem	Possible solutions
<p>Automated Main dimensions are active. Dimensions are not fixed and may adapt to input parameters.</p>	

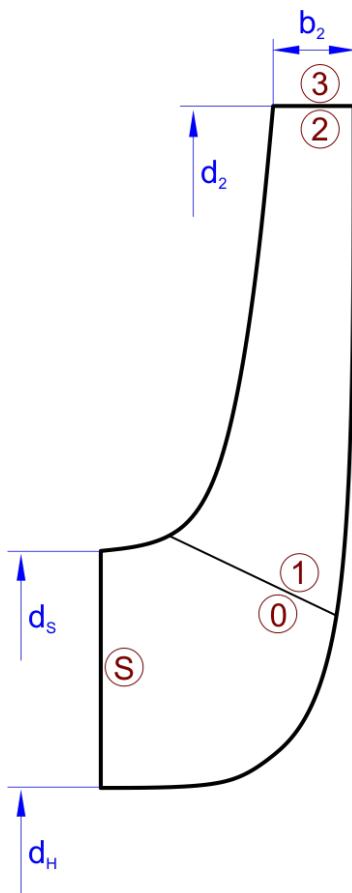
Problem	Possible solutions
Main dimensions are updated automatically if any input parameters are modified.	To fix the main dimensions you could uncheck the "Automatic" calculation. Then you have to manually start the calculation if required.
Automated Main dimensions are NOT active. Dimensions are fixed, but may not reflect input parameters.	
Main dimensions 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.
Specific speed of impeller is beyond the supported range.	
The specific speed nq of the impeller is much too low or too high. The supported range is $nq = 5 \dots 500$.	<p>Modify specific speed defined by n, Q, ρ in the impeller Main dimensions^[336] and/or in the Global Setup^[86].</p> <p>In some cases it may be necessary to select a different impeller type according to the specific speed.</p>
Specific speed of impeller is beyond the recommended range.	
<p>The specific speed is lower or higher than the recommended values.</p> <p>This warning is generated for</p> <ul style="list-style-type: none"> • radial/ mixed-flow impellers with specific speed $nq < 10$ or $nq > 160$ • axial impellers with specific speed $nq < 100$ or $nq > 400$ 	<p>Modify specific speed defined by n, Q, ρ in the impeller Main dimensions^[336] and/or in the Global Setup^[86].</p> <p>In some cases it may be necessary to select a different impeller type according to the specific speed.</p>
High pre-swirl numbers are unusual.	
The impeller is the first component of the project and the inflow swirl is defined by the Global setup ^[86] . With very high pre-swirl an impeller design might be impossible.	Adapt pre-swirl in the Global setup ^[86]
Thermodynamic state could not be calculated for given main dimensions. [compressors and turbines only]	

Problem	Possible solutions
The dimensions might be too tight for the specified mass flow and inlet conditions.	Increase the dimensions (width etc.) or change the Global setup ^[86] (e.g. decrease mass flow).

7.1.1 Radial/Mixed-flow Pump / Ventilator

? Impeller | Main dimensions

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](#)^[247]

→ [Parameters](#)^[249]

→ [Dimensions](#)^[256]

7.1.1.1 Setup

On page **Setup** you can specify some basic settings.

Main dimensions

Setup | Parameters | Dimensions

General

- ☐ Manual dimensioning
- ☐ Unshrouded Tip clearance xIn xOut mm
- ☐ Splitter blades
- Material density ρ kg/m³
- Impeller type
- ☒ Consider upstream swirl

Multi stage options

	Values	Meridian	Cordier	Velocity
Design point				
Volume flow	Q			454 m ³ /h
Rotational speed	n			1770 /min
Mass flow	m			125.88 kg/s
Head	H			30 m
Power output	PQ			37.048 kW
Specific speed (EU)	nq			49
Add'l. Hydraulic efficiency	η_{h+}			90 %

OK Cancel Help

General

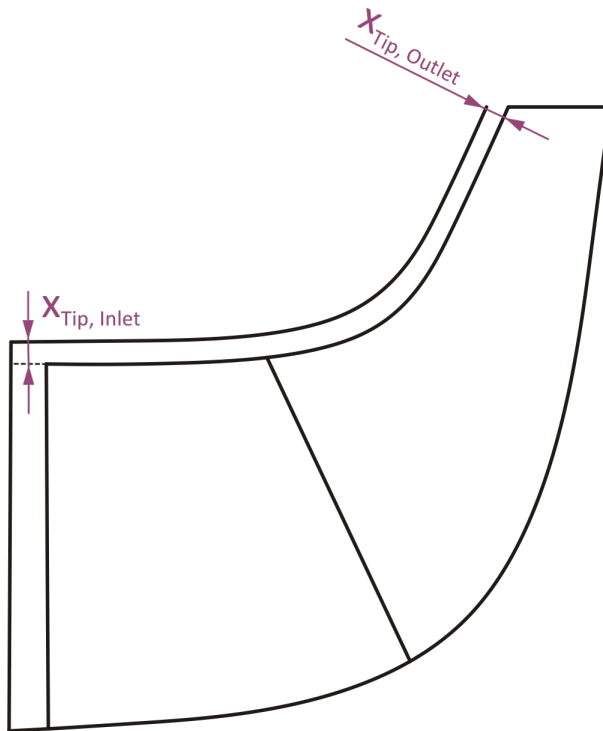
- **Manual dimensioning**


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 ventilators)**
 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](#)³³⁵ by pressing 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.
 - for ventilators: **Squirrel cage**
- Consider upstream swirl**
 If this option is chosen the outlet swirl of the upstream component will be used for the determination of the inlet swirl. If not, cu_1 of the actual component will be zero (no inlet swirl).

Multi stage

For a multi stage design the panel [Multi stage options](#)³³⁶ is available.

Initial default setting

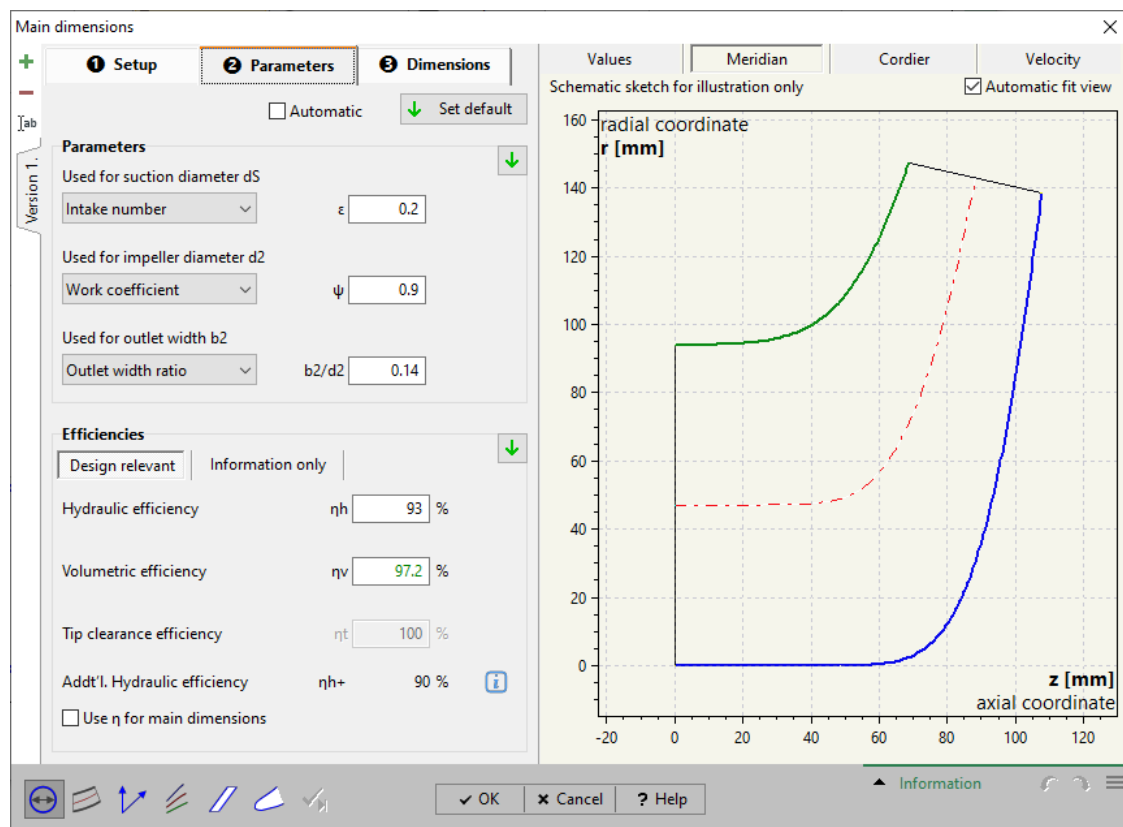
When creating a new design the initial default settings for some important properties are displayed in the panel **Initial default settings**. These settings are used in further design steps and can be modified by selecting the **Change settings** button. Of course these default settings can be modified manually in the appropriate design steps. See [Preferences: Impeller/ Stator settings](#)^[196] for more information.

Information

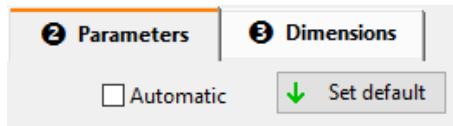
Some design point values are displayed in the right **Information** panel when selecting the page **Values** (see [Global setup](#)^[86]).

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 nq or flow rate Q . Separate functions exist for pumps and ventilators. Additionally some specific functions for waste water pumps are available. See [Approximation functions](#)^[198].

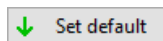


For details of how to handle the parameter edit fields please see [Edit fields with empirical functions](#)^[77].



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](#)^[86]).

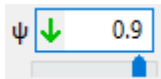
If the automatic mode is not selected the current default values can be specified by one of the following options:



globally by the button on top of the page



regionally by the default button within the **Parameters** or **Efficiency** region

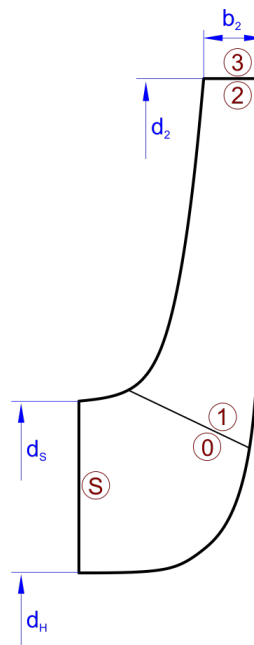


individually by the default button within the input field when selected

Parameters

The panel **Parameters** allows defining alternative parameters in each case for the calculation of the following impeller main dimensions:

for pumps	for ventilators
suction diameter d_s	inlet diameter d_1
	inlet width b_1
impeller diameter d_2	
impeller width b_2	



For d_s -calculation (**pumps**)

Intake coefficient	<ul style="list-style-type: none">▪ Ratio between meridional inflow velocity and specific energy $\varepsilon = c_0 / \sqrt{2Y}$▪ 0.05...0.4 (rising with nq)▪ (k_{m1} at Stepanoff)																		
Inflow angle β_{0a}	<ul style="list-style-type: none">▪ high \rightarrow smaller dimensions, lower friction losses▪ $< 20^\circ \rightarrow$ prevent the risk of cavitation▪ $> 15^\circ \rightarrow$ with regard to efficiency▪ $12^\circ \dots 17^\circ \rightarrow$ recommended for good suction capability																		
Minimal relative velocity w	<ul style="list-style-type: none">▪ small friction and shock losses▪ only if no cavitation risk !▪ $f_{dS}=1.15 \dots 1.05$ standard impeller, $nq=15 \dots 40$▪ $f_{dS}=1.25 \dots 1.15$ suction impeller																		
suction specific speed n_{ss}	<div>$n_{ss} = n \left[\text{min}^{-1} \right] \frac{\sqrt{Q \left[\text{m}^3/\text{s} \right]}}{(\text{NPSH}_R \left[\text{m} \right])^{3/4}}$<p>(European definition for illustration)</p><table><tr><td>Standard suction impeller</td><td>$u_1 < 50 \text{ m/s}$</td><td>160...220</td></tr><tr><td>Suction impeller, axial inflow</td><td>$u_1 < 35 \text{ m/s}$</td><td>220...280</td></tr><tr><td>Suction impeller, cont. shaft</td><td>$u_1 < 50 \text{ m/s}$</td><td>180...240</td></tr><tr><td>High pressure pump</td><td>$u_1 > 50 \text{ m/s}$</td><td>160...190</td></tr><tr><td>Standard inducer</td><td>$u_1 > 35 \text{ m/s}$</td><td>400...700</td></tr><tr><td>Rocket inducer</td><td></td><td>$>> 1000$</td></tr></table></div>	Standard suction impeller	$u_1 < 50 \text{ m/s}$	160...220	Suction impeller, axial inflow	$u_1 < 35 \text{ m/s}$	220...280	Suction impeller, cont. shaft	$u_1 < 50 \text{ m/s}$	180...240	High pressure pump	$u_1 > 50 \text{ m/s}$	160...190	Standard inducer	$u_1 > 35 \text{ m/s}$	400...700	Rocket inducer		$>> 1000$
Standard suction impeller	$u_1 < 50 \text{ m/s}$	160...220																	
Suction impeller, axial inflow	$u_1 < 35 \text{ m/s}$	220...280																	
Suction impeller, cont. shaft	$u_1 < 50 \text{ m/s}$	180...240																	
High pressure pump	$u_1 > 50 \text{ m/s}$	160...190																	
Standard inducer	$u_1 > 35 \text{ m/s}$	400...700																	
Rocket inducer		$>> 1000$																	
Min. NPSH	<div>$\text{NPSH}_R = \lambda_c \frac{c_{m1}^2}{2g} + \lambda_w \frac{w_1^2}{2g}$<ul style="list-style-type: none">▪ λ_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▪ λ_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</div>																		

for d_1 calculation (ventilator)

Diameter ratio d_1/d_2	$\frac{d_1}{d_2} = 1.25 \frac{\sqrt{\psi} \sigma^{5/6}}{\sqrt{\eta_v}}$
--------------------------	---

for b_1 calculation (ventilator)

Meri. deceleration c_{m1}/c_{mS}	$\frac{c_{m1}}{c_{mS}} \approx \frac{d_1}{4b_1} = 2.16 \sigma^{1/6} \leq 2$
---------------------------------------	---

For d_2 -calculation

Work coefficient (= pressure and head coefficient)	<ul style="list-style-type: none"> dimensionless expression for the specific energy: $\psi = Y / (u_2^2 / 2)$ and $\psi = Y_{\text{eff}} / (u_2^2 / 2)$ 0.7 ...1.3 radial impeller 0.25...0.7 mixed-flow impeller 0.1 ...0.4 axial impeller high \rightarrow small d_2, flat characteristic curve low \rightarrow high d_2, steep characteristic curve
Diameter coefficient	<ul style="list-style-type: none"> according to Cordier diagram (see Dimensions ^[256])
Outflow angle β_3	<ul style="list-style-type: none"> 6°...13°: recommended for stable performance curve (with nq rising)

For b_2 -calculation

Outlet width ratio b_2/d_2	<ul style="list-style-type: none"> 0.04...0.30 (rising with nq)
for pumps: Mer. deceleration c_{m3}/c_{mS}	<ul style="list-style-type: none"> 0.60...0.95 (rising with nq)
for pumps:	<ul style="list-style-type: none"> Ratio between meridional outlet velocity and specific energy

Outlet coefficient η_2	<ul style="list-style-type: none"> 0.08...0.26 (rising with nq) $(k_{m2}$ at Stepanoff)
for ventilators: Shroud angle ϵ_{Shr}	$\epsilon_{Shr} = \arctan\left(2 \frac{b_1 - b_2}{d_2 - d_1}\right) = 0^\circ \dots 20^\circ$

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 η_h
- volumetric efficiency η_v
- tip clearance efficiency η_T
- additional hydraulic efficiency η_h^+ (displayed for information only, see [Global setup](#)^[86])

Information only

- side friction efficiency η_s
- mechanical efficiency η_m
- motor efficiency η_{mot}

The additional hydraulic efficiency η_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**.

Internal and mechanical efficiency form the overall efficiency (coupling efficiency) of the stage η_{st} .

When considering motor losses additionally the overall efficiency of the stage incl. motor η_{st}^* is defined.

$$\eta_{St} = \frac{P_Q}{P_D} = \eta_l \eta_m$$

P_Q : pump 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:

classification		efficiencies		Relevant for impeller design
stage	internal	h^+	additional hydraulic	yes: for energy transmission
		h	hydraulic	
		T	tip	
		v	volumetric	yes: for flow rate
		s	side friction	no: for overall information only
stage incl. motor	electrical	m	mechanical	
		mot	motor	

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](#)^[198] 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:

$$\eta_h = \frac{Y}{\tilde{Y}} \approx \sqrt{\eta} \approx 0.85 \dots 0.93$$

The volumetric efficiency is a quantity for the deviation of effective flow rate Q from total flow rate inside the impeller \tilde{Q} which also includes the circulating flow within the pump casing:

$$\eta_v = \frac{Q}{\tilde{Q}} \approx 0.93 \dots 0.99$$

(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 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 β_2 .

$$\eta_T = 1 - f_{\eta} A_{\text{Ratio}} \quad f_{\eta} = f(\eta_q, A_{\text{Ratio}}) \quad A_{\text{Ratio}} \approx x_T / b_2$$

The side friction efficiency contains losses caused by rotation of fluid between hub/ shroud and housing:

$$\eta_s = 1 - \frac{P_s}{P} \approx \begin{matrix} 0.5 \dots 0.985 & \text{für } n_q < 40 \\ 0.985 \dots 0.995 & \text{für } n_q > 40 \end{matrix}$$

The mechanical efficiency mainly includes the friction losses in bearings and seals:

$$\eta_m = 1 - \frac{P_m}{P} \approx 0.95 \dots 0.995$$

(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 for main dimensions (otherwise for B2 only)**" is set, then main dimension calculation is done on the basis of $Y_{\text{eff}} = 0.5(Y/ + Y)$. 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	$P_L = P_D - P_Q = P_D (1 - \eta_{St})$
Internal efficiency	$\eta_I = \eta_h \eta_v \eta_S \eta_T \eta_h^+$
Stage efficiency	$\eta_{St} = \frac{P_Q}{P_D} = \eta_I \eta_m$
Stage efficiency incl. motor	$\eta_{St}^* = \frac{P_Q}{P_{el}} = \eta_{St} \eta_{mot}$

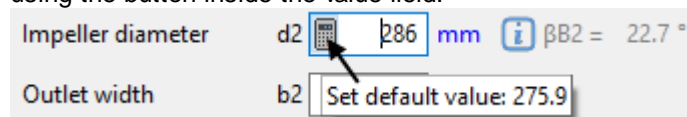
7.1.1.3 Dimensions

On page **Dimensions**, panel **Shaft/ hub**, the required shaft diameter is computed and the hub diameter is determined by the user.

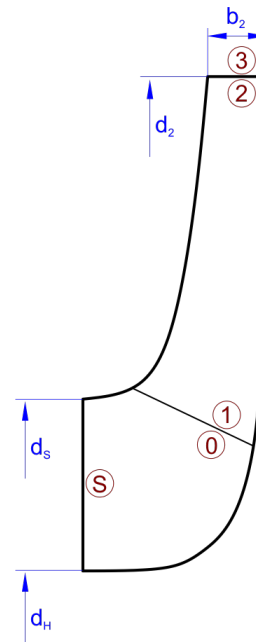
→ [Shaft/Hub](#) 337

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**.

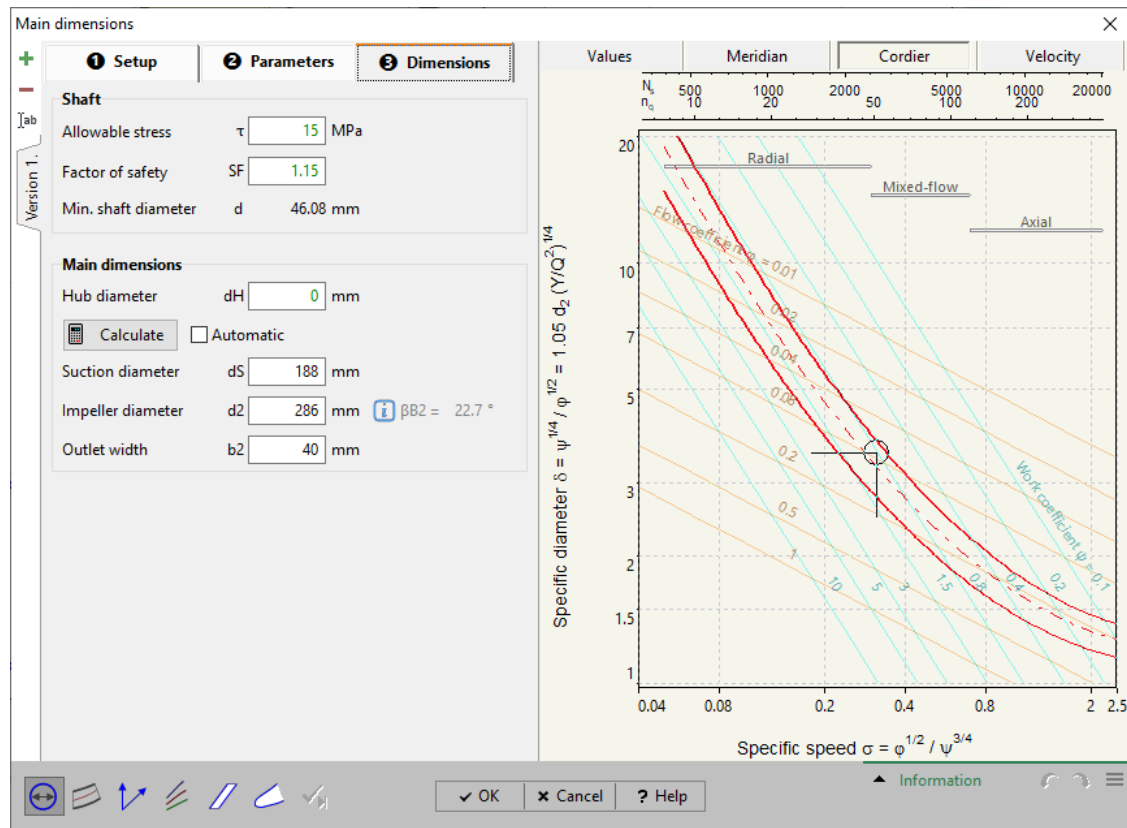
Individual main dimensions can be calculated separately using the button inside the value field.



You may accept the proposed values or you can modify them slightly, e.g. to meet a certain normalized diameter.



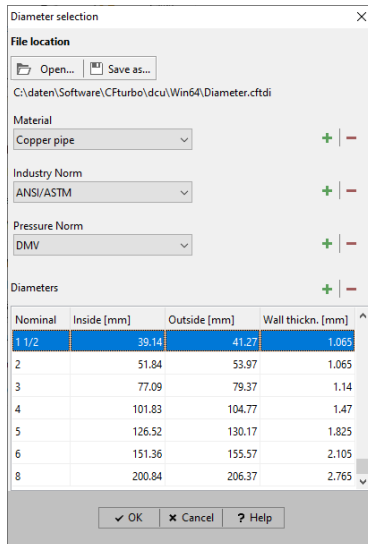
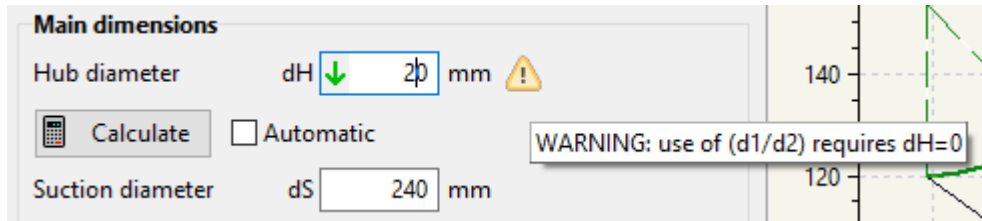
In case the checkbox **Automatic** is activated a new calculation will be accomplished after any change of parameter. Then the manual alteration of the main dimensions is not possible.






Due to the Euler equation the impeller diameter d_2 and the blade angles β_{B2} are coupled (see [Outlet triangle](#)^[395]). Lower d_2 values result in higher β_{B2} (higher blade loading) and vice versa. For that reason the resulting average β_{B2} value is displayed for information right beside the calculated/specified d_2 value.

Impeller diameter d_2 mm i $\beta_{B2} = 22.7^\circ$

A specific problem exists for ventilator 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](#)^[249]).



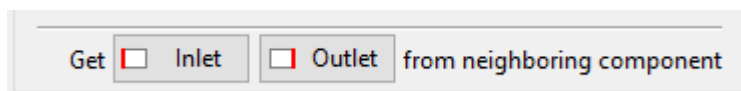
You can select a value for the diameters d_s from standard specifications. For that purpose you have to press the button  right beside the input field.

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.cfdi** 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](#)^[42] components or side-by-side impellers.

Information

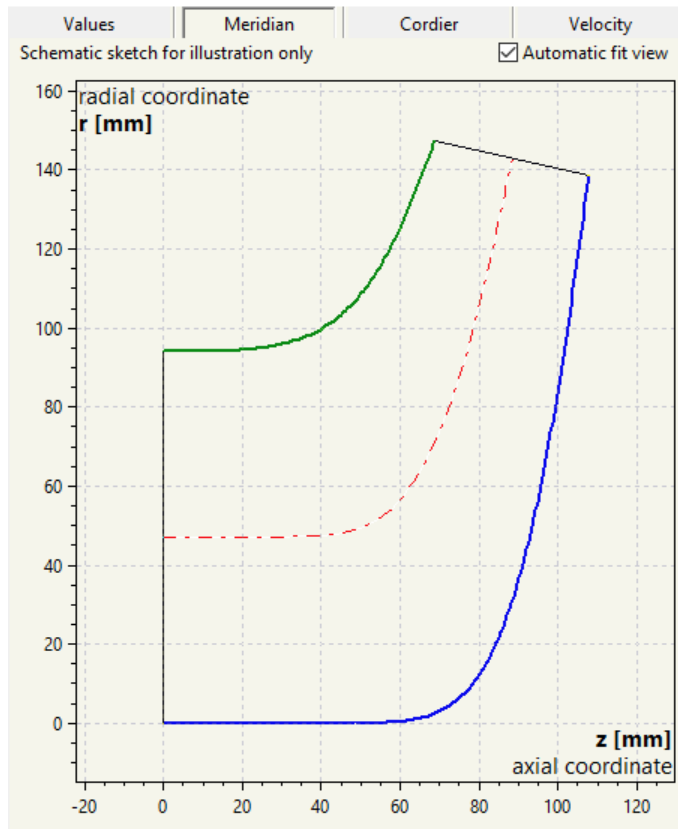
In the right panel of any tab sheet an **information** panel is situated, which holds the computed variables in accordance to the actual state of design, the resulting [Meridional section](#)^[260] as well as the [Cordier-Diagramm](#)^[261] with the location of the best point. These three sections can 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:

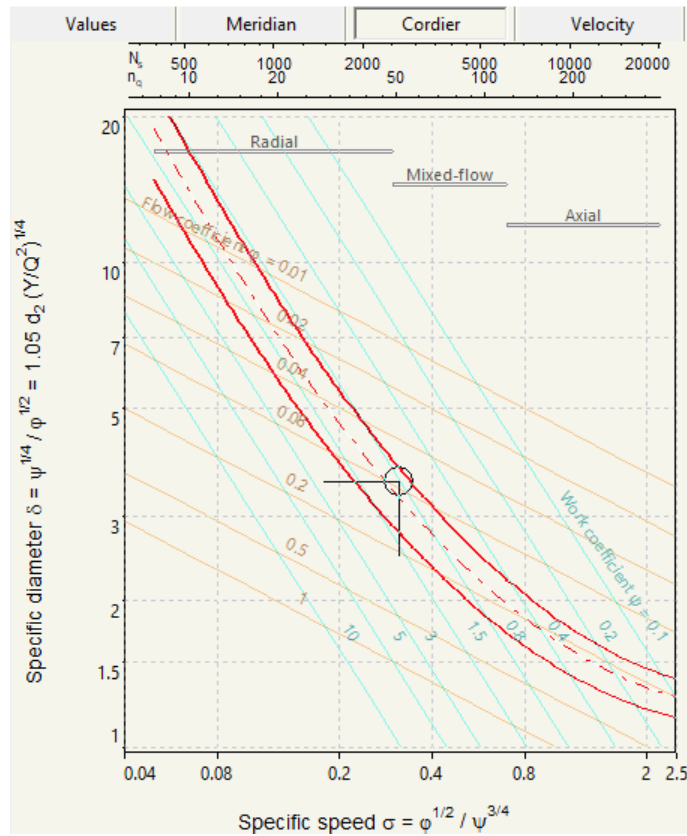
Work coefficient	$\psi = \frac{Y}{u_2^2 / 2}$
Flow coefficient	$\varphi_t = \frac{Q}{\frac{\pi}{4} d_2^2 u_2}$
Meridional flow coefficient	$\varphi_m = \frac{Q}{\pi d_2 b_2 u_2} = \frac{c_{m2}}{u_2}$
Diameter coefficient	$\delta = \frac{\psi^{1/4}}{\varphi_t^{1/2}} = 1.05 d_2 \left(\frac{Y}{Q^2} \right)^{1/4}$
Average inlet velocity	$\bar{c}_{mS} = \frac{Q/\eta_v}{\pi/4 (d_s^2 - d_N^2)}$
Average inlet velocity (net)	$\bar{c}_{mS}^* = \frac{Q}{\pi/4 (d_s^2 - d_N^2)}$
Average outlet velocity	$\bar{c}_{m3} = \frac{Q/\eta_v}{\pi d_2 b_2}$
Average outlet velocity (net)	$\bar{c}_{m3}^* = \frac{Q}{\pi d_2 b_2}$
NPSH _R estimation	<p>Pfleiderer</p> <p>with loss coefficients $c = 1.1 \dots 1.35, \quad w = (0.03) 0.1 \dots 0.3$</p>

	<p>Gülich</p> $\text{NPSH}_R = H \cdot \left(n_q / n_{ss} \right)^{1/3} \quad \text{or}$ $\text{NPSH}_R = \left(h\sqrt{Q} / n_{ss} \right)^{1/3}$ <p>with suction specific speed $n_{ss} = 160 \dots 280$</p>
	<p>Stepanoff</p> $\text{NPSH}_R = \sigma \cdot H$ <p>with cavitation number $\sigma = 1.22 \cdot 10^{-3} \cdot n_q^{4/3}$</p>
	<p>Petermann</p> $\text{NPSH}_R = 1/g \cdot \left(h\sqrt{Q} / S_q \right)^{1/3}$ <p>with suction number $S_q = (0.2) \ 0.4 \dots 0.6 \ (2.0)$</p>
	<p>Europump</p> $\text{NPSH}_R = (0.3 \dots 0.5) \cdot n\sqrt{Q}$
Outlet width ratio	b_2/d_2
Meridional deceleration	$d_{cm} = \bar{c}_{m3} / \bar{c}_{m5}$
Estimated axial force	$F_{ax} = 0.9 \rho g H \cdot \pi/4 \left(d_s^2 - d_N^2 \right)$

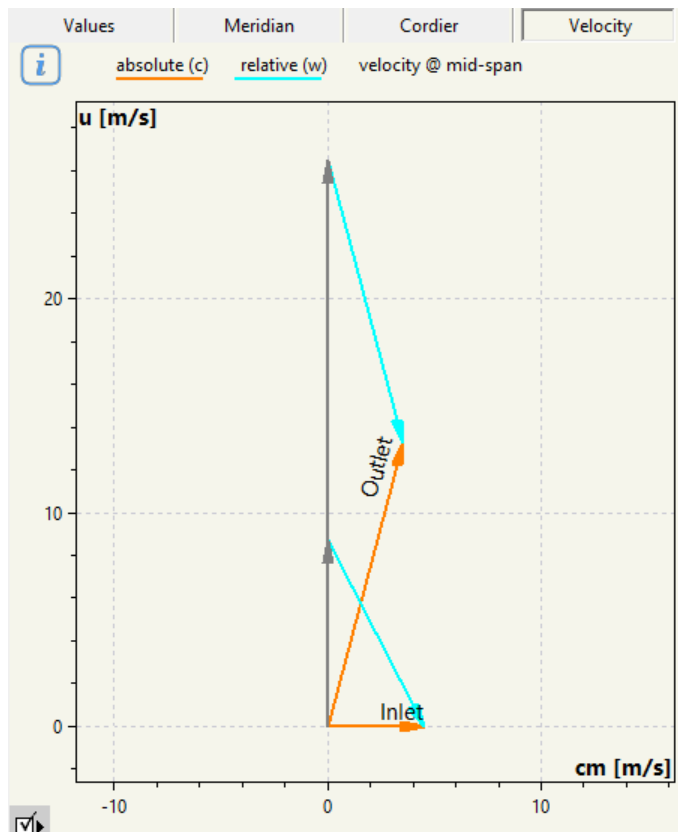
The **Meridional preview** is until now based on the main dimensions only.



The **Cordier diagram** is based on an intensive empirical analysis of proved turbomachinery using extensive experimental data.



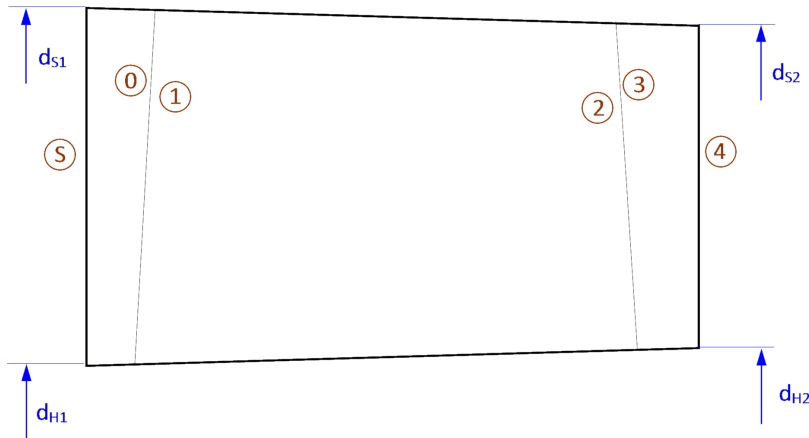
The **Velocity triangles** are the result of a mid-span calculation and are based on the [design point](#)^[86] and the main dimensions.



7.1.2 Axial Pump / Ventilator

? 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).

Details

→ [Setup](#) ²⁶⁵

→ Pump:
[Parameters](#) ²⁶⁷

→ Ventilator:
[Parameters](#) ²⁷⁴

→ [Dimensions](#) ²⁸⁰

7.1.2.1 Setup

On page **Setup** you can specify some basic settings.

Main dimensions

Setup | Parameters | Dimensions

General

- ☐ Manual dimensioning
- ☒ Unshrouded Tip clearance xIn 0.5 xOut 0.5 mm
- Material density ρ 7750 kg/m³
- Impeller type Standard
- ☒ Consider upstream swirl
- Blade design mode Airfoil using pre-defined blade profiles

Multi stage options

Design point

	Values	Meridian	Cordier	Velocity
Volume flow	Q			9540 m ³ /h
Total-to-total pressure difference	Δp_t			0.0336 bar
Rotational speed	n			4500 /min
Mass flow	m			3.18 kg/s
Power output	PQ			8.904 kW
Specific speed (EU)	nq			105
Add'l. Aerodynamic efficiency	η_{h+}			100 %

OK Cancel ? Help

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](#) ^[335] by pressing 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 Ventilators: **Automotive cooling**

- **Consider upstream swirl**

If this option is chosen the outlet swirl of the upstream component will be used for the determination of the inlet swirl. If not, cu_1 of the actual component will be zero (no inlet swirl).

- **Blade design mode**

[Airfoil/ Hydrofoil](#)^[456]: Design according to Airfoil/Hydrofoil design theory.

[Mean line](#)^[371]: 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](#)^[457].

Multi stage

For a multi stage design the panel [Multi stage options](#)^[336] is available.

Initial default setting

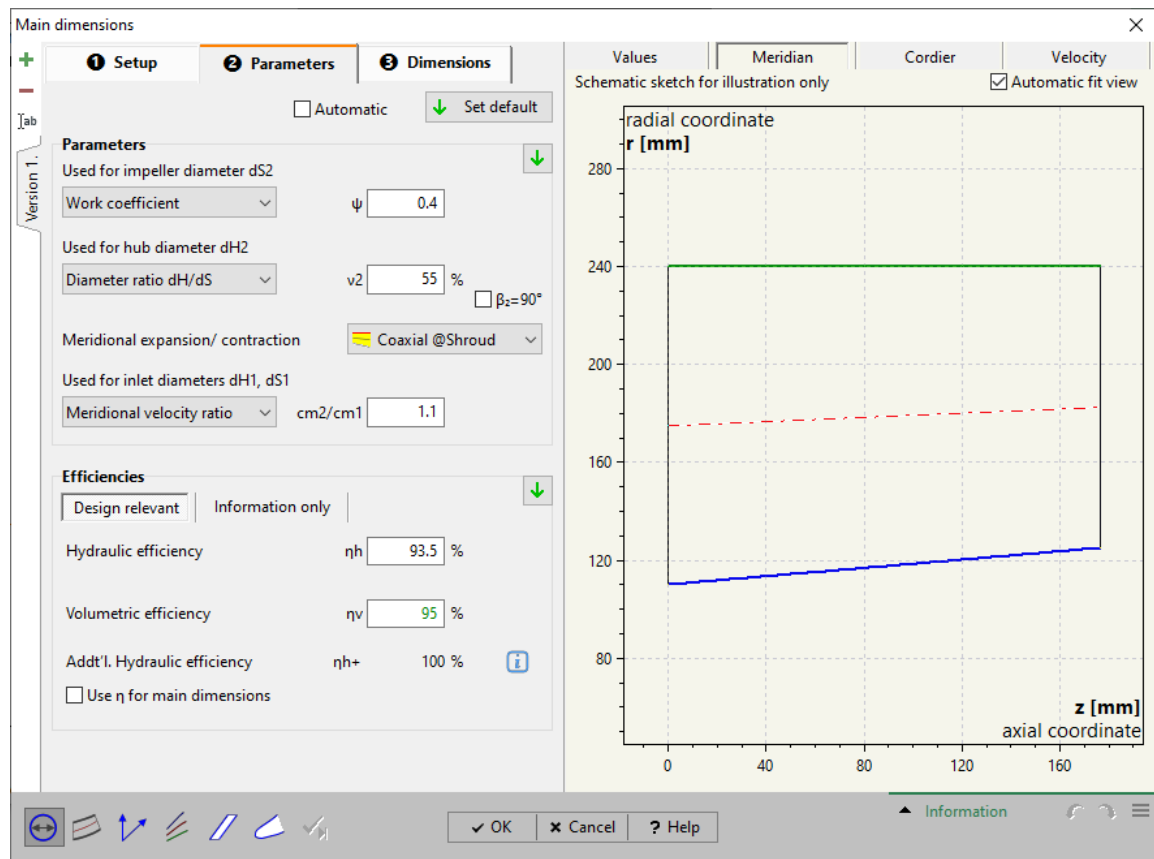
When creating a new design the initial default settings for some important properties are displayed in the panel **Initial default settings**. These settings are used in further design steps and can be modified by selecting the **Change settings** button. Of course these default settings can be modified manually in the appropriate design steps. See [Preferences: Impeller/ Stator settings](#)^[196] for more information.

Information

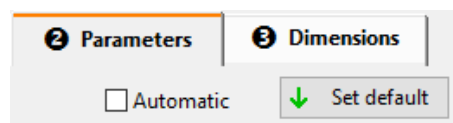
Some design point values are displayed in the right **Information** panel when selecting the page **Values** (see [Global setup](#)^[86]).

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 nq or flow rate Q . See [Approximation functions](#) ^[198].

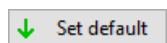


For details of how to handle the parameter edit fields please see [Edit fields with empirical functions](#) ^[77].



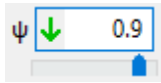
Parameter and efficiency values can be handled manually or can be switched to automatic update by the checkbox on top of the page. Then the default values are used always, even after design point modifications (see [Global setup](#) ^[86]).

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

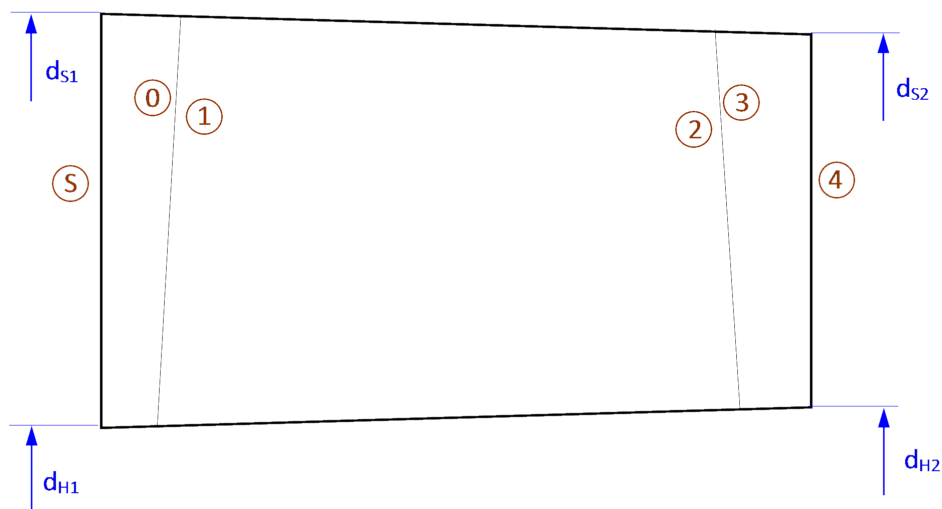


individually by the default button above the input field when selected

Parameters

The panel **Parameters** allows defining alternative parameters in each case for the calculation of the following impeller diameters:

inlet	outlet
d_{S1} , d_{H1}	d_{S2} , d_{H2}



The following is focusing on normal axial pumps - for [inducers](#) ²⁷² special correlations are used.





For d_{S2} -calculation

<p>Work coefficient (= pressure and head coefficient)</p>	<ul style="list-style-type: none"> dimensionless expression for the specific energy: $\psi = Y / (u_2^2 / 2)$ and $\psi = Y_{\text{eff}} / (u_2^2 / 2)$ 0.7 ... 1.3 radial 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
Diameter coefficient	<ul style="list-style-type: none"> according to Cordier diagram (see Dimensions ^[280])

For d_{H2} calculation

Diameter ratio d_{H2}/d_{S2}	$\frac{d_{H2}}{d_{S2}} = 0.4 \dots 0.9$ <p>If the check box " $\alpha_2 = 90^\circ$ " is set the diameter ratio is set to:</p> $\frac{d_{H2}}{d_{S2}} = \frac{\sqrt{Y}}{u_{S2}}$ <p>Under the assumptions: $c_u \cdot u = Y = \text{const.}$</p>
--------------------------------	--

For d_{S1}/d_{H1} -calculation

Meridional velocity ratio c_{m2}/c_{m1}	$\frac{c_{m2}}{c_{m1}} = 0.9 \dots 1.1$
Diameter ratio d_{H1}/d_{S1}	$\frac{d_{H1}}{d_{S1}} = 0.4 \dots 0.9$ <div><div> strictly axial</div><div>$d_{H2} = d_{H1}$ and $d_{S2} = d_{S1}$</div><div> const. hub</div><div>$d_{H2} = d_{H1}$</div><div> const. mid</div><div>$d_{M2} = d_{M1}$</div><div> const. shroud</div><div>$d_{S2} = d_{S1}$</div></div>

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 η_h
- volumetric efficiency η_v
- additional hydraulic efficiency η_h^+ (displayed for information only, see [Global setup](#) ^[86])

Information only

- mechanical efficiency η_m
- motor efficiency η_{mot}

The additional hydraulic efficiency η_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**.

$$\eta_l = \eta_h \cdot \eta_v \cdot \eta_h^+$$

Internal and mechanical efficiency form the overall efficiency (coupling efficiency) of the stage η_{st} .

When considering motor losses additionally the overall efficiency of the stage incl. motor η_{st}^* is defined.

$$\eta_{st} = \frac{P_Q}{P_D} = \eta_l \eta_m$$

P_Q : pump 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:

classification	efficiencies	Relevant for impeller design
----------------	--------------	---------------------------------

stage	internal	h^+	additional hydraulic	yes: for energy transmission
		h	hydraulic	
		v	volumetric	yes: for flow rate
		m	mechanical	no: for overall information only
stage incl. motor	electrical	mot	motor	

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](#)^[198] 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 Q from total flow rate inside the impeller \tilde{Q} which also includes the circulating flow within the ventilator:

$$\eta_v = \frac{Q}{\tilde{Q}} \approx 0.70 \dots 0.95$$

(rising with decreasing tip clearance)

The mechanical efficiency mainly includes the friction losses in bearings and seals:

$$\eta_m = 1 - \frac{P_m}{P} \approx 0.95 \dots 0.995$$

(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 for main dimensions (otherwise for B2 only)**" is set, then main dimension calculation is done on the basis of $Y_{eff} = 0.5(Y/ + Y)$. 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	$P_L = P_D - P_Q = P_D (1 - \eta_{St})$
Internal efficiency	$\eta_I = \eta_h \cdot \eta_v \cdot \eta_h^+$
Stage efficiency	$\eta_{St} = \frac{P_Q}{P_D} = \eta_I \eta_m$
Stage efficiency incl. motor	$\eta_{St}^* = \frac{P_Q}{P_{el}} = \eta_{St} \eta_{mot}$

7.1.2.2.1 Inducer

Inducers are placed in front of radial pump impellers normally in order to improve the suction performance (reduce $NPSH_R$) of the pump.

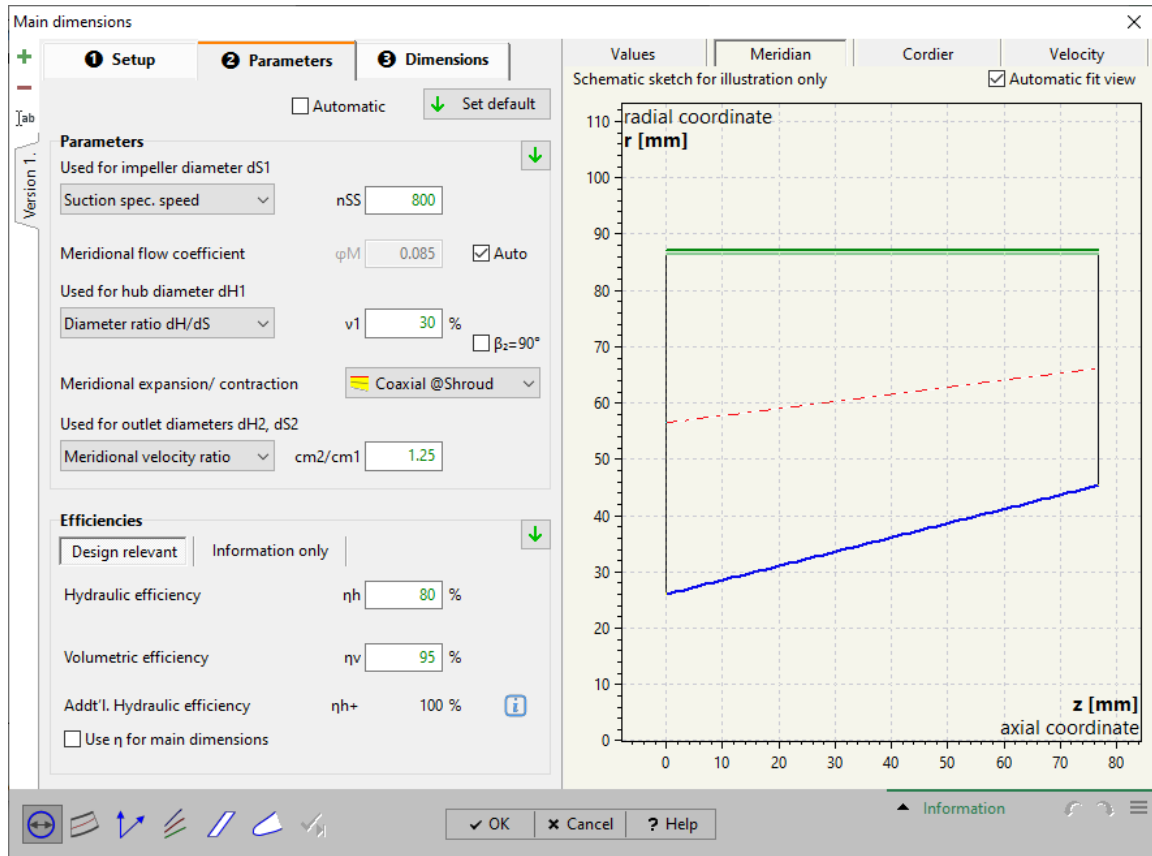
For inducers the inlet section is the primary one. The important suction diameter d_{S1} is calculated using the meridional flow coefficient φ_m :

$$\varphi_m = \frac{Q}{A_S u_{S1}} = \frac{4Q}{\pi(d_{S1}^2 - d_{H1}^2) \pi d_{S1} n} = \frac{c_{m1}}{u_{S1}} = \tan \beta_{0S}$$

In CFturbo the so called Brumfield curve is used to estimate an appropriate φ_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](#) ¹⁹⁵⁾)

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 m value can be calculated automatically from the given n_{ss} value or modified manually. There is a limit of $m \approx 0.06$, lower values will result in backflow at blade tip and cavitation induced flow instability.

Alternatively you can specify the rel. inlet flow angle α_{os} or the meridional flow coefficient m directly. Furthermore the parameters for [classic axial pump](#) ^[267] design could be used alternatively.

The inlet hub diameter d_{H1} is calculated using the diameter ratio v_1 :

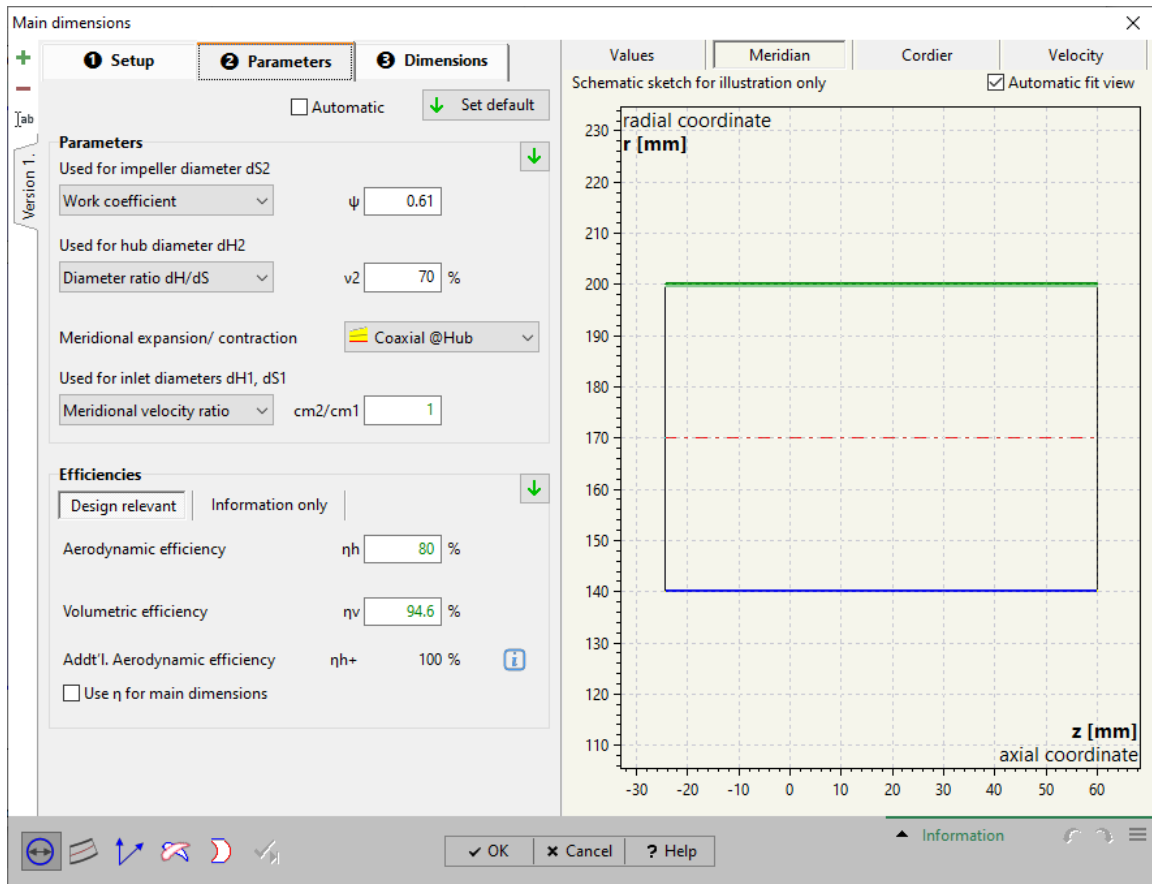
$$v_1 = \frac{d_{H1}}{d_{S1}} \approx 0.2 \dots 0.4$$

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:

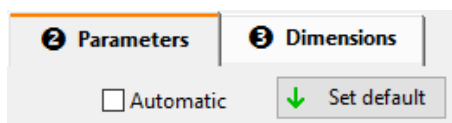
Alternatively the diameter ratio $v_2 = d_{H2}/d_{S2}$ at outlet similar to the inlet side can be used.

7.1.2.3 Parameters Ventilator

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](#) ^[198].

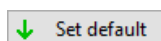


For details of how to handle the parameter edit fields please see [Edit fields with empirical functions](#) ^[77].



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](#) ^[86]).

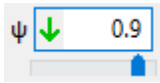
If the automatic mode is not selected the current default values can be specified by one of the following options:



globally by the button on top of the page



regionally by the default button within the **Parameters** or **Efficiency** region

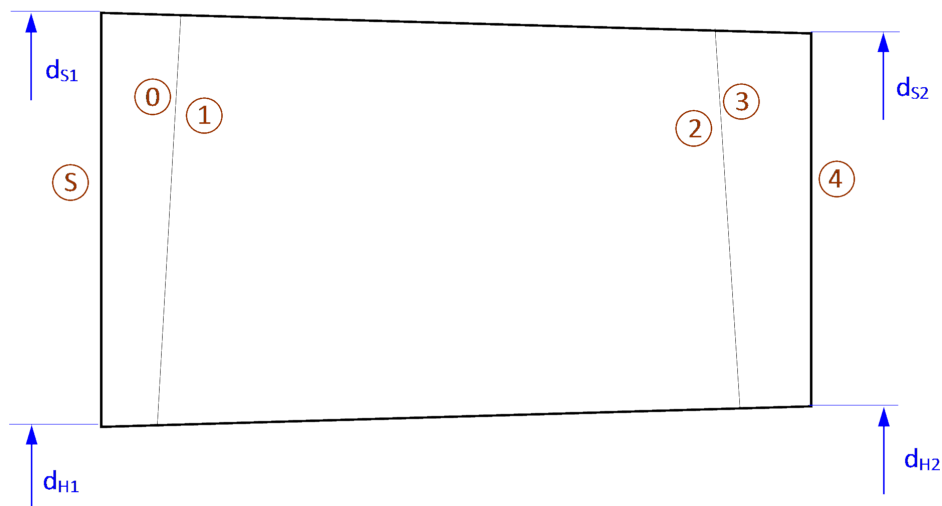


individually by the default button within the input field when selected

Parameters

The panel **Parameters** allows defining alternative parameters in each case for the calculation of the following impeller diameters:

inlet	outlet
d_{S1} , d_{H1}	d_{S2} , d_{H2}



For d_{S2} -calculation





<p>Work coefficient (= pressure and head coefficient)</p>	<ul style="list-style-type: none"> dimensionless expression for the specific energy: <ul style="list-style-type: none"> 0.7 ...1.3 radial 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
---	---

Diameter coefficient	<ul style="list-style-type: none"> according to Cordier diagram (see Dimensions ²⁹⁷)
Flow coefficient φ_t	<ul style="list-style-type: none"> dimensionless flow rate $\varphi_t = \frac{Q_{t,s}}{\frac{\pi}{4} d_2^2 u_2}$ 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}	<p>Can be estimated under assumption $\alpha_2 = 90^\circ$ @ hub and $c_u \cdot u = \text{const.}$ by:</p> $\frac{d_{H2}}{d_{S2}} = \sqrt{\frac{\psi}{2}}$
--------------------------------	--

For d_{S1}/d_{H1} -calculation

Meridional velocity ratio c_{m2}/c_{m1}	$\frac{c_{m2}}{c_{m1}} = 0.9 \dots 1.1$
Diameter ratio d_{H1}/d_{S1}	$\frac{d_{H1}}{d_{S1}} = 0.4 \dots 0.9$ <div style="display: flex; align-items: flex-start;"> <div style="margin-right: 20px;">  strictly axial  const. hub  const. mid  const. shroud </div> <div> $d_{H2} = d_{H1}$ and $d_{S2} = d_{S1}$ $d_{H2} = d_{H1}$ $d_{M2} = d_{M1}$ $d_{S2} = d_{S1}$ </div> </div>

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 η_{tt}

- volumetric efficiency η_v
- additional total-total efficiency η_{tt}^+ (displayed for information only, see [Global setup](#)^[86])

Information only

- mechanical efficiency η_m
- motor efficiency η_{mot}

The additional total-total efficiency η_{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 η_{St} .

When considering motor losses additionally the overall efficiency of the stage incl. motor η_{St}^* is defined.

$$\eta_{St} = \frac{P_Q}{P_D} = \eta_I \eta_m$$

P_Q : ventilator 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:

classification		efficiencies		Relevant for impeller design
stage	impeller	η_{tt}^+	additional total-total	yes: for energy transmission
		η_{tt}	total-total	
		η_v	volumetric	yes: for flow rate

		m	mechanical	no: for overall information only
stage incl. motor	electrical	mot	motor	

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](#)^[198] 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 Q from total flow rate inside the impeller \tilde{Q} which also includes the circulating flow within the ventilator:

$$\eta_v = \frac{Q}{\tilde{Q}} \approx 0.70 \dots 0.95$$

(rising with decreasing tip clearance)

The mechanical efficiency mainly includes the friction losses in bearings and seals:

$$\eta = 1 - \frac{P_m}{P} \approx 0.95 \dots 0.995$$

(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.


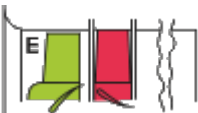


If the check box "**Use for main dimensions (otherwise for B2 only)**" is set, then main dimension calculation is done on the basis of $Y_{\text{eff}} = 0.5(Y/ + Y)$. Otherwise Y - specific work without losses - is used.

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	$P_L = P_D - P_Q = P_D (1 - \eta_{St})$
Internal efficiency	$\eta_{lm} = \eta_{tt} \eta_v$
Stage efficiency	$\eta_{St} = \frac{P_Q}{P_D} = \eta_l \eta_m$
Stage efficiency incl. motor	$\eta_{St}^* = \frac{P_Q}{P_{el}} = \eta_{St} \eta_{mot}$

Assembly	Pressure difference	Remark
	$\Delta p_{tt} = \Delta p_{tt, Imp} - (\zeta_D + \zeta_{GV}) \frac{\rho}{2} c_{m2}^2$ $\zeta_D = 0.1 \cdot \left(2 \left(\frac{d_{H2}}{d_{S2}} \right)^2 - \left(\frac{d_{H2}}{d_{S2}} \right)^4 \right)$ $\zeta_{GV} = 0.1 \cdot \left(1 - \frac{c_{u2}^2}{c_{m2}^2} \right)$	<p>inline installation,</p> <p>resistance coefficients according to Wallis ^[567]</p>
	$\Delta p_{tt} = \Delta p_{tt, Imp} - (\zeta_D + \zeta_{GV}) \frac{\rho}{2} c_{m2}^2$ $\zeta_D = \left(1 - \left(-0.5 \frac{d_{H2}}{d_{S2}} + 0.95 \right) \right) \cdot \left(2 \left(\frac{d_{H2}}{d_{S2}} \right)^2 - \left(\frac{d_{H2}}{d_{S2}} \right)^4 \right)$ $\zeta_{GV} = 0.1 \cdot \left(1 - \frac{c_{u2}^2}{c_{m2}^2} \right)$	<p>inline installation, Carnot type diffuser</p> <p>resistance coefficients according to Wallis ^[567]</p>
		<p>inline installation, swirl assumed to dissipate in duct → pseudo total-to-total pressure rise</p> <p>resistance coefficients according to Wallis ^[567]</p>

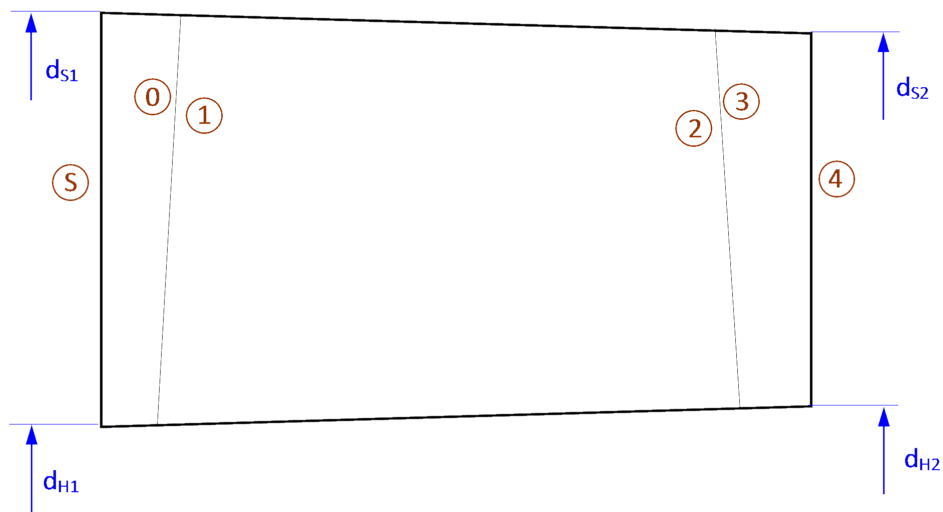
	$\Delta p_{ts} = \Delta p_{tt,A} - \zeta_D \frac{\rho}{2} c_a^2$ $c_a = c_{m2} \cdot \left(1 - \left(\frac{d_{H2}}{d_{S2}} \right)^2 \right)$	exhaust installation potential pressure recovery in exhaust jet not taken into account
	$\Delta p_{ts} = \Delta p_{tt,B} - \zeta_D \frac{\rho}{2} c_a^2$ $c_a = c_{m2} \cdot \left(1 - \left(\frac{d_{H2}}{d_{S2}} \right)^2 \right)$	
	$\Delta p_{ts} = \Delta p_{tt,Imp} - \zeta_{GV} \frac{\rho}{2} c_{m2}^2 - \frac{\rho}{2} c_{m2}^2$ $\zeta_{GV} = 0.1 \cdot \left(1 - \frac{c_{u2}^2}{c_{m2}^2} \right)$	
	$\Delta p_{ts} = \Delta p_{tt,Imp} - \frac{\rho}{2} c_{m2}^2 - \frac{\rho}{2} c_{u2}^2$	

7.1.2.4 Dimensions

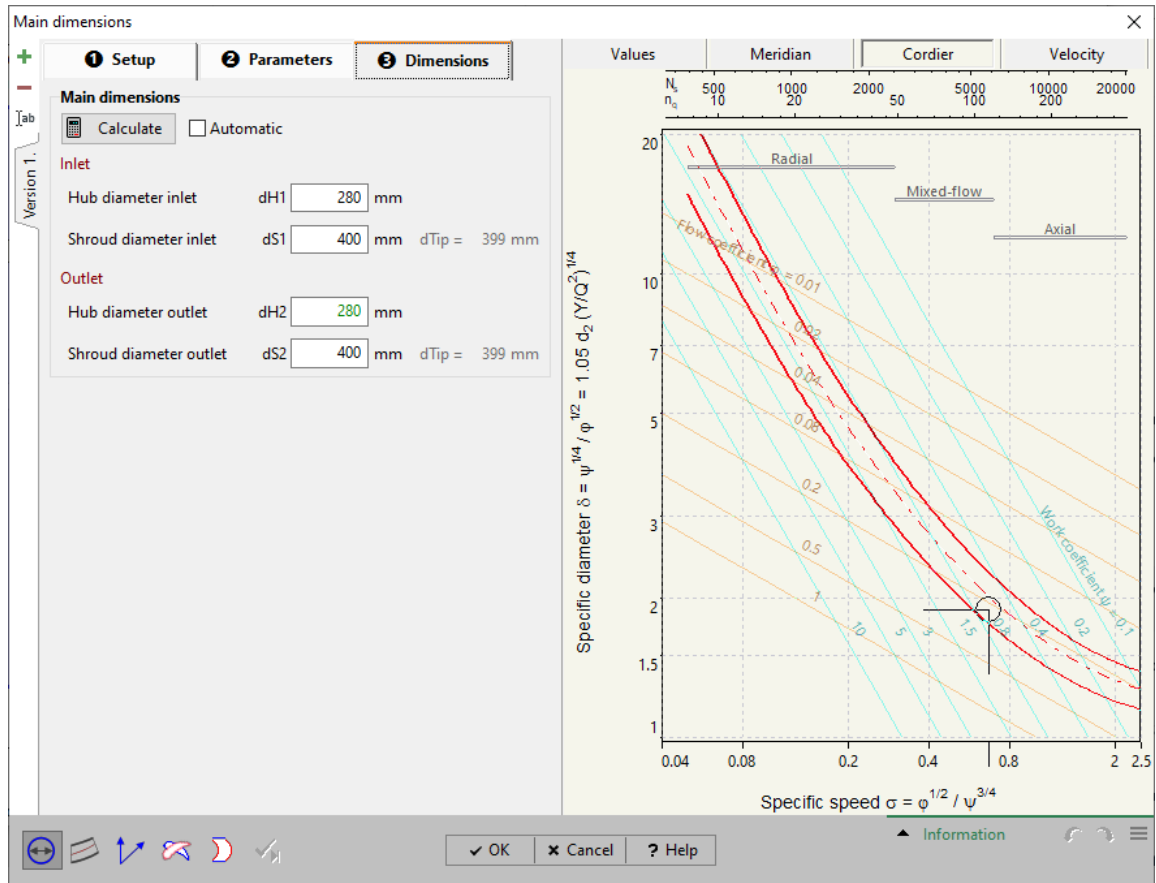
The main dimensions of a designed impeller - suction diameter d_{S1} and d_{H1} and outlet diameter d_{S2} and d_{H2} - can be seen on **Main dimensions** panel. They can be recomputed by pressing the **Calculate**-button. The computation is based on "Euler's Equation of Turbomachinery", on the continuity 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.

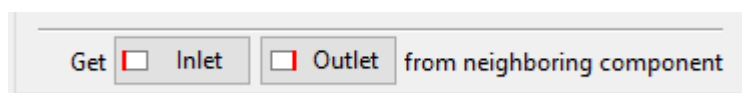


In case the checkbox **Automatic** is activated a new calculation will be accomplished after any change of parameter. Then the manual alteration of the main dimensions is not possible.



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](#)^[42] components or side-by-side impellers.

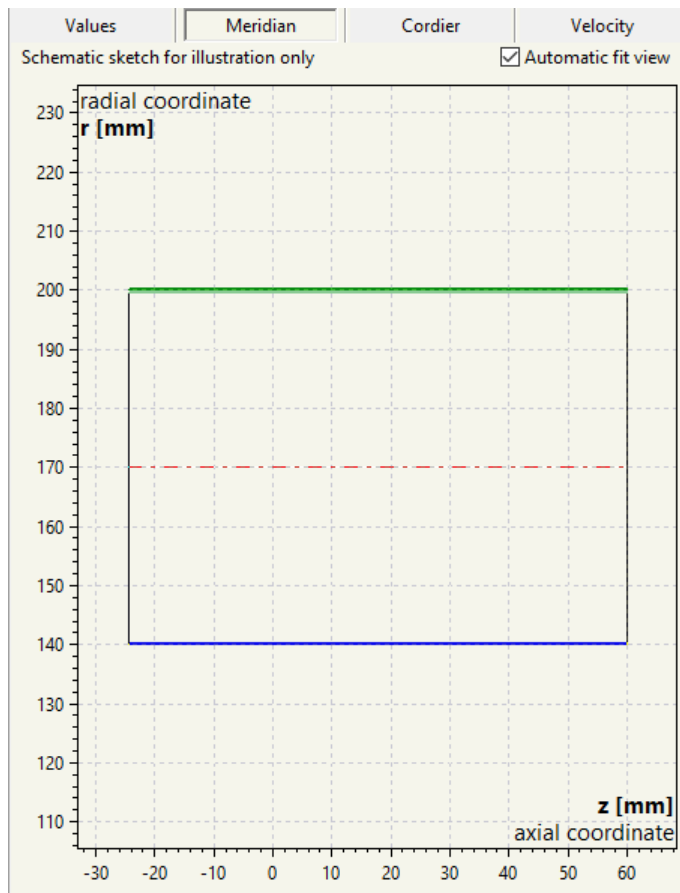
Information

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](#)^[260] as well as the [Cordier-Diagramm](#)^[261] with the location of the best point. These three sections can be chosen by the appropriate soft buttons in the heading.

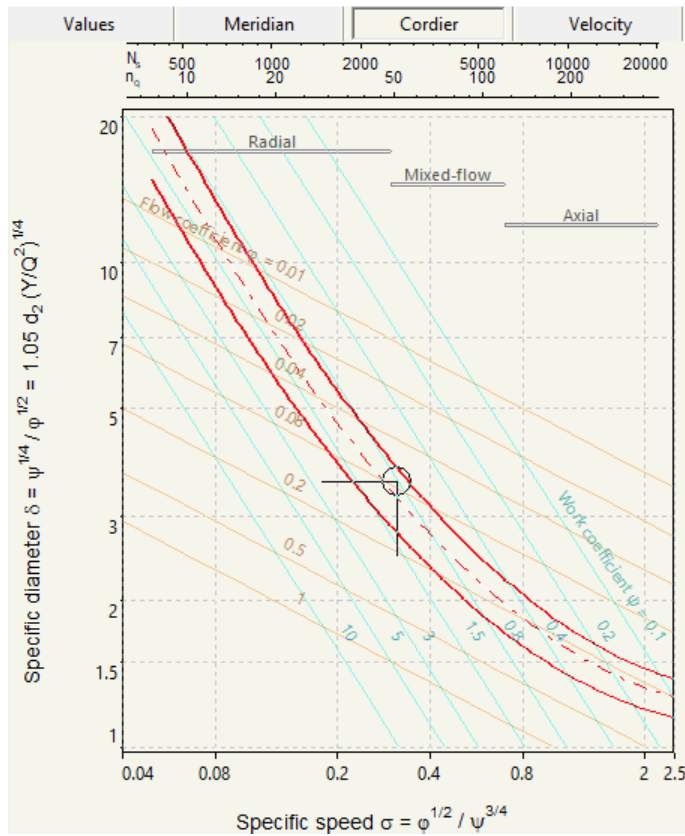
In the **Value** section the following variables are displayed for information which result from calculated or determined main dimensions:

Work coefficient	$\Psi = \frac{Y}{u_{2S}^2/2}$
Flow coefficient	$\varphi_t = \frac{Q}{\frac{\pi}{4} d_{2S}^2 u_{2S}}$
Meridional flow coefficient	$\varphi_m = \frac{Q_2}{\frac{\pi}{4} (d_{2S}^2 - d_{2H}^2) u_2} = \frac{c_{m2}}{u_2}$
Diameter coefficient	$\delta = \frac{\Psi^{1/4}}{\varphi_t^{1/2}} = 1.05 d_{2S} \left(\frac{Y}{Q_{tS}^2} \right)^{1/4}$
Average inlet velocity	$c_{m1} = \frac{Q}{\pi/4 (d_{S1}^2 - d_{H1}^2)}$
Inlet abs. circ. velocity component	c_{u1}
Inlet relative velocity	w_1
Average outlet velocity	$c_{m2} = \frac{Q}{\pi/4 (d_{S2}^2 - d_{H2}^2)}$
Outlet circ. velocity component	$c_{u2} = \frac{1}{u_2} (Y - u_1 c_{u1})$
Outlet relative velocity	w_2
Meridional velocity ratio	
Relative velocity ratio	

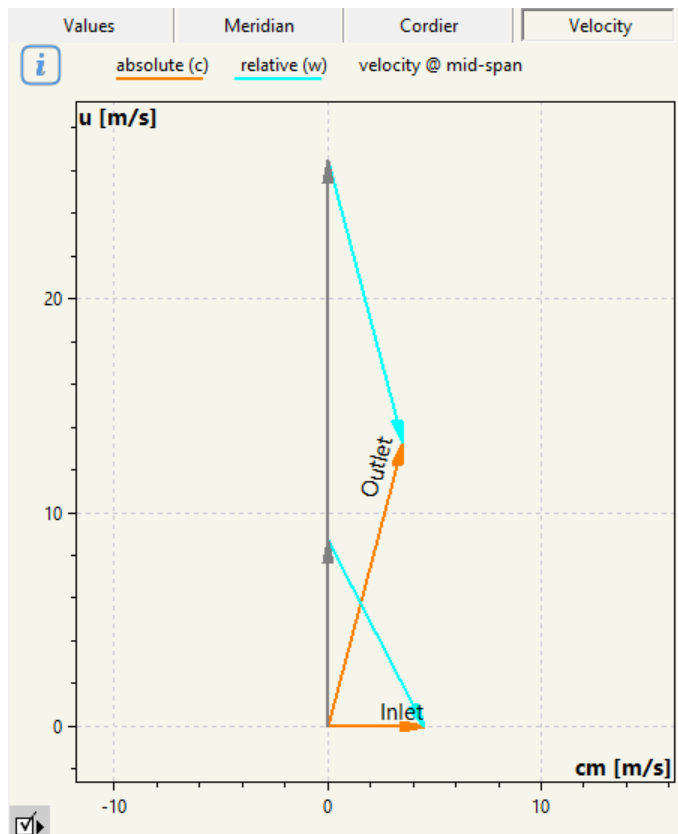
The **Meridional preview** is until now based on the main dimensions only.



The **Cordier diagram** is based on an intensive empirical analysis of proved turbomachinery using extensive experimental data.



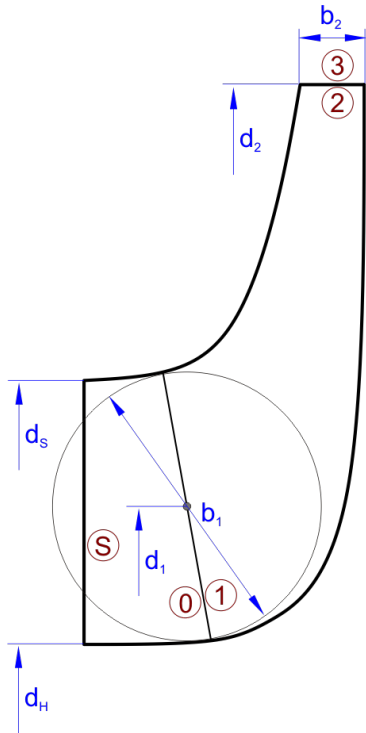
The **Velocity triangles** are the result of a mid-span calculation and are based on the [design point](#)^[86] and the main dimensions.



7.1.3 Centrifugal Compressor

? Impeller | Main dimensions

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](#) ²⁸⁸

→ [Parameters](#) ²⁹⁰

→ [Dimensions](#) ²⁹⁷

7.1.3.1 Setup

On page **Setup** you can specify some basic settings.

Main dimensions

Setup | Parameters | Dimensions

General

- ☐ Manual dimensioning
- ☒ Unshrouded Tip clearance xIn 1.5 xOut 2 mm
- ☒ Splitter blades
- Material density ρ 7750 kg/m³
- Impeller type Standard
- ☒ Consider upstream swirl

Multi stage options

Values	Meridian	Cordier	Velocity
Design point			
Rotational speed	n	14000	/min
Mass flow	m	5.31	kg/s
Specific work	Y	85980	m ² /s ²
Power output	PQ	456.55	kW
Specific speed (EU)	nq	33	
Add'l. Total-to-total efficiency	η_{tt+}	100	%
Volume flow (total)	QtS	16139	m ³ /h

OK Cancel Help

General

- Manual dimensioning**

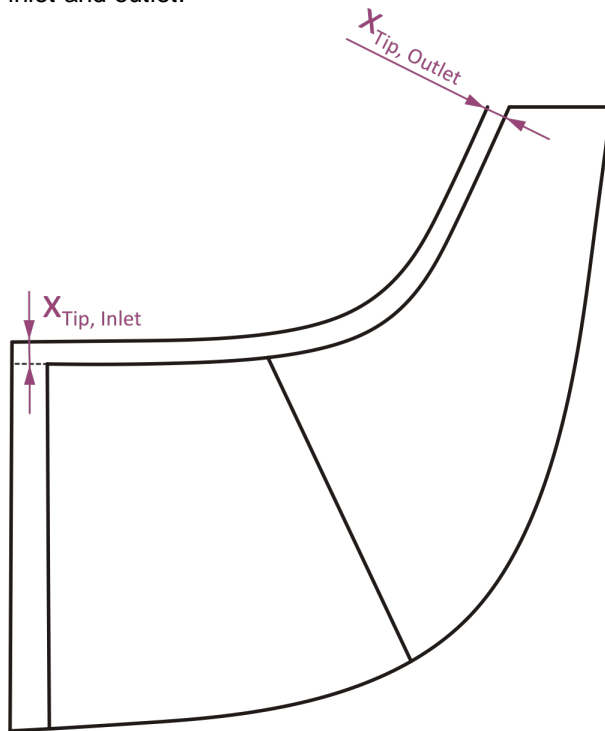
In manual dimensioning mode the main dimensions and blade angles are not calculated by CFturbo. All these values are user-defined input values.

- 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](#) ³³⁵ by pressing button  next to the input area.

- **Consider upstream swirl**

If this option is chosen the outlet swirl of the upstream component will be used for the determination of the inlet swirl. If not, cu_1 of the actual component will be zero (no inlet swirl).

Multi stage

For a multi stage design the panel [Multi stage options](#) ³³⁶ is available.

Initial default setting

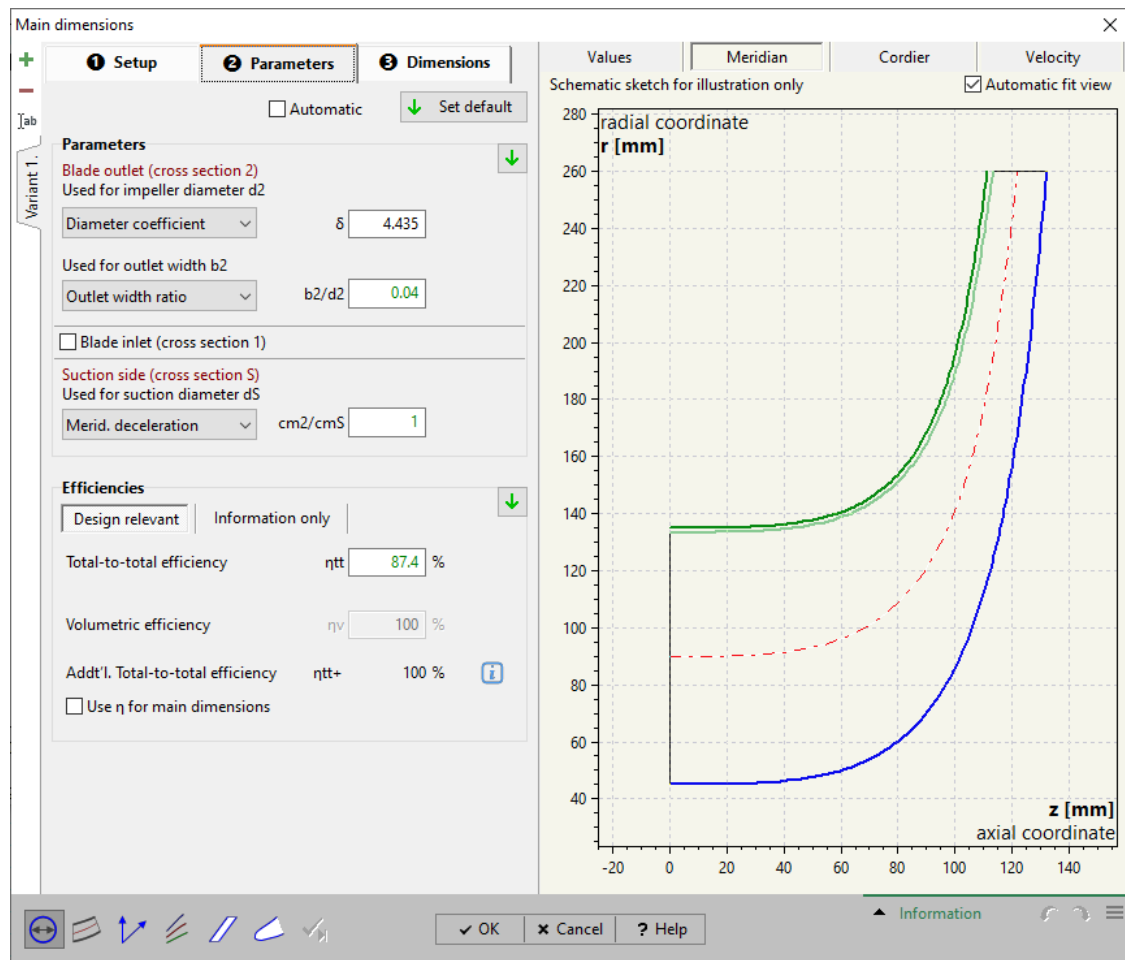
When creating a new design the initial default settings for some important properties are displayed in the panel **Initial default settings**. These settings are used in further design steps and can be modified by selecting the **Change settings** button. Of course these default settings can be modified manually in the appropriate design steps. See [Preferences: Impeller/ Stator settings](#) ¹⁹⁶ for more information.

Information

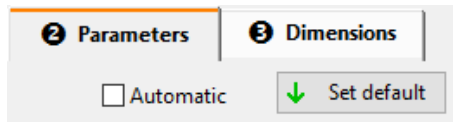
Some design point values are displayed in the right **Information** panel when selecting the page **Values** (see [Global setup](#) ⁸⁶⁾).

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](#) ¹⁹⁸⁾).

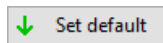


For details of how to handle the parameter edit fields please see [Edit fields with empirical functions](#) ⁷¹⁾.



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](#)^[86]).

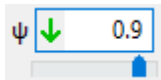
If the automatic mode is not selected the current default values can be specified by one of the following options:



globally by the button on top of the page



regionally by the default button within the **Parameters** or **Efficiency** region

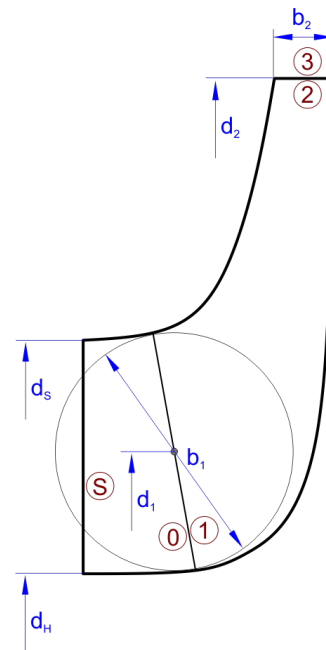


individually by the default button within the input field when selected

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 (= pressure and head coefficient)	<ul style="list-style-type: none"> ▪ dimensionless expression for the specific enthalpy $h_{is}=Y$ and $h=Y_{eff}$ resp.
---	---

	$\psi = \frac{\Delta h_{is}}{u_2^2/2} \quad \text{and} \quad \psi = \frac{\Delta h}{u_2^2/2}$ <ul style="list-style-type: none"> high \rightarrow small d_2, flat characteristic curve low \rightarrow high d_2, steep characteristic curve
(Total) Flow coefficient φ_t	<ul style="list-style-type: none"> dimensionless flow rate $\varphi_t = \frac{Q_{t,s}}{\frac{\pi}{4} d_2^2 u_2}$ <p>0.01 narrow radial impeller, untwisted blades 0.15 mixed-flow impeller, twisted blades</p>
Diameter coefficient	<ul style="list-style-type: none"> according to Cordier diagram (see Dimensions^[297])
Machine Mach number Ma_u	<ul style="list-style-type: none"> dimensionless peripheral speed of impeller related to total inlet speed of sound $Ma_u = \frac{u_2}{a_{t,s}}$
Peripheral speed u_2	<ul style="list-style-type: none"> Limiting values due to strength as a function of the material

For b_2 -calculation

Outlet width ratio b_2/d_2	<ul style="list-style-type: none"> 0.01...0.15 (with nq rising)
Meridional flow coefficient φ_m	<ul style="list-style-type: none"> dimensionless flow rate $\varphi_m = \frac{Q_2}{\pi d_2 b_2 u_2} = \frac{c_{2m}}{u_2}$ <p>0.10...0.50 (with nq rising)</p>

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 $f(b_2/d_2)$
---------------------------------	---------------------------------

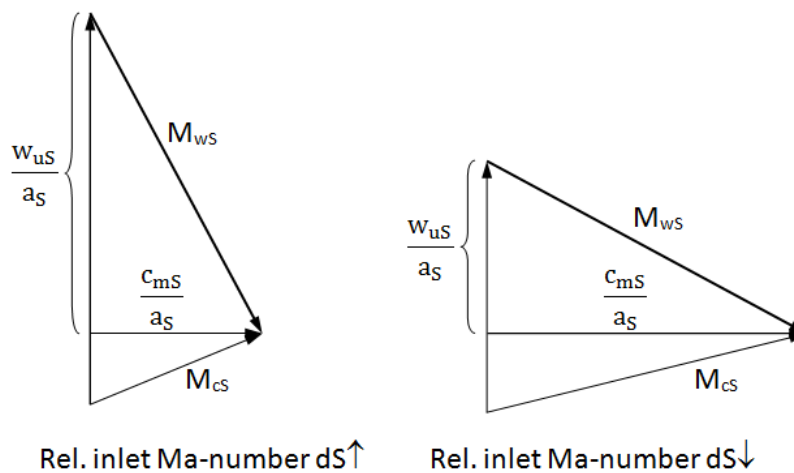
For b_1 -calculation (optional)

Meridional deceleration c_{m2}/c_{m1}	$c_{m2}/c_{m1} = 0.8 \dots 1.25$
---	----------------------------------

for d_s -calculation

Meridional deceleration c_{m1}/c_{mS} or c_{m2}/c_{mS}	$c_{m1}/c_{mS} = 0.9 \dots 1.1$ $c_{m2}/c_{mS} = 0.7 \dots 1.3$
Relative inlet flow angle β_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}	$M_{wS} = \frac{w_s}{a_s} = \frac{\sqrt{c_{mS}^2 + w_{uS}^2}}{a_s} \leq 0.75 \dots 0.85$
Diameter ratio dS/d_2	$dS/d_2 = 0.65 \dots 0.8$

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 η_{tt}
- volumetric efficiency η_v
- additional total-total efficiency η_{tt}^+ (displayed for information only, see [Global setup](#) ^[86])

Information only

- mechanical efficiency η_m
- motor efficiency η_{mot}

The additional total-total efficiency η_{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 η_{st} .

When considering motor losses additionally the overall efficiency of the stage incl. motor η_{st}^* is defined.

$$\eta_{st} = \frac{P_Q}{P_D} = \eta_i \eta_m$$

P_Q : output power, 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:

classification		efficiencies		Relevant for impeller design
stage		c	additional casing	yes: for energy transmission
		tt	total-total	
	impeller	v	volumetric	yes: for flow rate
		m	mechanical	no: for overall information only
stage incl. motor	electrical	mot	motor	

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](#)^[198] are representing the theoretical reachable values and they should be corrected by the user if more information about the impeller or the whole machine are available.

The impeller efficiency η_{tt} describes the energy losses 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 impeller efficiency is the ratio

between the actual specific energy Y and the energy transmitted by the impeller blades without any losses:

$$\eta_{tt} = \frac{Y}{\tilde{Y}}$$

The volumetric efficiency is a quantity for the deviation of effective flow rate Q from total flow rate inside the impeller \tilde{Q} which also includes the circulating flow within the casing:

$$\eta_v = \frac{Q}{\tilde{Q}} \approx 0.93 \dots 0.99$$

(rising with impeller size)

The mechanical efficiency mainly includes the friction losses in bearings and seals:

$$\eta_m = 1 - \frac{P_m}{P} \approx 0.95 \dots 0.995$$

(rising with impeller size)

Impeller efficiency and volumetric efficiency are most important for the impeller dimensioning because of their influence to \tilde{Q} and/or \tilde{Y} . The mechanical efficiency is affecting only the required driving power of the machine.

If the check box "**Use for main dimensions (otherwise for B2 only)**" is set, then main dimension calculation is done on the basis of $h = 0.5(h_{is}/ + h_{is})$. Otherwise 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:

Required driving power	$P_D = \frac{P_Q}{\eta_{st}}$
Power loss	
Internal efficiency	
Stage efficiency	

Stage efficiency incl. motor	$\eta_{St}^* = \frac{P_Q}{P_{el}} = \eta_{St} \eta_{mot}$
Total-to-static efficiency	$\eta_{ts} = \frac{\pi_t^{\frac{\kappa-1}{\kappa}} \left(1 - \frac{c_2^2}{2c_p T_{ts}} \right) - 1}{\tau_t - 1}$ <p>(perfect gas model)</p>
Polytropic efficiency	$\eta_p = \frac{\frac{n}{n-1}}{\frac{\kappa}{\kappa-1}}$ <p>(n .. polytropic exponent .. isentropic exponent)</p>

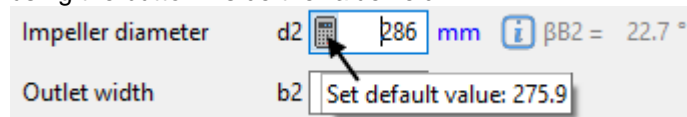
7.1.3.3 Dimensions

On page **Dimensions**, panel **Shaft/ hub**, the required shaft diameter is computed and the hub diameter is determined by the user.

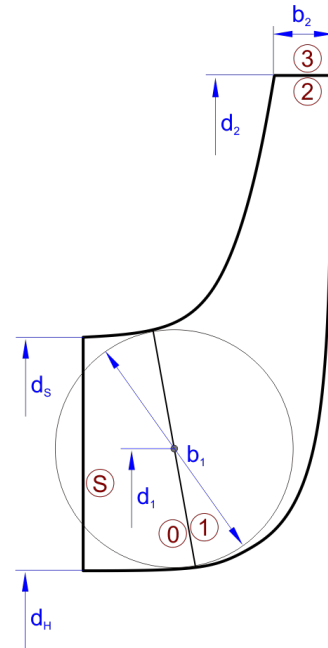
→ [Shaft/Hub](#) 

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**.

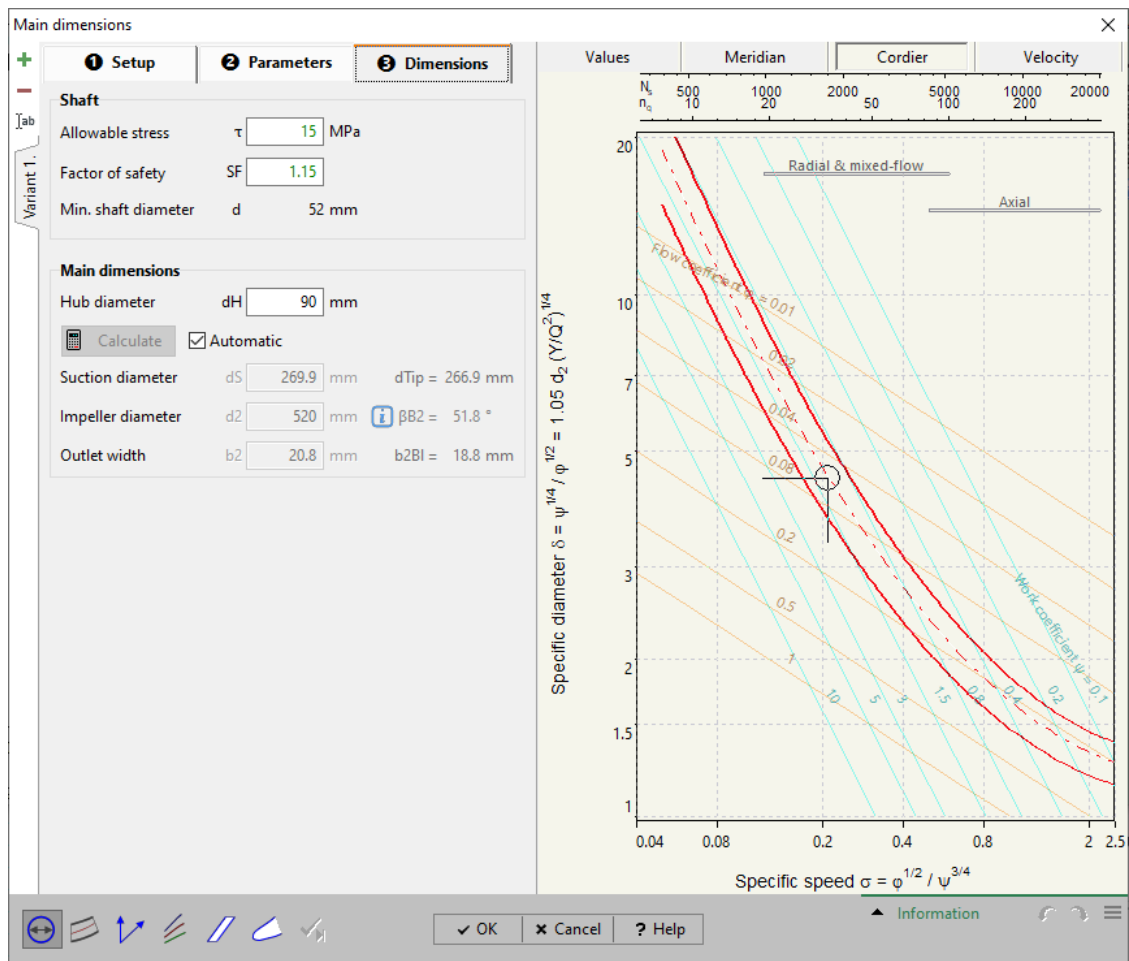
Individual main dimensions can be calculated separately using the button inside the value field.



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 be accomplished after any change of parameter. Then the manual alteration of the main dimensions is not possible.

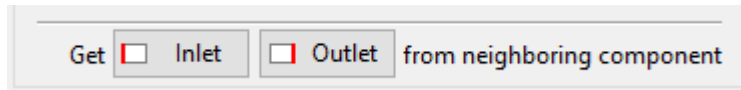


Due to the Euler equation the impeller diameter d_2 and the blade angles β_{B2} are coupled (see [Outlet triangle](#)³⁹⁵). Lower d_2 values result in higher β_{B2} (higher blade loading) and vice versa. For that reason the resulting average β_{B2} value is displayed for information right beside the calculated/ specified d_2 value.

Impeller diameter $d2$ mm $\beta_{B2} \approx 52^\circ$ [i](#)

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](#)^[42] 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](#)^[302] as well as the [Cordier-Diagramm](#)^[303] 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:

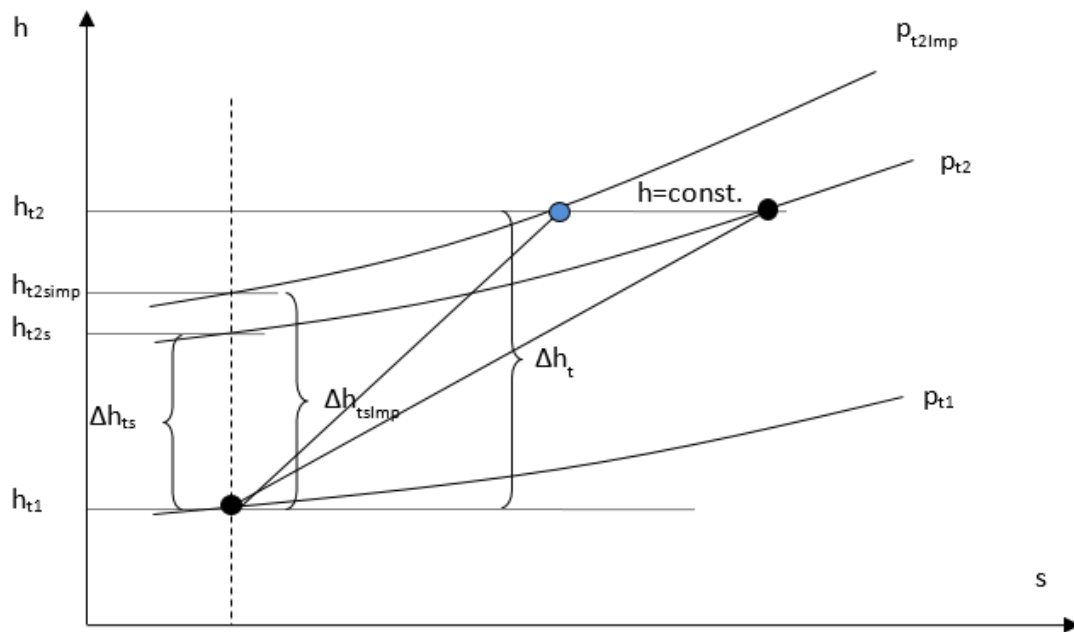
Work coefficient	$\Psi = \frac{Y}{u_2^2/2} = 0.6 \dots 1.5$
Flow coefficient	$\varphi_t = \frac{Q_{ts}}{\frac{\pi}{4} d_2^2 u_2} = 0.01 \dots 0.15$
Meridional flow coefficient	$\varphi_m = \frac{Q_2}{\pi d_2 b_2 u_2} = \frac{c_{m2}}{u_2} = 0.1 \dots 0.5$
Diameter coefficient	$\delta = \frac{\Psi^{1/4}}{\varphi_t^{1/2}} = 1.05 d_2 \left(\frac{Y}{Q_{ts}^2} \right)^{1/4}$
Tangential force coefficient	$c_t = \frac{\Psi}{\eta_{tt} \varphi_m} \approx 3 \dots 6$
Outlet width ratio	$b_2/d_2 = 0.01 \dots 0.15$
Diameter ratio	d_s/d_2
Inlet Mach number	

Outlet Mach number	$\text{Ma}_{c2} = \frac{1}{\sqrt{\left(\frac{a_{t,2}}{c_2}\right)^2 - \frac{\kappa-1}{2}}} \leq 1$ <p>(perfect gas model)</p>
Degree of Reaction	$R = 1 - \frac{c_2^2}{2Y}$
thermodynamic values for - impeller inlet (cross section S) - impeller outlet (cross section 2)	$\rho, p, T, c_m, c_u, w, u, \rho_t, p_t, T_t$

In the impeller outlet (cross section 2) two different total pressure values are given: p_{t2} and $p_{t2\text{Imp}}$.

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](#)^[86]. 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 $p_{t2\text{Imp}}$ to the value at the inlet of the next non-rotating component p_{t2} . In accordance to following diagram these two different pressure values are calculated (assuming perfect gas behavior) by:

$$p_{t2} = p_{t1} \left(\frac{h_{t2s}}{h_{t1}} \right)^{\frac{\kappa}{\kappa-1}} \cdot \eta_{tt+} \cdot p_{t2\text{Imp}} = p_{t1} \left(\frac{h_{t2s\text{Imp}}}{h_{t1}} \right)^{\frac{\kappa}{\kappa-1}} = p_{t1} \left(1 + \frac{\Delta h_{ts}}{h_{t1} \cdot \eta_{tt+}} \right)^{\frac{\kappa}{\kappa-1}}$$

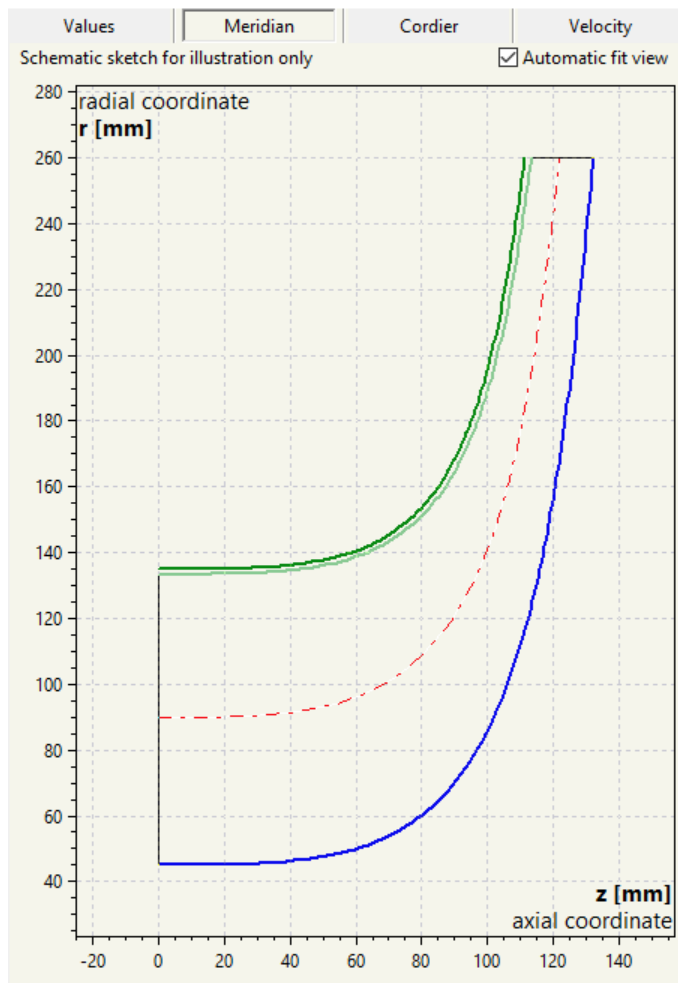


All thermodynamic values are calculated on the basis of the total pressure p_{t2Imp} in cross section 2.

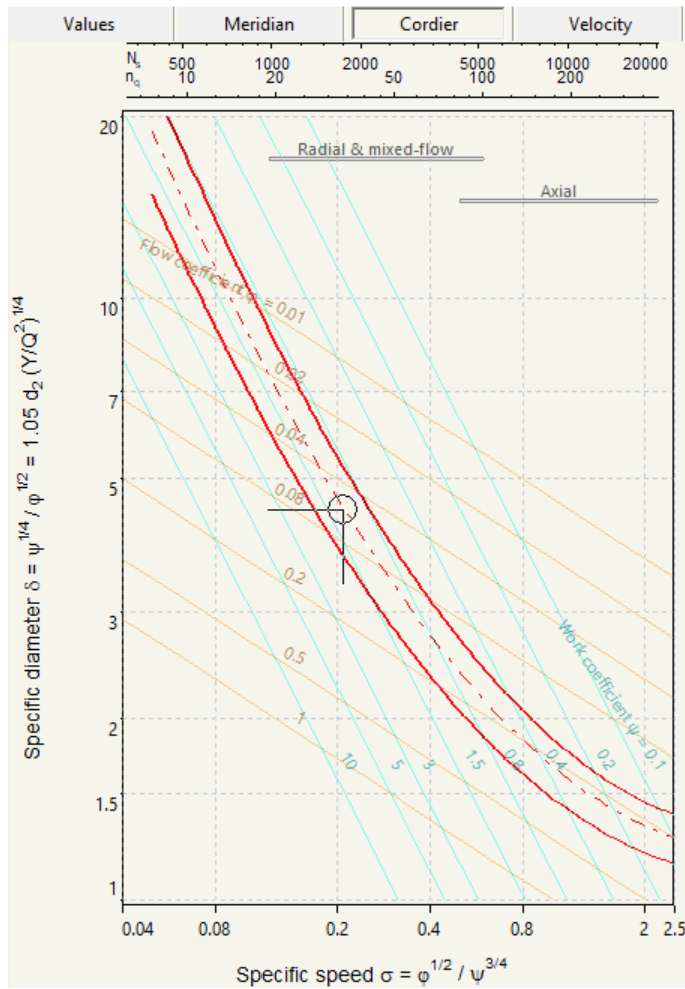
If a stator is located prior the impeller, boundary conditions for the determination of the main dimensions will be calculated on the basis of the thermodynamic state at the outlet of this stator. In case of an undefined thermodynamic state at this location the inlet boundary conditions (i.e. total pressure 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.



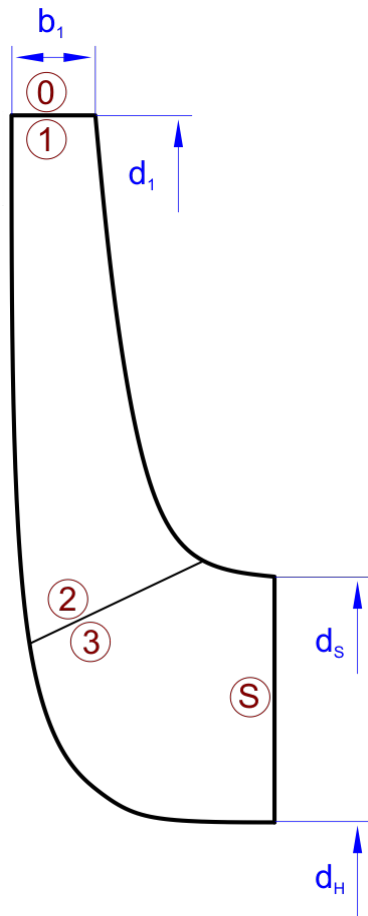
The **Cordier diagram** is based on an intensive empirical analysis of proved turbomachinery using extensive experimental data.



7.1.4 Radial-inflow Turbine

? Rotor | Main dimensions

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.

Details

→ [Setup](#) ³⁰⁶

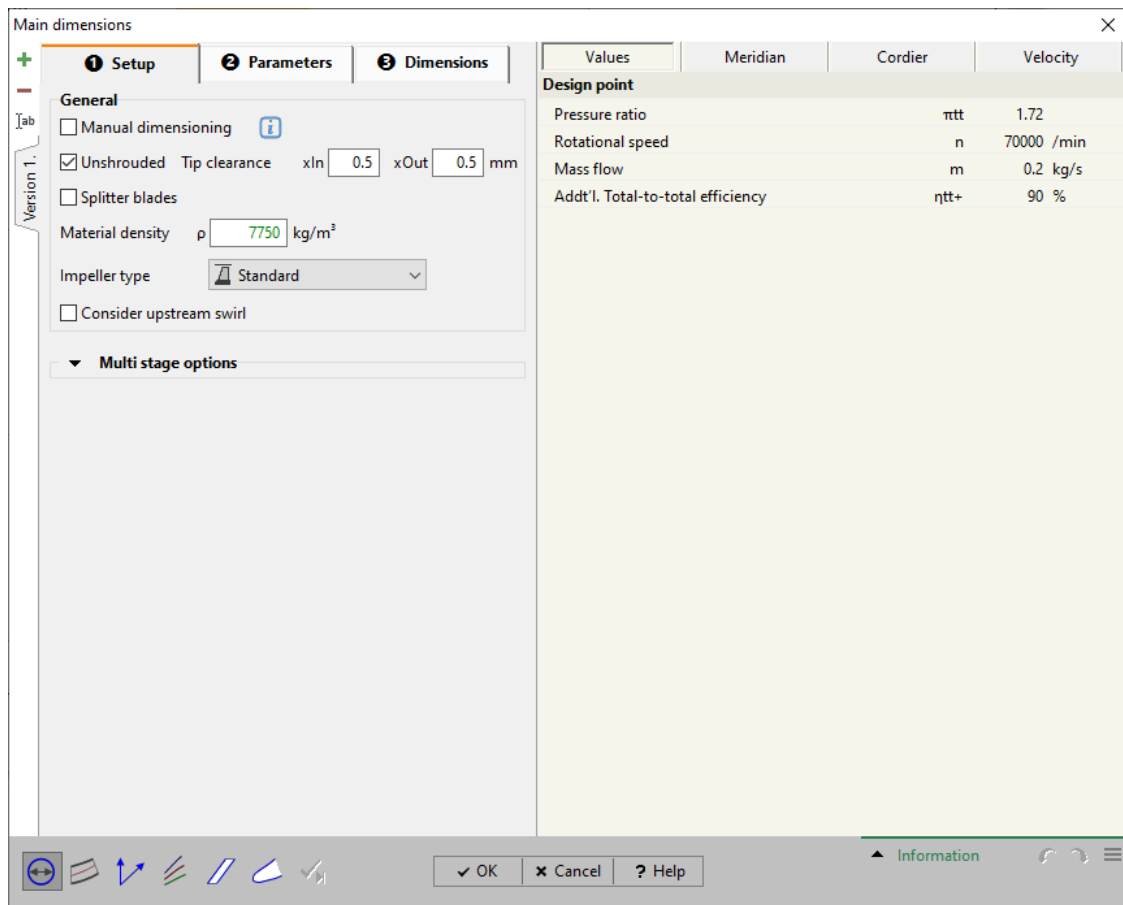
→ [Assumptions](#) ³⁰⁸

→ [Dimensions](#) ³⁰⁸

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](#)³³⁵ by pressing button  next to the input area.

- **Consider upstream swirl**

If this option is chosen the outlet swirl of the upstream component will be used for the determination of the inlet swirl. If not, cu_1 of the actual component will be zero (no inlet swirl).

Initial default setting

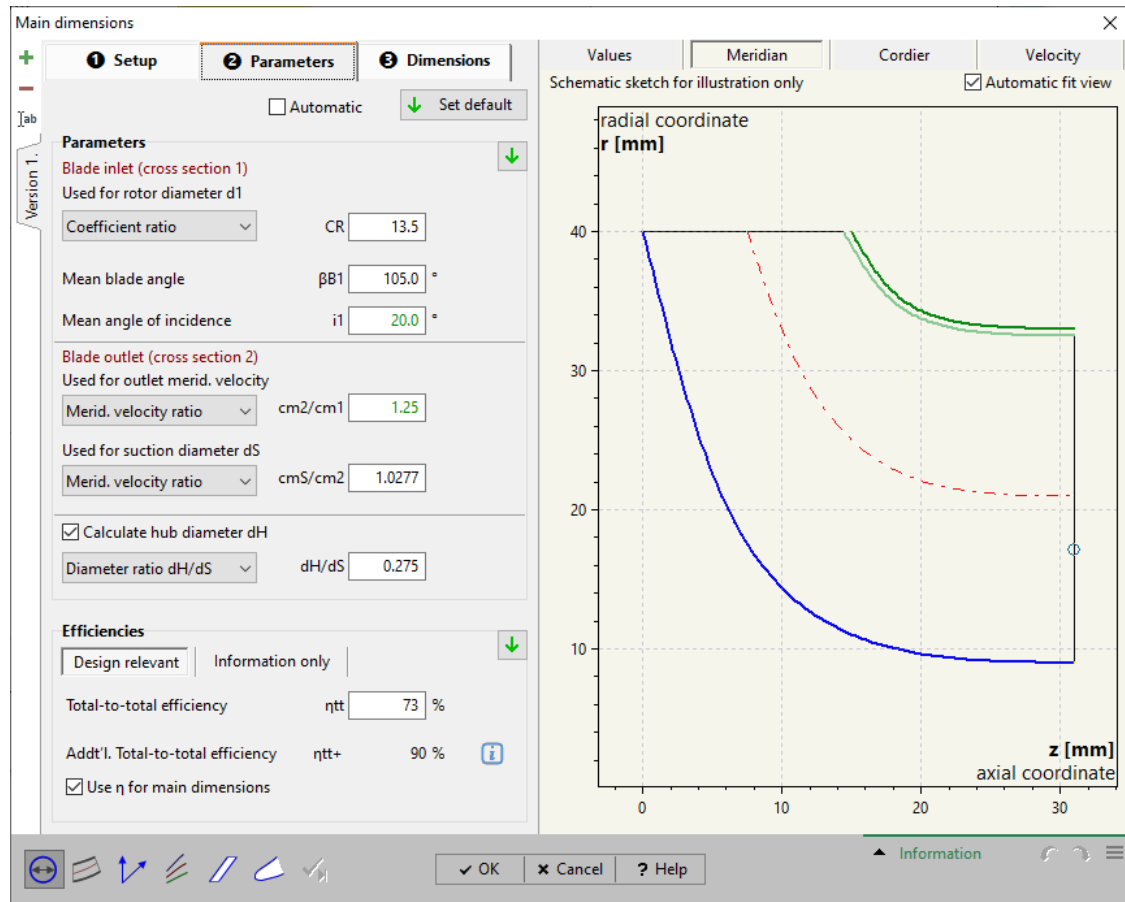
When creating a new design the initial default settings for some important properties are displayed in the panel **Initial default settings**. These settings are used in further design steps and can be modified by selecting the **Change settings** button. Of course these default settings can be modified manually in the appropriate design steps. See [Preferences: Impeller/ Stator settings](#)¹⁹⁶ for more information.

Information

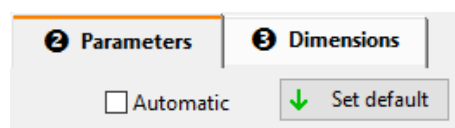
Some design point values are displayed in the right **Information** panel when selecting the page **Values** (see [Global setup](#)⁸⁶).

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](#)^[198]).

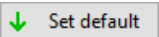


For details of how to handle the parameter edit fields please see [Edit fields with empirical functions](#)^[77].



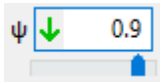
Parameter and efficiency values can be handled manually or can be switched to automatic update by the checkbox on top of the page. Then the default values are used always, even after design point modifications (see [Global setup](#)^[86]).

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

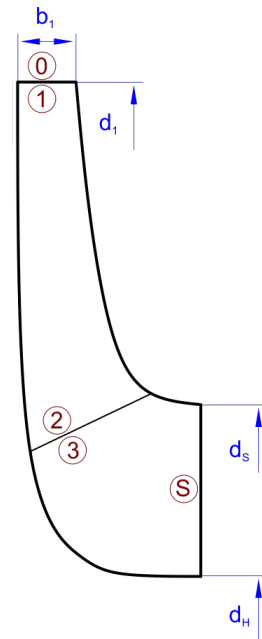



individually by the default button within the input field when selected

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 design concept is based on a mean flow area, therefore a mean blade angle β_1 as well as a mean incidence angle i has to be given. In order to yield best efficiency the angle of incidence should be 20..30°. The relative flow angle α_1 based on these two parameters is:

One of the following parameters has to be specified for the calculation of the rotor diameter d_1 .

Work coefficient (= pressure and head coefficient)	▪ dimensionless expression of the specific enthalpy
---	---

	$\psi = \frac{\Delta h_{is}}{u_1^2/2} \quad \text{and} \quad \psi = \frac{\Delta h}{u_1^2/2}$ <ul style="list-style-type: none"> big \rightarrow small d_1 small \rightarrow big d_1 Guideline ~ 2
Flow coefficient φ_m	<ul style="list-style-type: none"> dimensionless mass flow $\varphi_m = \frac{c_{m1}}{u_1}$ <ul style="list-style-type: none"> in accordance to Cordier-Diagramm ^[318]
Tangential force coefficient $c_t = \quad / \quad m$	<ul style="list-style-type: none"> Coefficient of a flow force pointing in tangential direction <p>3 ... 4 Francis high-speed turbine 4 ... 8 Normal-speed turbine 8 ... 10 Low-speed turbine</p>
Coefficient ratio $c_R = \quad / \quad m^2$	<ul style="list-style-type: none"> Ratio of work to the square of the meridional speed <p>6 ... 10 Francis high-speed turbine 10 ... 12 Normal-speed turbine 12 ... 30 Low-speed turbine</p>

Between the work coefficient ψ , the relative flow angle β_1 and the tangential force coefficient c_t / m there is the following relation:

$$\psi = \frac{1}{\frac{1}{2} + \frac{1}{\psi / \varphi_m} \cot(\beta_1)}$$

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.

For all further geometric variables guess values have to be given:

Diameter ratio d_2/d_1	~ 0.5
--------------------------	------------

Meridional acceleration c_{m2}/c_{m1}	1.005..1.05
Meridional acceleration (suction side) c_{mS}/c_{m2} or Diameter ratio d_S/d_1	1.005..1.05 ~0.7
Diameter ratio d_H/d_S	~0.3

There are three modes for the definition of the hub diameter d_H :

- Direct input in the tab sheet [Dimensions](#)^[313] (check box "Calculate hub diameter" deactivated)
- Combo box option "Diameter ratio d_H/d_S ": automatic calculation with $d_H = d_H/d_S * d_S$.
- Combo box option "Diameter ratio d_2/d_1 ": automatic calculation with $d_H = d_H/d_S * d_S$. Here the diameter ratio d_H/d_S will be adjusted in a way that the guideline of the [geometrical ratios](#)^[317] will be met.

With option "diameter ratio d_S/d_1 " for the d_S -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 η_{tt} (total-total)

Information only

- Mechanical efficiency η_m

Internal and mechanical efficiency form the overall efficiency (coupling efficiency):

$$\eta_{ttSt} = \frac{P_D}{P_Q} = \eta_{tt} \eta_m$$

P_Q : (isentropic) Rotor power
 P_D : Power output (coupling/ driving power)

The rotor efficiency (or blade efficiency) η_{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 \tilde{Y} and the specific work at loss less transmission:

$$\eta_{tt} = \frac{\tilde{Y}}{Y}$$

The mechanical efficiency mainly includes the friction losses in bearings and seals:

$$\eta_m = 1 - \frac{P_m}{P} \approx 0.95 \dots 0.995$$

(rising with impeller size)

If the check box "**Use for main dimensions (otherwise for B1 only)**" is set, then main dimension calculation is done on the basis of $h = h_{is}$. Otherwise h_{is} - the isentropic specific enthalpy - is used.

Information

In the right panel of the tab sheet **Parameter** some variables are displayed for **Information**:

actual Power P_D	$P_D = P_Q \cdot \eta_{ttSt}$
Power loss P_L	$P_L = P_Q - P_D$
Flow Q	calculated with total density in the outlet:
Total pressure inlet p_{t1}	$p_{t1} = \pi p_{t2}$
Pressure ratio total-total	

Pressure ratio total-static	π_{ts}
Stage efficiency total-total	η_{tSt}
Efficiency total-static	η_{ts}
Polytropic efficiency	$\eta_p = \frac{\frac{\kappa}{\kappa-1}}{\frac{n}{n-1}}$ <p>(n .. polytropic exponent .. isentropic exponent)</p>

In general for cost reasons single-stage & single-intake machines are preferred covering a range of about $10 < nq < 400$. In exceptional cases it may become necessary to design a rotor for extremely low specific speed values ($nq < 10$). 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 ? 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 ($nq > 400$) do occur one can reduce rotational speed n or mass flow rate ? if feasible. Another option would be to operate several single-stage turbines - having a lower nq - in parallel.

Please note: CFturbo® is preferably used between $10 < nq < 150$ – radial and mixed-flow rotors.

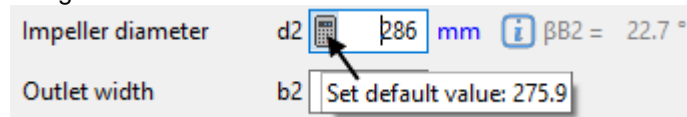
7.1.4.3 Dimensions

In the panel **Shaft**, the required shaft diameter is computed.

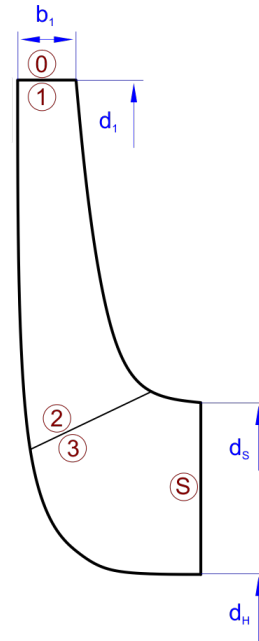
→ [Shaft/ Hub](#) 

The main dimensions of a rotor - suction diameter d_s , hub diameter d_H , rotor diameter d_1 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**.

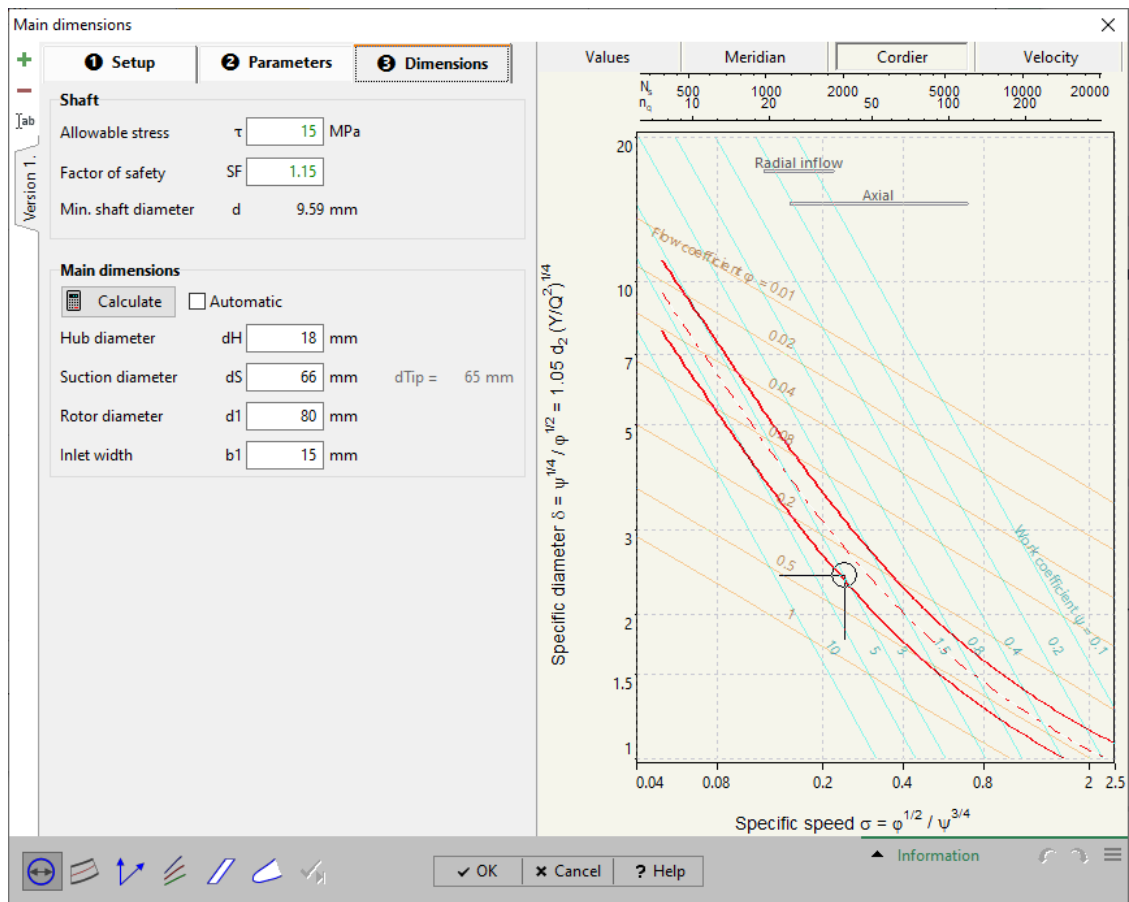
Individual main dimensions can be calculated separately using the button inside the value field.



One may accept the proposed values or can modify them slightly, e.g. to meet a certain normalized diameter.

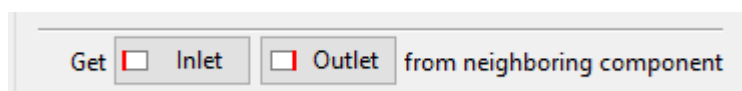


In case the checkbox **Automatic** is activated a new calculation will be accomplished after any change of parameter. Then the manual alteration of the main dimensions is not possible.



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](#)^[42] 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](#)^[317] as well as the [Cordier-Diagramm](#)^[318] 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**:

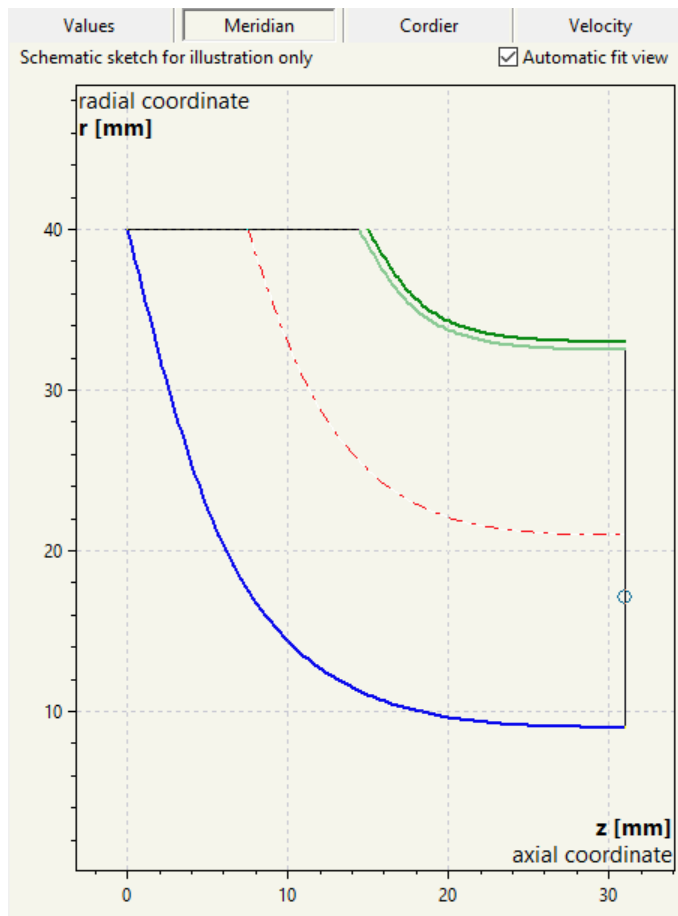
Work coefficient	$\psi = \frac{\Delta h_{ttis} \cdot \eta_{tt}}{u_1^2 / 2}$	
Flow coefficient	$\varphi_t = \frac{Q_{ts}}{\frac{\pi}{4} d_1^2 u_1}$	
Meridional flow coefficient	$\varphi_m = \frac{Q_1}{\pi d_1 b_1 u_1} = \frac{c_{m1}}{u_1}$	
Diameter coefficient	$\delta = 1.054 \cdot d_1 \cdot \frac{\Delta h_{ttis}^{1/4}}{Q_{ts}^{1/2}}$	
Specific speed n_q (different unit definitions: see Preferences ¹⁹²)	$n_q = n \left[\text{min}^{-1} \right] \frac{\sqrt{Q \left[\frac{\text{m}^3}{\text{s}} \right]}}{\left(Y \left[\frac{\text{m}^2}{\text{s}^2} \right] \frac{1}{g} \right)}$	points to machine type and general shape of rotor
Inlet pressure, density and temperature	$p_1, T_1, \rho_1, p_{t1}, T_{t1}, \rho_{t1}$	static and total values
Inlet velocities	c_1, c_{u1}, c_{m1}, w_1	
Peripheral speed at inlet	$u_1 = \sqrt{\frac{\psi}{2 \cdot Y \cdot \eta_{tt}}}$	
Machine-Mach-number	$M_1 = \frac{u_1}{a_1}$	
Blade width at inlet		
Outlet pressure, density and temperature	$p_2, T_2, \rho_2, p_{t2}, T_{t2}, \rho_{t2}$	static and total values
Outlet velocities	c_2, c_{u2}, c_{m2}, w_2	
Peripheral speed at outlet		

Outlet Ma-Number	$M_2 = \frac{c_2}{a_2}$	
Mean diameter at outlet	$d_2 = \left(\frac{d_2}{d_1} \right) d_1$	
Width at outlet	$b_2 = 1.025 \frac{\dot{m}}{\pi d_2 \cdot c_{m2} \cdot \rho}$	
Ratio Width-diameter at inlet	b_1/d_1	guideline: 0.05..0.15
Diameter ratio	d_2/d_{2min} with: $d_{2min} = \sqrt{\frac{1}{2} (d_s^2 - d_N^2)}$	guideline: 1.005..1.05
Ratio radius-width at outlet	$\frac{(r_s - r_N)}{b_2} = \frac{(d_s - d_N)}{2 \cdot b_2}$	guideline: 1.005..1.05
Isentropic velocity ratio	$v_{ts} = \frac{u_1}{\sqrt{2\Delta h_{is}}}$	

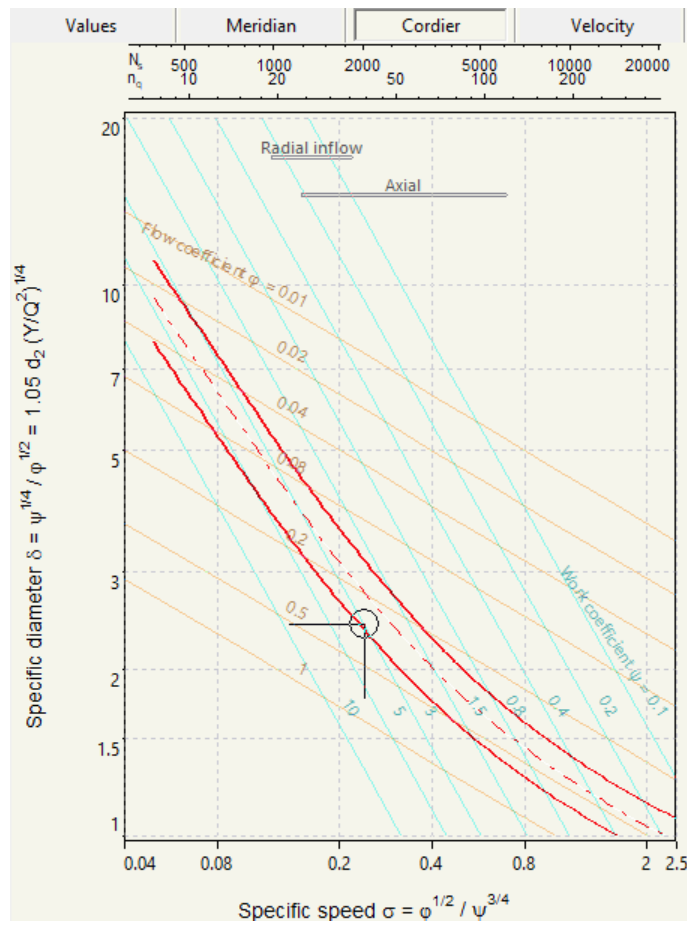
The guidelines given in the last column of the last three rows, should be matched within the design.

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.

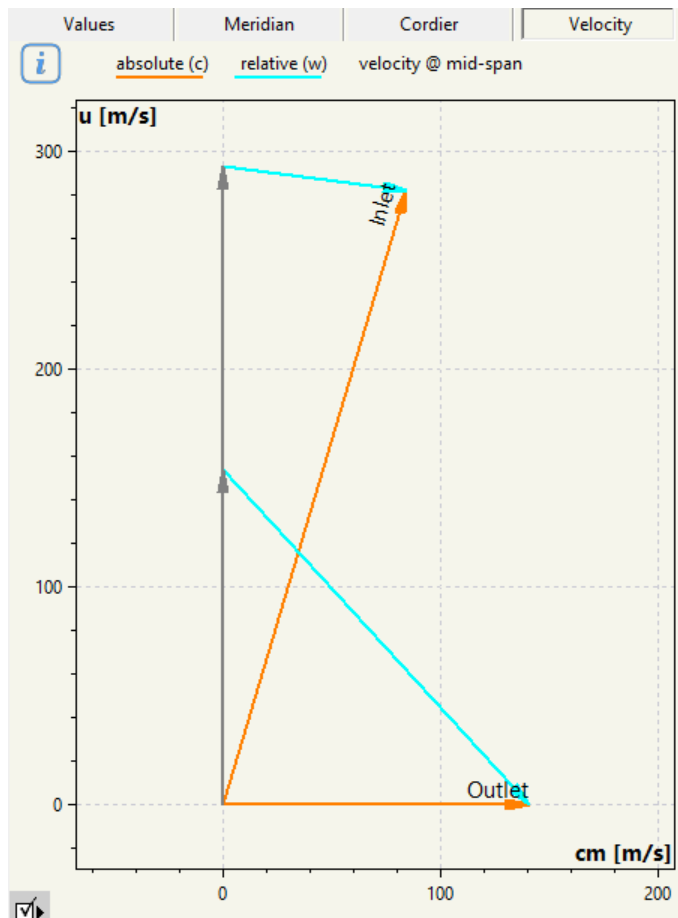
The Meridional preview is based on the main dimensions designed until this point.



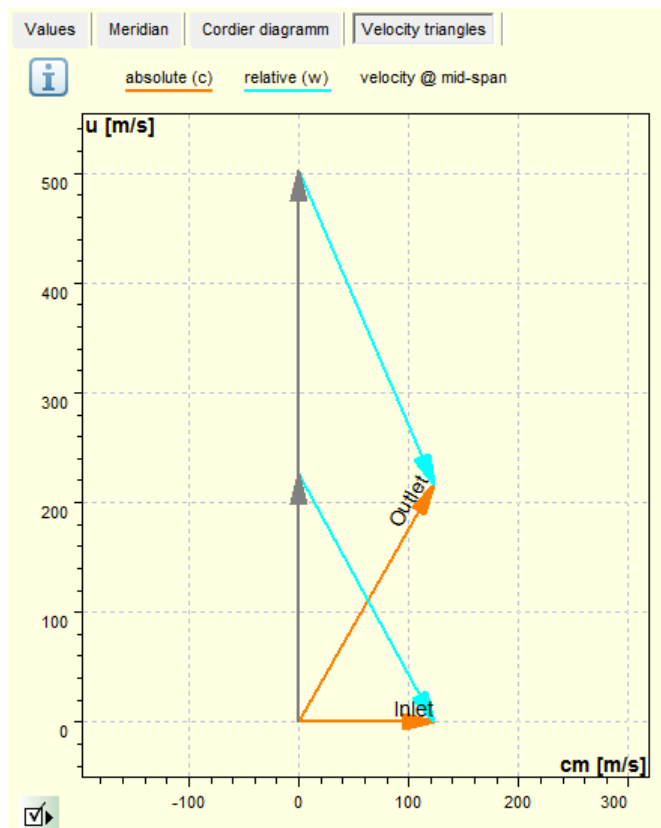
The **Cordier diagram** is based on an intensive empirical analysis of proved turbomachinery using extensive experimental data.



The **Velocity triangles** are the result of a mid-span calculation and are based on the [design point](#)^[86] and the main dimensions.



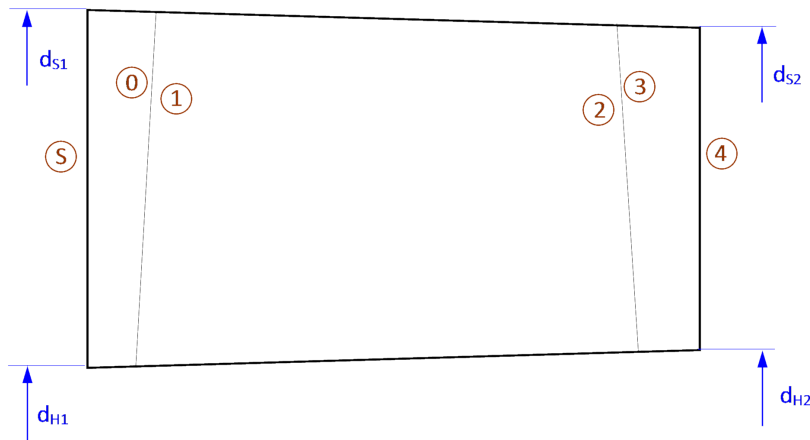
The **Velocity triangles** are the result of a mid-span calculation and are based on the [design point](#)^[86] and the main dimensions.



7.1.5 Axial Turbine

? 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 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: 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](#) ³²³

→ [Parameters](#) ³²⁵

→ [Dimensions](#) ³²⁹

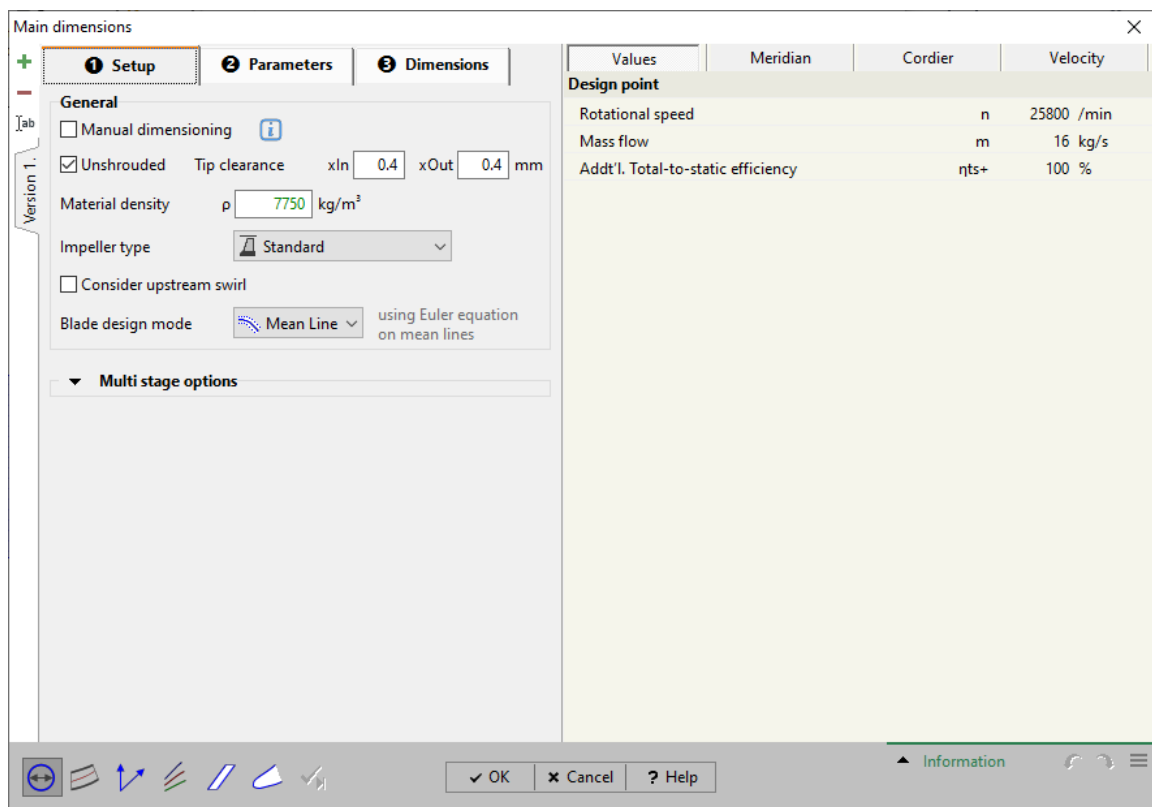
Possible warnings

Problem	Possible solution
Inlet boundary conditions do not fit previous component.	
If a stator is located prior the 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.	Adjust the stator geometry (dimensions or blade angles) or change the Global setup ⁸⁶ (e.g. decrease mass flow).

Problem	Possible solution
Inlet static pressure is smaller than the outlet static pressure.	
If a vaned stator is located prior the rotor, its blade angles might yield a very high circumferential velocity and therefore a comparable small static pressure. In case this pressure, which is the inlet static pressure of the rotor, is smaller than the estimated static outlet pressure, an reasonable expansion cannot be established.	Adjust the precursor stator trailing edge blade angles and decrease herewith the circumferential velocity component.

7.1.5.1 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) 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](#)^[335] by pressing button  next to the input area.

- **Impeller type**

Select either **Standard** or **Rocket engine** rotor type.

- **Consider upstream swirl**

If this option is chosen the outlet swirl of the upstream component will be used for the determination of the inlet swirl. If not, cu_1 of the actual component will be zero (no inlet swirl).

- **Blade design mode**

Currently design mode [Mean line](#)^[371] is available that is using Euler's equation on mean lines.

Multi stage

For a multi stage design the panel [Multi stage options](#)^[336] is available.

Initial default setting

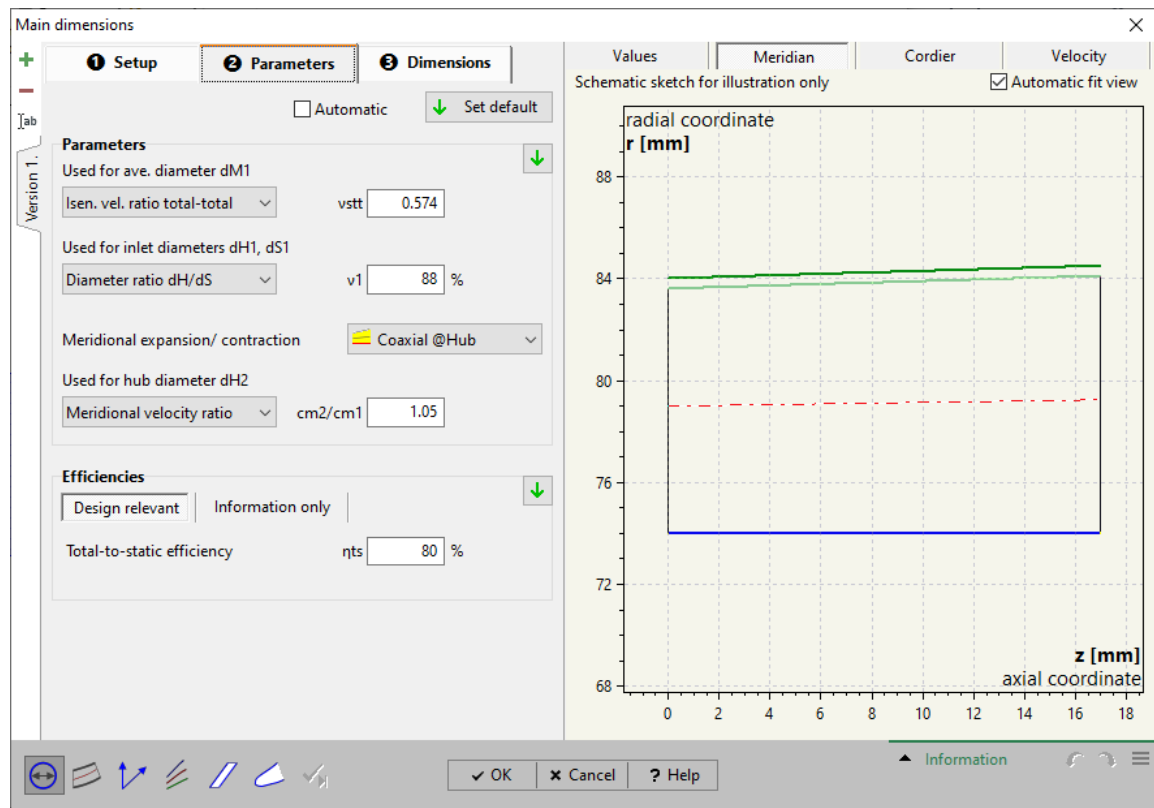
When creating a new design the initial default settings for some important properties are displayed in the panel **Initial default settings**. These settings are used in further design steps and can be modified by selecting the **Change settings** button. Of course these default settings can be modified manually in the appropriate design steps. See [Preferences: Impeller/ Stator settings](#)^[196] for more information.

Information

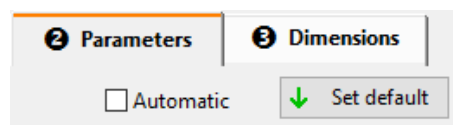
Some design point values are displayed in the right **Information** panel when selecting the page **Values** (see [Global setup](#)^[86]).

7.1.5.2 Parameters

On page **Parameters** one has to put in or to modify parameters resulting from approximation functions in dependence on specific speed nq (see [Approximation functions](#)^[198]).

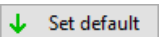


For details of how to handle the parameter edit fields please see [Edit fields with empirical functions](#)^[77].

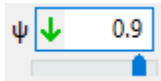


Parameter and efficiency values can be handled manually or can be switched to automatic update by the checkbox on top of the page. Then the default values are used always, even after design point modifications (see [Global setup](#)^[86]).

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

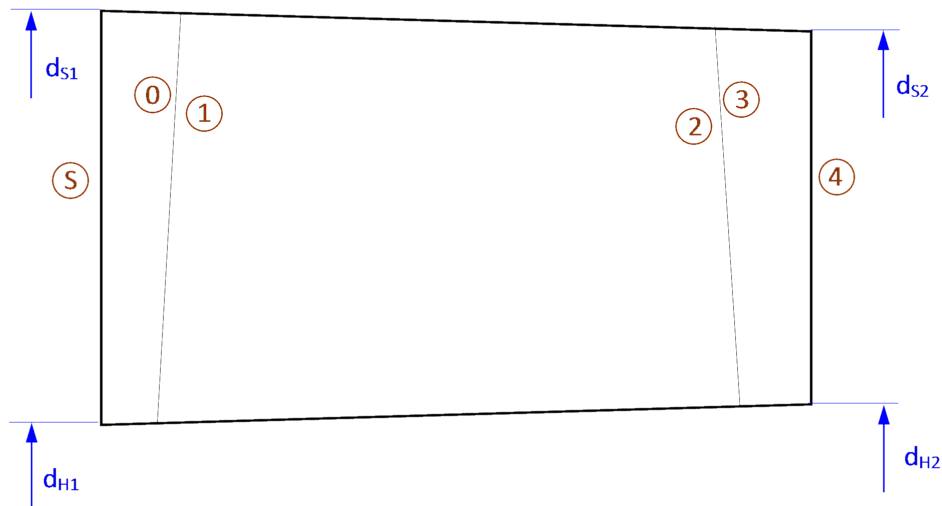


individually by the default button within the input field when selected

Parameters

The panel **Parameters** allows defining alternative parameters in each case for the calculation of the following impeller diameters:

inlet	outlet
d_{S1} , d_{H1}	d_{S2} , d_{H2}







With the help of the following parameters the inlet of the rotor can be calculated. If the Diameter ratio is chosen the inlet is determined independent of the upstream swirl of an upstream component. If Degree of reaction is chosen, the upstream swirl is either determined by the upstream component (check box "**Consider upstream swirl**" checked, see [setup](#) ³²³) or by the absolute inlet flow angle

1°

Isentropic velocity ratio $_{is}$	Mean inlet diameter $0.5(d_{S1} + d_{H1})$
-----------------------------------	--

Degree of reaction R	Inlet diameter hub and tip, d_{H1} & d_{S1} $R = \frac{\Delta h_{ts}}{\Delta h_{tsis}} = 1 - \frac{c_1^2}{2 \cdot \Delta h_{tsis}}$
Absolute inlet flow angle α_1	Inlet diameter hub and tip, d_{H1} & d_{S1}
Diameter ratio d_H/d_S	Inlet hub diameter d_{H1} $d_{H1} = \frac{d_H}{d_S} d_{S1}$

The outlet section can be calculated with:

Meridional velocity ratio c_{m2}/c_{m1}	<p>0.9..1.1</p> <p> strictly coaxial $d_{H2} = d_{H1}$ and $d_{S2} = d_{S1}$</p> <p> coaxial @Hub $d_{H2} = d_{H1}$</p> <p> coaxial @Mid-span $d_{M2} = d_{M1}$</p> <p> coaxial @Shroud $d_{S2} = d_{S1}$</p>
--	--

Efficiency

In the group **Efficiency** the following efficiencies need to be given:

Design relevant

- Rotor efficiency η_{ts} (total-static)

$$\eta_{ts} = \frac{\Delta h_{tt}}{\Delta h_{tsis}}$$

Information only

- Mechanical efficiency η_m

Internal and mechanical efficiency form the overall efficiency (coupling efficiency):

P_Q : (isentropic) Rotor power

P_D : Power output (coupling/ driving power)

The rotor efficiency (or blade efficiency) η_{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:

$$\eta_{tt} = \frac{\Delta h_{tt}}{\Delta h_{ttis}}$$

The mechanical efficiency mainly includes the friction losses in bearings and seals:

$$\eta_m = 1 - \frac{P_m}{P} \approx 0.95 \dots 0.995,$$

rising with impeller size.

Information

In the right panel of the tab sheet **Parameter** some variables are displayed for **Information**:

actual Power P_D	$P_D = P_Q \cdot \eta_{ttSt}$
Power loss P_L	$P_L = P_Q - P_D$
Flow Q_t	calculated with total density in the outlet: $Q_t = \frac{\dot{m}}{\rho_{t2}}$
Pressure ratio total-total	π_{tt}
Pressure ratio total-static	
Efficiency total-total	η_{tt}
Efficiency total-static	η_{ts}

Polytropic efficiency	$\eta_p = \frac{\frac{\kappa}{\kappa-1}}{\frac{n}{n-1}}$ <p>(n .. polytropic exponent .. isentropic exponent)</p>
-----------------------	---

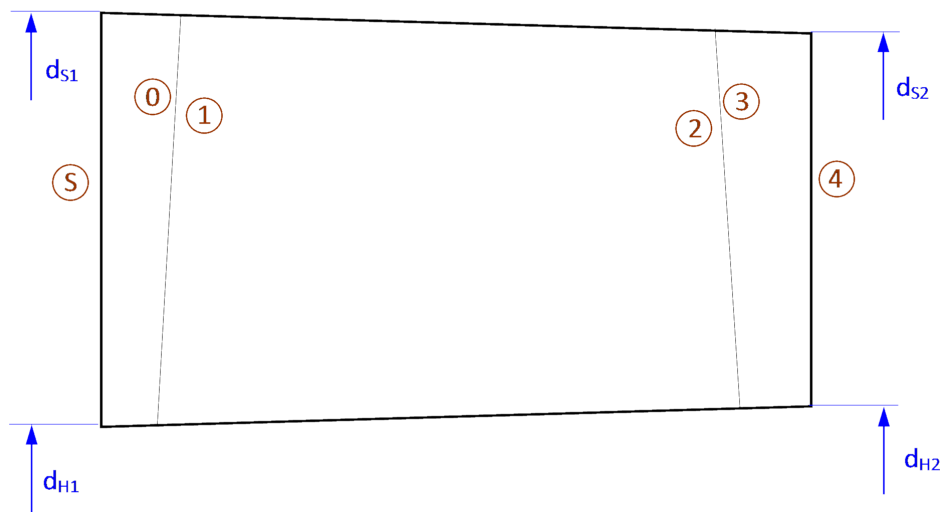
In general for cost reasons single-stage & single-intake machines are preferred covering a range of about $10 < nq < 400$. If especially high specific speed values ($nq > 400$) do occur one can reduce rotational speed n or mass flow rate \dot{m} if feasible. Another option would be to operate several single-stage turbines - having a lower nq - in parallel.

7.1.5.3 Dimensions

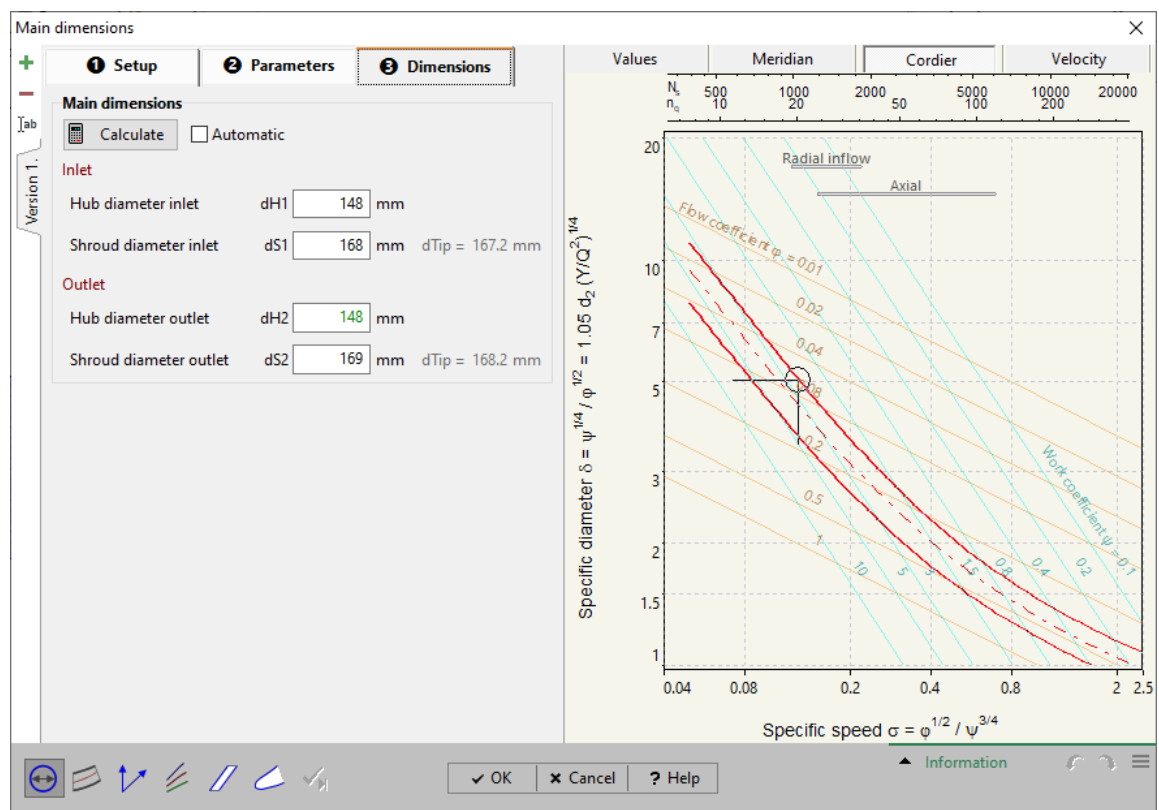
The main dimensions of a rotor - inlet diameter d_{S1} and d_{H1} and outlet diameter d_{S2} and d_{H2} - can be seen on **Main dimensions** panel. They can be recomputed by pressing the **Calculate**-button. The computation is based on "Euler's Equation of Turbomachinery", on the continuity equation 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.

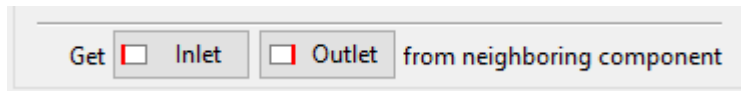


In case the checkbox **Automatic** is activated a new calculation will be accomplished after any change of parameter. Then the manual alteration of the main dimensions is not possible.



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](#)^[42] components or side-by-side impellers.

Information

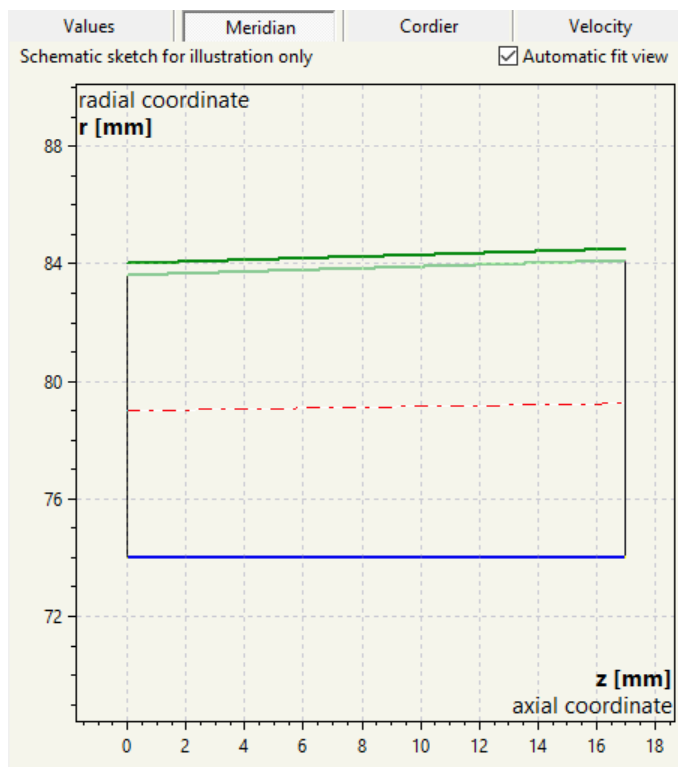
In the right panel of any tab sheet an information panel is situated, which holds the computed variables in accordance to the actual state of design, the resulting [Meridional section](#)^[317] as well as the [Cordier-Diagramm](#)^[318] with the location of the best point. These three sections can be 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**:

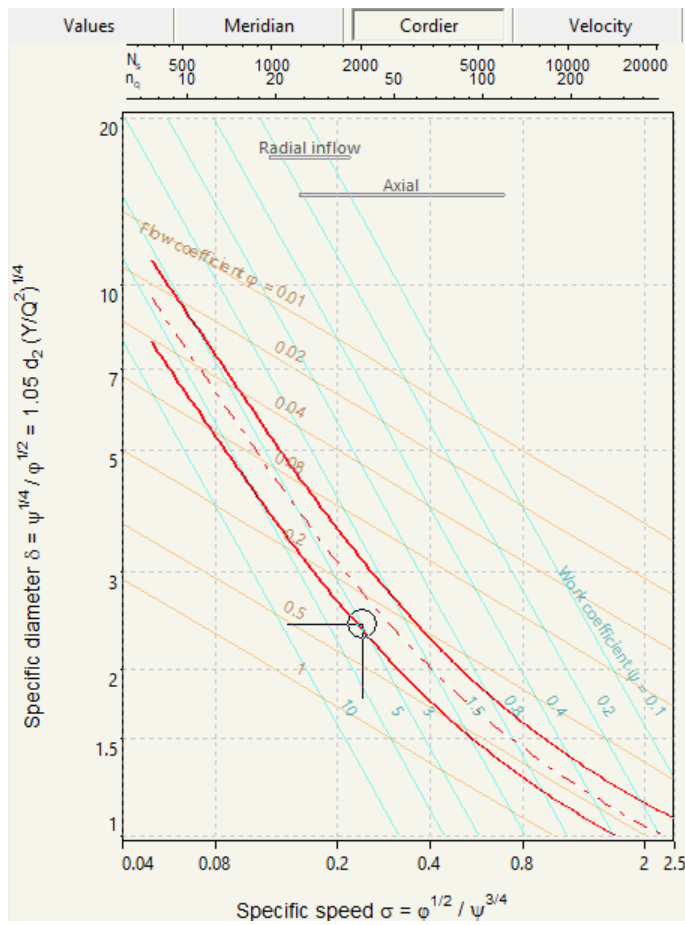
Work coefficient	$\psi = \frac{\Delta h_{ttis} \cdot \eta_{tt}}{u_1^2 / 2}$	
Flow coefficient	$\varphi_t = \frac{Q_{t1}}{\frac{\pi}{4} d_1^2 u_1}$	
Meridional flow coefficient	$\varphi_m = \frac{Q_1}{\frac{\pi}{4} (d_{1S}^2 - d_{1H}^2) u_1} =$	
Diameter coefficient	$\delta = 1.054 \cdot d_{S1} \cdot \frac{\Delta h_{ttis}^{1/4}}{Q_{t1}^{1/2}}$	
Inlet pressure, density and temperature	$p_1, T_1, \rho_1, p_{t1}, T_{t1}, \rho_{t1}$	static and total values
Inlet velocities	$c_1, c_{u1}, c_{m1}, w_1, u_1$	
Inlet Mach-number		

Outlet pressure, density and temperature	$p_2, T_2, \rho_2, p_{t2}, T_{t2}, \rho_{t2}$	static and total values
Outlet velocities	$c_2, c_{u2}, c_{m2}, w_2, u_2$	
Outlet Ma-Number	$M_2 = \frac{c_2}{a_2}$	
Isentropic velocity ratio	$v_{ts} = \frac{u_1}{\sqrt{2\Delta h_{ttis}}}$	

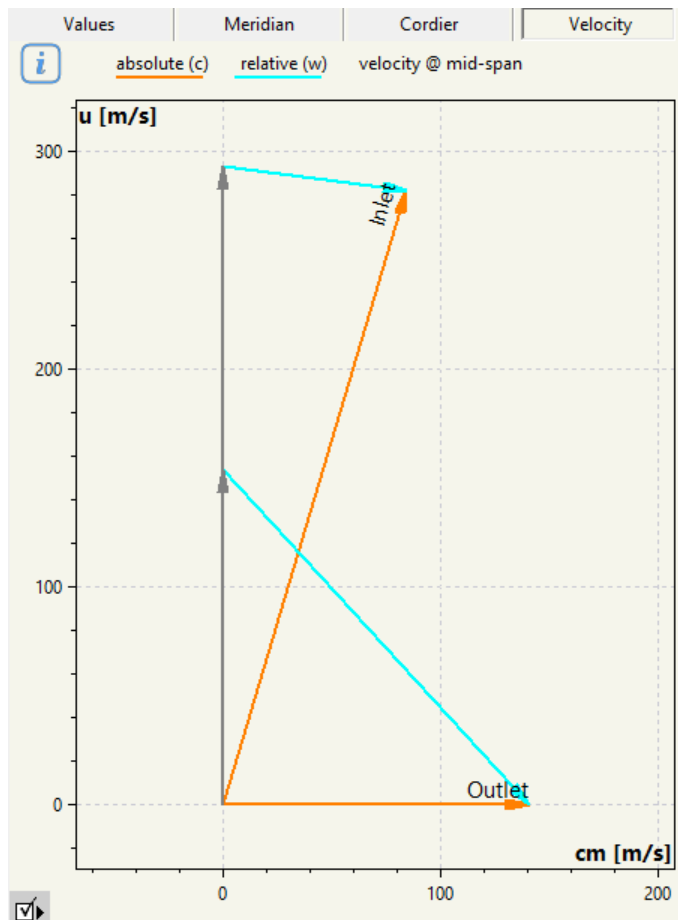
The Meridional preview is based on the main dimensions designed until this point.



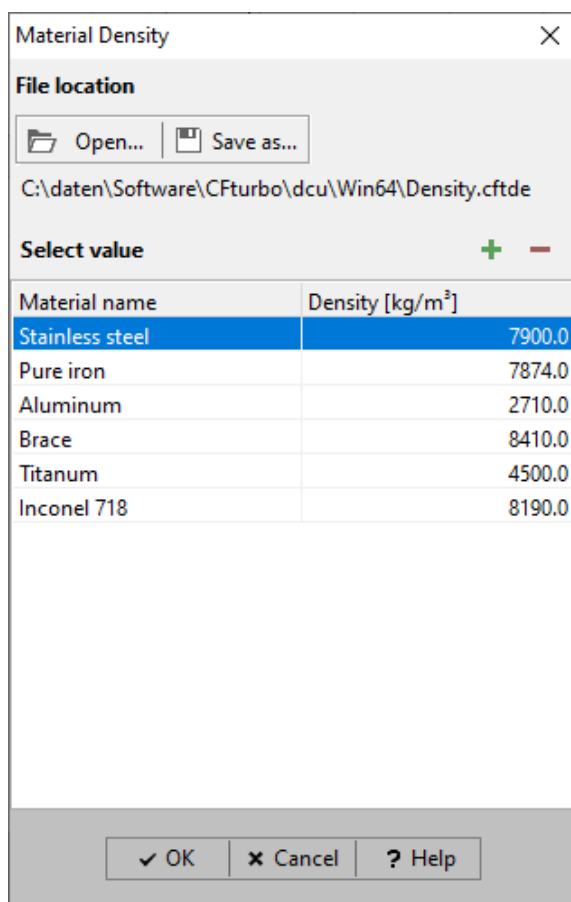
The **Cordier diagram** is based on an intensive empirical analysis of proved turbomachinery using extensive experimental data.



The **Velocity triangles** are the result of a mid-span calculation and are based on the [design point](#)^[86] and the main dimensions.



7.1.6 Material density

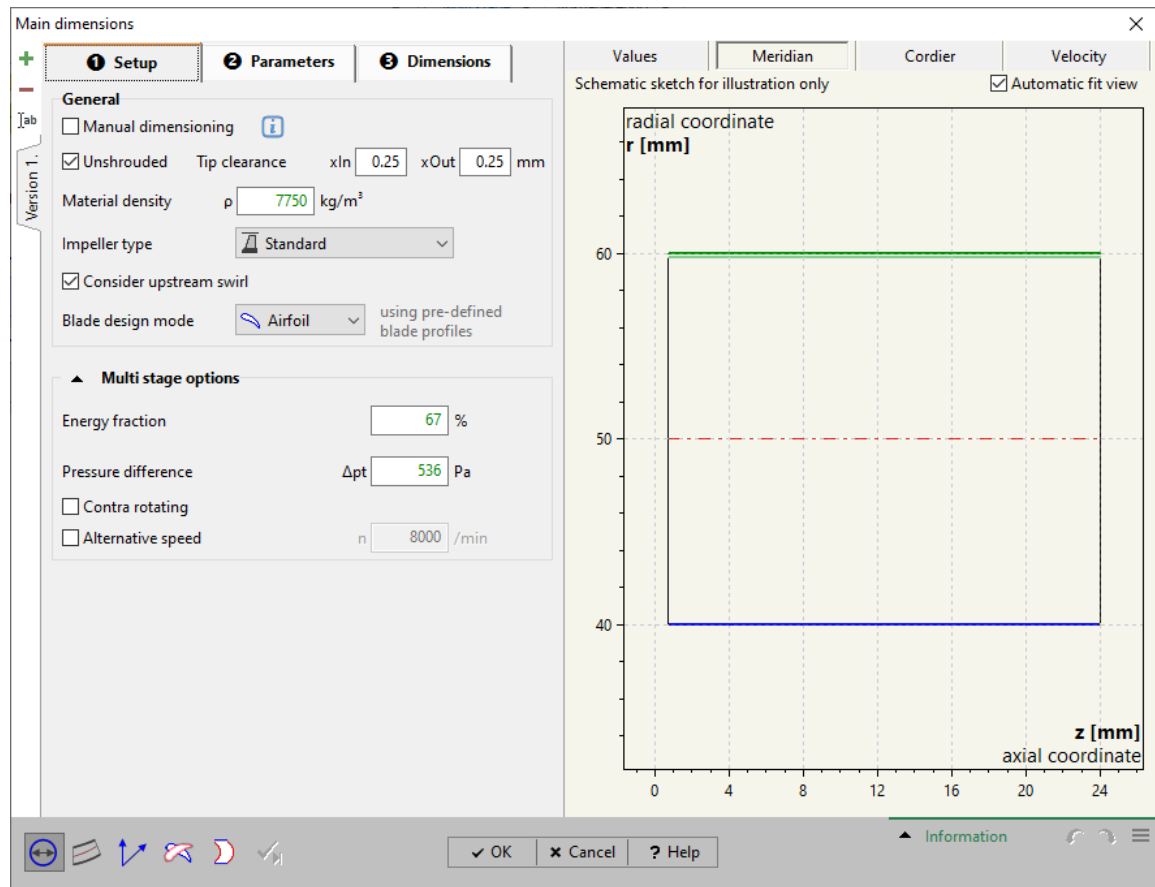


In the table a material with its density can be selected. The list can be extended or reduced by **+** and **-** button respectively. A confirmation of the selected value is done by pressing the **OK**-button.

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.

7.1.7 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**.



[Pump, Ventilator, Compressor]

The [design point](#)^[86] (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:

$$,$$

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.

[Axial turbine]

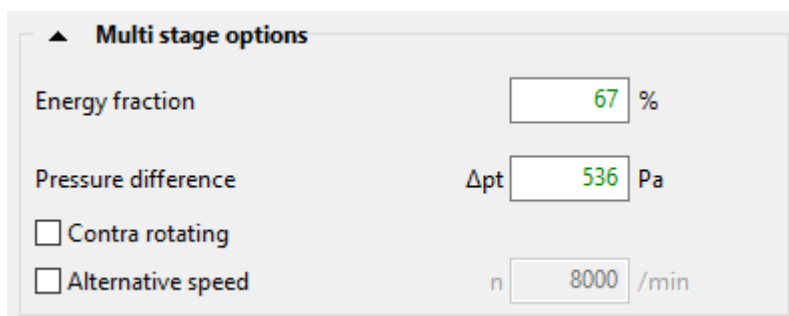
In case more than 1 rotor is contained in the project the [design point](#)^[86] (Power output, pressure ratio) can be distributed amongst the rotors using the **power splitting**. The energy goal used for the design of the selected rotor (index i) is determined by:

$$P_i = e_i \cdot P_{\text{Global}}$$

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 has been specified in the [Global setup](#)^[86] 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



Multi stage options

Energy fraction %

Pressure difference Δp_t Pa


☐ Contra rotating

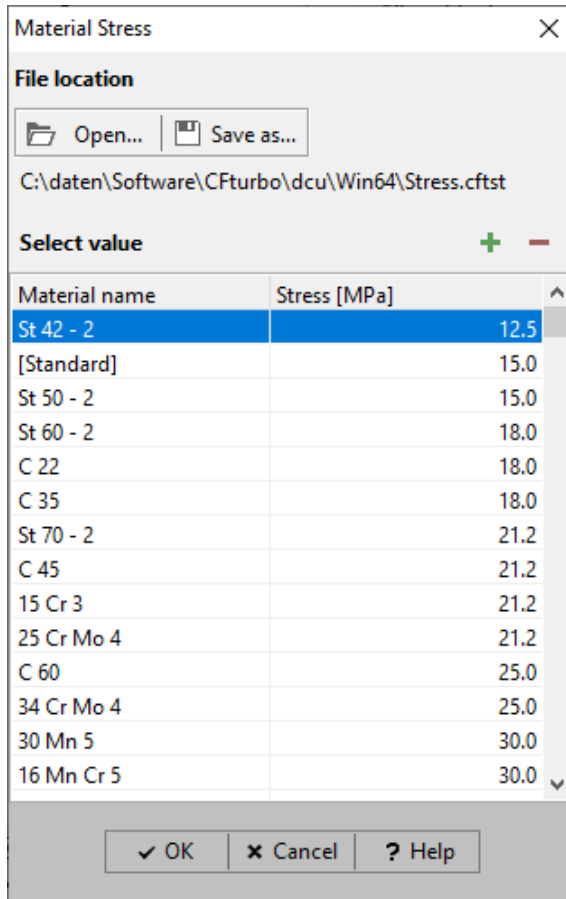
☐ Alternative speed n /min

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](#)^[86]. 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.

7.1.8 Shaft/Hub

Dimensioning of the shaft diameter is made under application of strength requirements. It is a result of torque $M=P/\omega$ to be transmitted by the shaft and the allowable torsional stress τ of the material.

You can directly enter allowable stress or select the value from a list by pressing button  right beside the input area.



In a small dialog window you can see some materials and its allowable stress. The list can be extended or reduced by **+** and **-** 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.cfst** 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.

$$d \geq \sqrt[3]{\frac{8\rho QY \cdot SF}{\pi^2 n \tau \eta}}$$

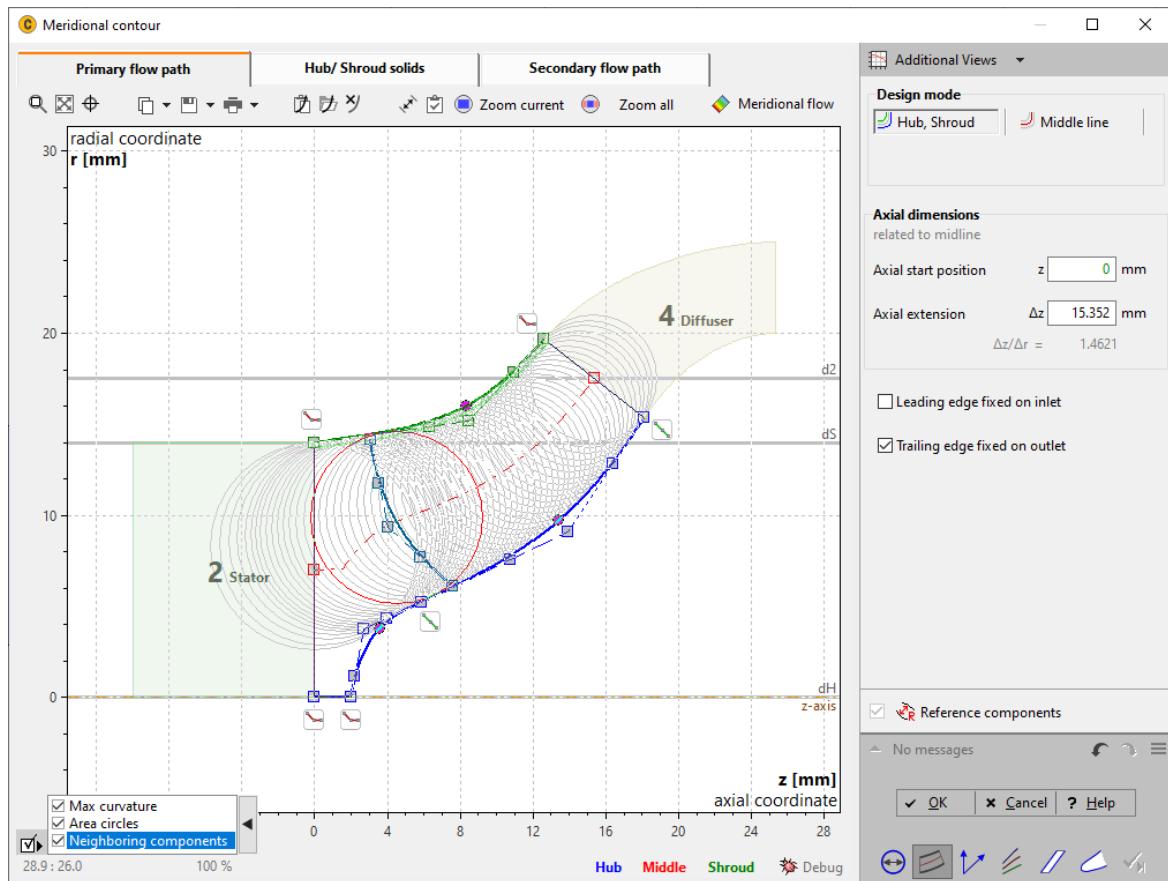
The hub diameter d_H is usually selected as small as possible and depends on the kind of connection of hub and shaft.

7.2 Meridional contour

? IMPELLER | 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](#)^[341]
This contains the design of the primary flow path. Necessary for the following design steps.
- [Hub/ Shroud solids](#)^[360] (optionally)
This contains the design of hub and/or shroud solids. This is an optional part focusing on stress analysis.
- [Secondary flow path](#)^[367] (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

Problem	Possible solution
Inlet hub diameter: deviations between Meridional Contour and Main Dimensions are larger than 0.1%	
The difference between the hub diameter and the corresponding geometric size in the meridian is too large. This is possible for imported polylines only.	Adjust either the main dimensions ^[244] or the imported curve.
Inlet shroud diameter: deviations between Meridional contours and main dimension are larger than 0.1%	
The difference between the suction diameter and the corresponding geometric size in the meridian is too large. This is possible for imported polylines only.	Adjust either the main dimensions ^[244] or the imported curve.
Outlet diameter: deviations between Meridional Contour and Main Dimensions are larger than 0.1%	
The difference between the impeller diameter and the corresponding geometric size in the meridian is too large. This is possible for imported polylines only.	Adjust either the main dimensions ^[244] or the imported curve.
Outlet width: deviations between Meridional Contour and Main Dimensions are larger than 0.1%	
The difference between the outlet width and the corresponding geometric size in the meridian is too large. This is possible for imported polylines only.	Adjust either the main dimensions ^[244] or the imported curve.
Hub/ Shroud contour has discontinuities inside blade region.	
The hub resp. shroud contour is divided into sub-curves who are not connected smoothly in blade region.	Adjust hub resp. shroud contour and apply smoothness at connectors who are inside blade region.
Angle between hub/ shroud contour and inlet/ outlet is not recommended.	
The current angle between hub/shroud contour and inlet/outlet can cause problems in Model finishing ^[487] .	<p>Manipulate hub/shroud contour or move inlet/outlet to change the current angle to inlet/outlet.</p> <p>Contour may contain extremely small and unnecessary parts which should be removed.</p>
Hub contour intermittently touching z-axis (r=0) is not supported.	

Problem	Possible solution
The hub curve is touching the z-axis internally. Before and behind it the radius is greater than 0 creating a complete constriction.	Avoid hub regions at $r = 0$ internally or split geometry into different components.
Meridional contour has an invalid topology. (inside out)	
The sense of circulation of the closed wire containing the inlet, shroud, outlet and hub curve in this specific order is counter-clockwise.	Manipulate meridional curves to guarantee a clockwise sense of circulation or adjust inlet and outlet in main dimensions.

7.2.1 Primary flow path

Content of this section is split into these topics:

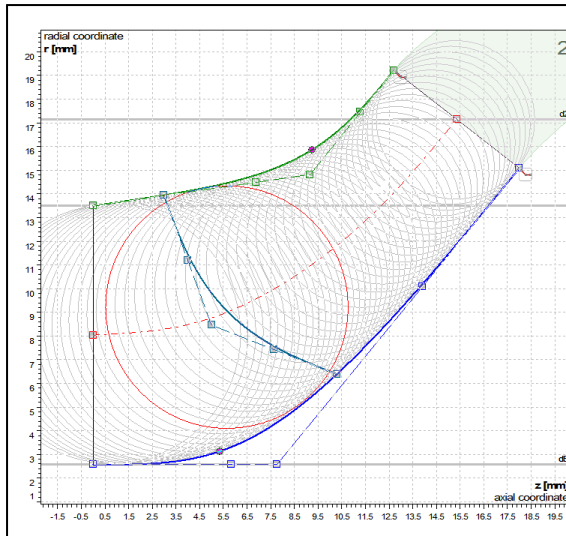
- [Hub, Shroud](#) ³⁴³
- [Leading, Trailing edge](#) ³⁵⁴
- [Meridional flow calculation](#) ³⁵⁶

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.

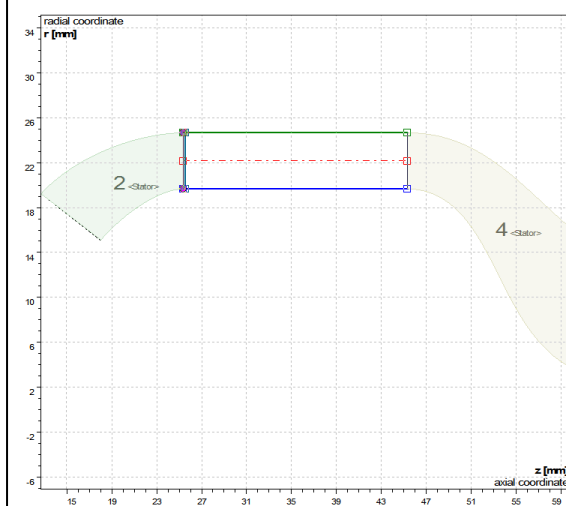
Display Options

In the **Display Options** panel some graphical representations can be activated for illustration:



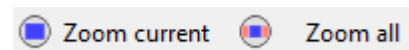
Area circles

used for calculation of cross section area

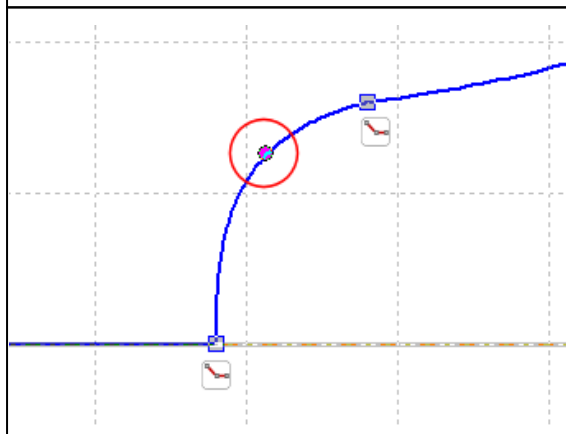


Neighboring components

on inlet and outlet side are displayed for information.



Use the buttons above the diagram to zoom the current meridional shape only or the entire geometry.



Max. curvature

point is displayed on hub and shroud curves

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 radial 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](#) ³⁴⁵

- **Circular Arc + Straight line**

The curve consists of a circular arc and a straight line.

→ [Details](#) ³⁵¹

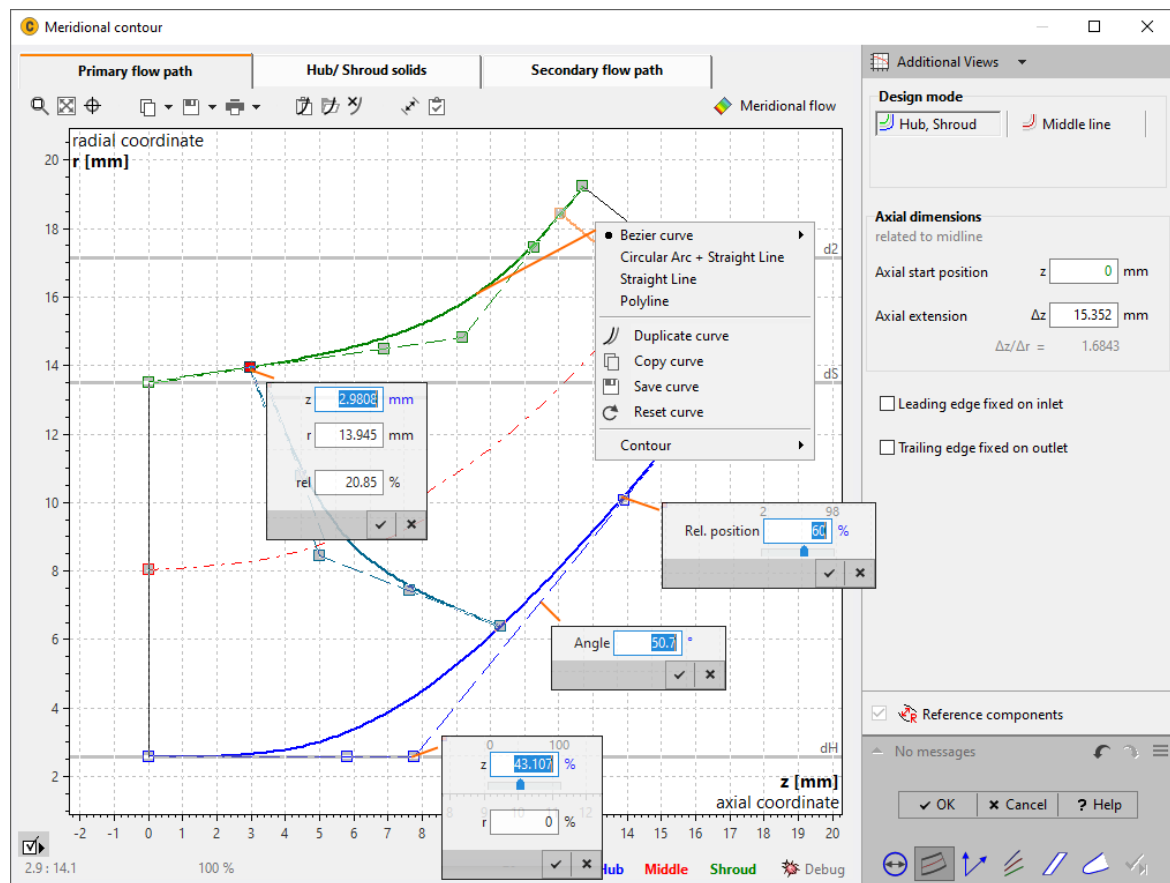
- **Straight line**

The contour is defined by a straight line between start and endpoint.

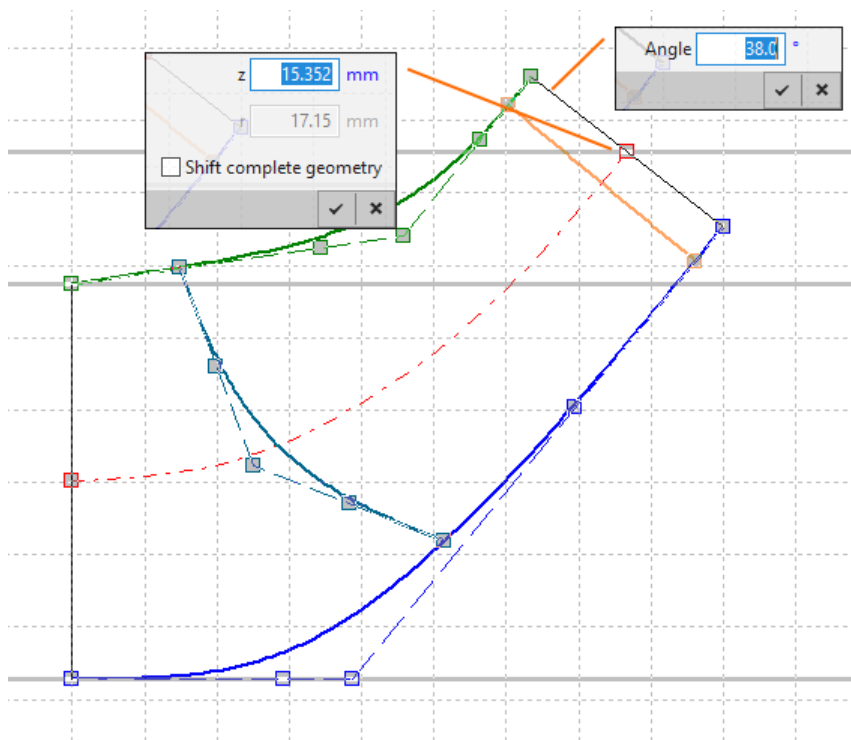
- **Polyline**

The curve is fixed and cannot be modified interactively. Import of point sets from file is possible (**Load polyline**).

Radial ventilator 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.

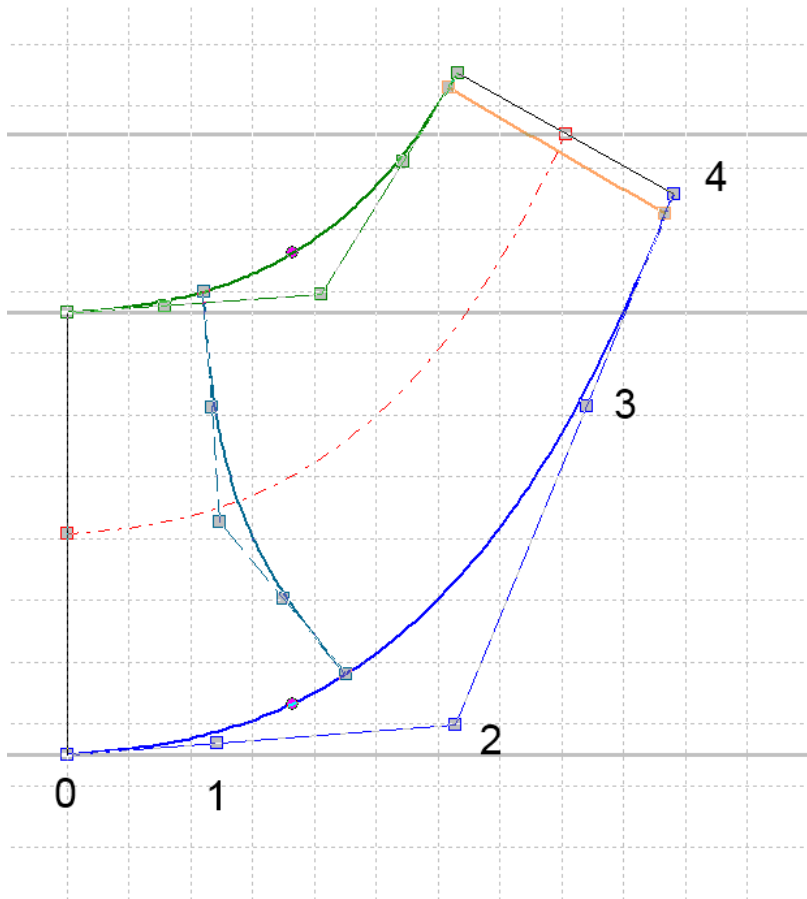


7.2.1.1.1 Bezier

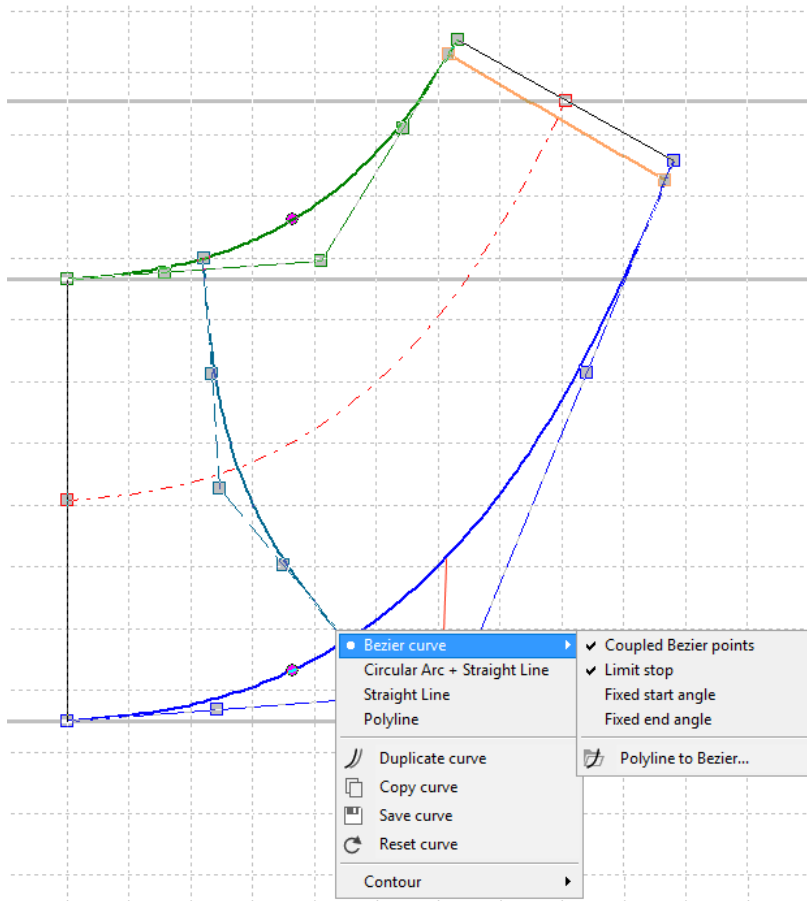
Bezier curves

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.

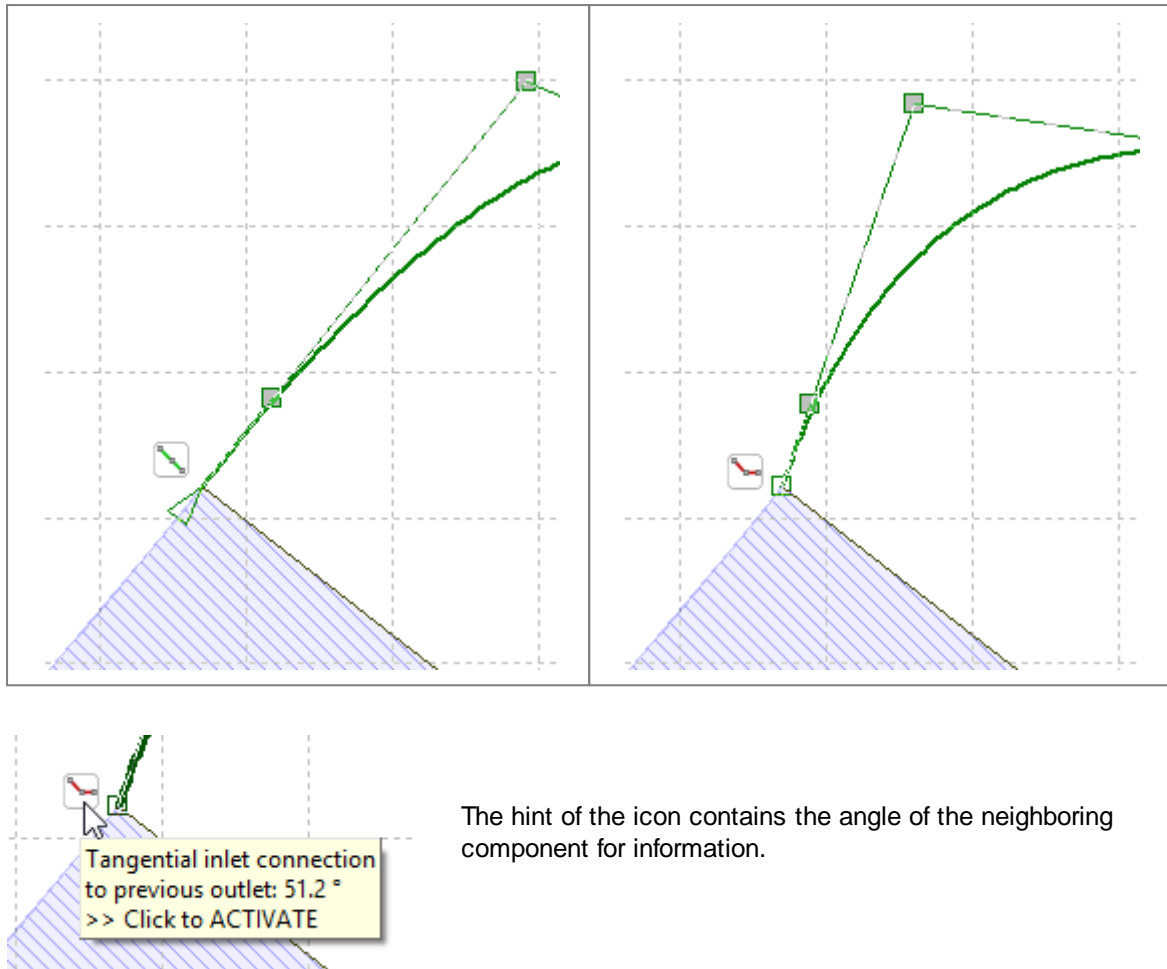
The above mentioned coupling between the Bezier points can be switched on or off by the curve context menu option **Coupled Bezier points**.

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](#) ^[244]: d_H , d_S , d_2 , b_2
- Inclination angle g of trailing edge to horizontal (see [Approximation functions](#) ^[198])
- Inclination angle e of hub and shroud to vertical (see [Approximation functions](#) ^[198])
- Axial extension: pumps, ventilators 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).

$$b) \Delta z = (d_{1/2} - d_H)/2$$

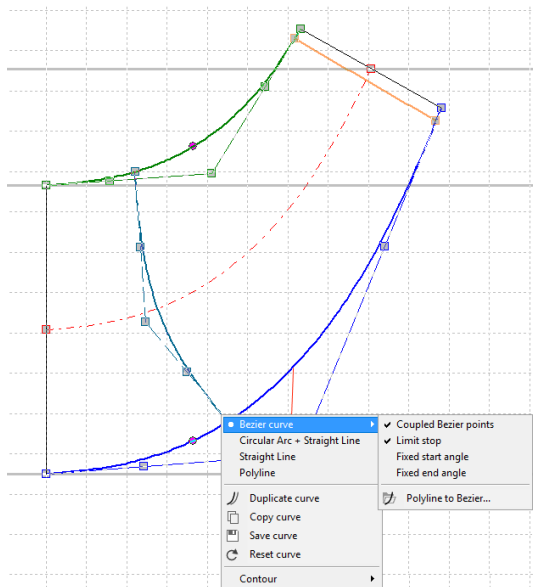
$$c) \Delta z = d_2 \left(0.014 + 0.023 \cdot \frac{d_2}{d_H} + 1.58 \cdot \varphi \right)$$

Point 1 is primary 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.

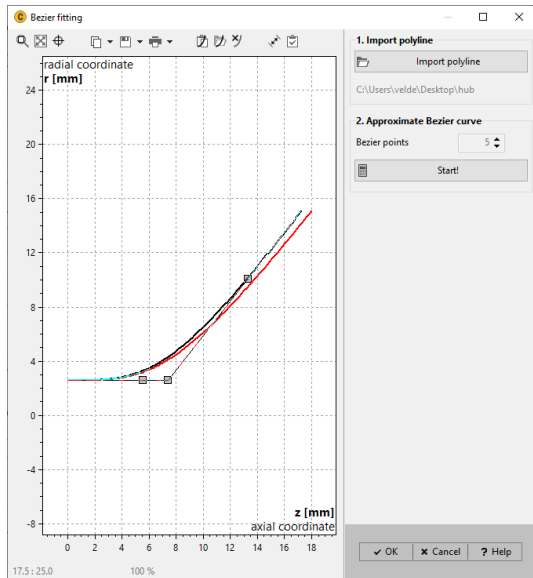
The manipulation of the contours can be achieved by shifting the positions of the Bezier points. As an alternative the position of Bezier points can be realized by input of numerical values (see [Graphical dialogs](#)^[67]). Trailing edge can be rotated by moving Bezier points 4. If <Ctrl> key is pressed 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_{rb} should grow from the suction to the impeller diameter as uniformly as possible.

7.2.1.1.1.1 Converting Polyline / Bezier

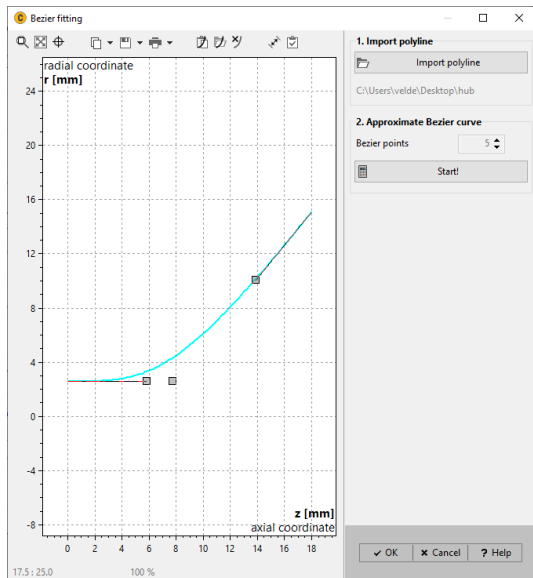


If using simple polyline for hub and/or shroud - e.g. for imported meridional geometrie - this curve can be converted to a Bezier curve. Thus, it's possible to make systematic modifications of existing geometries.

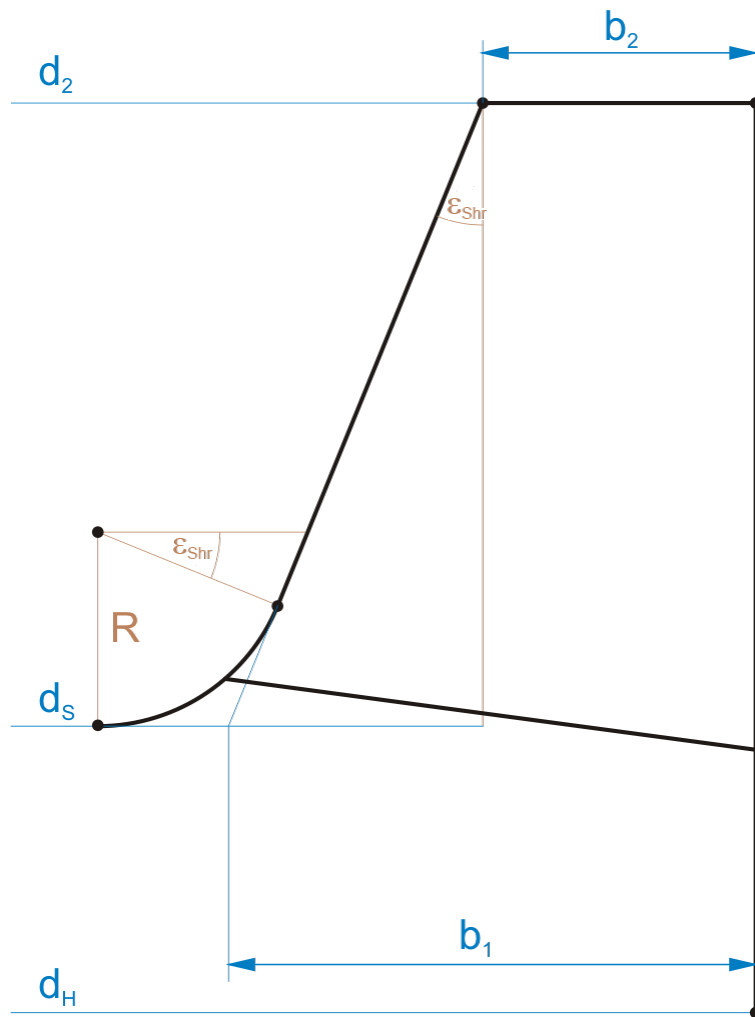


First the desired polyline is imported via **Import from file**.

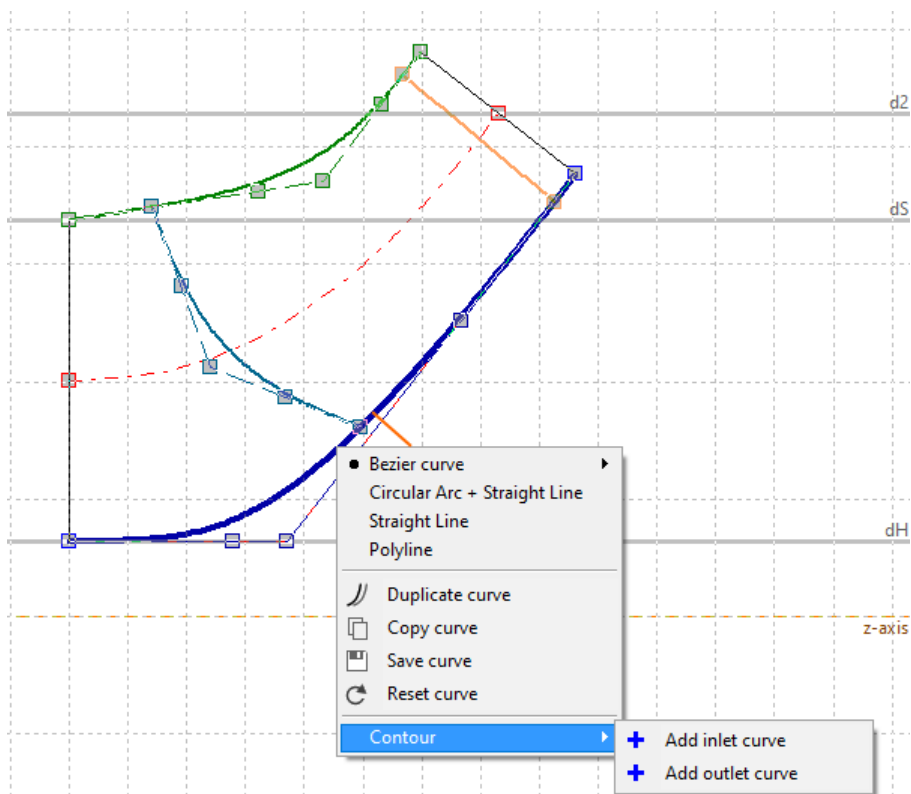
The imported curve is displayed red, the original curve blue.



By pressing the **Start!** button the position of the Bezier points is calculated in such a way that the imported poyline is replicated as exact as possible.



7.2.1.1.3 Contour

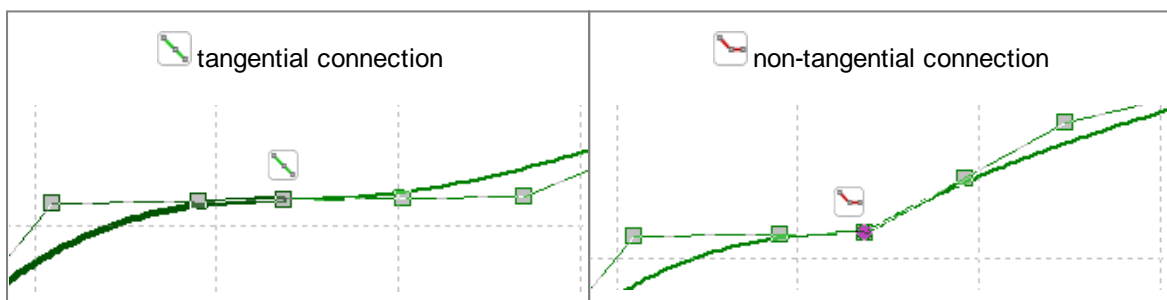


The design of hub and shroud can be expanded optionally. Therefore additional curves can be added on inlet and outlet side in order to design complex contour curves.

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 = constant**
The Leading edge runs on constant radius, i.e. parallel to rotational axis.
- **z = 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](#)³⁵⁶.

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.

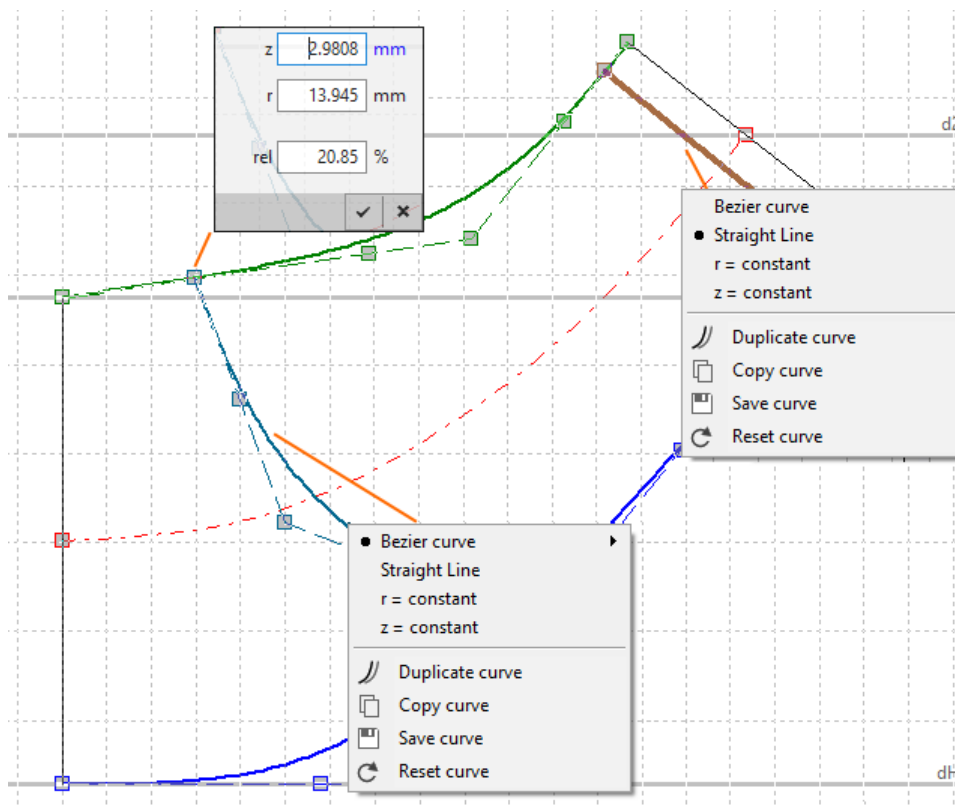
In case of **Splitter blades** each leading edge can be designed individually.

The turbine rotors and compressor impellers have straight leading edges by default, in case of turbines z = constant additionally.

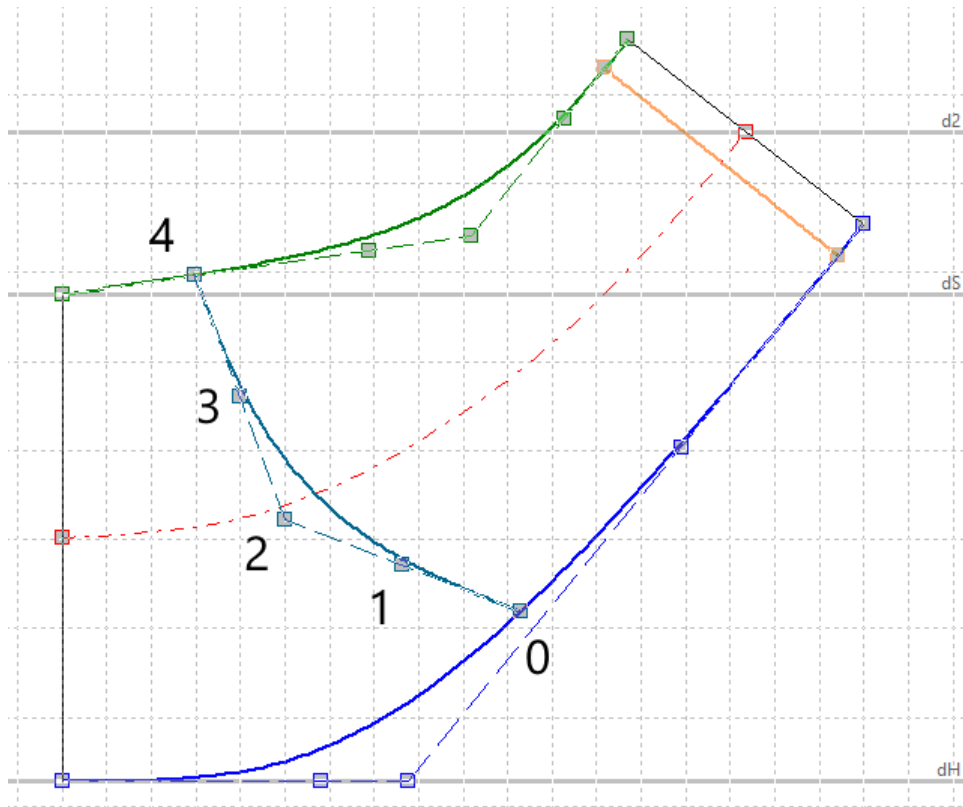
Leading edge can be designed as a straight line by selecting **Straight** in the context menu of the curve (controlled by 2 Bezier points). Additionally the edge can be strictly axial or radial (**z = const.** or **r = const.**, controlled by 1 Bezier point).

For radial impellers having $nq \approx 10 \dots 30$ 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 nq 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.



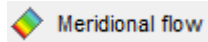
Leading/Trailing edge fixed on ...

The leading/trailing edge is fixed on meridional inlet/outlet and can not be designed.

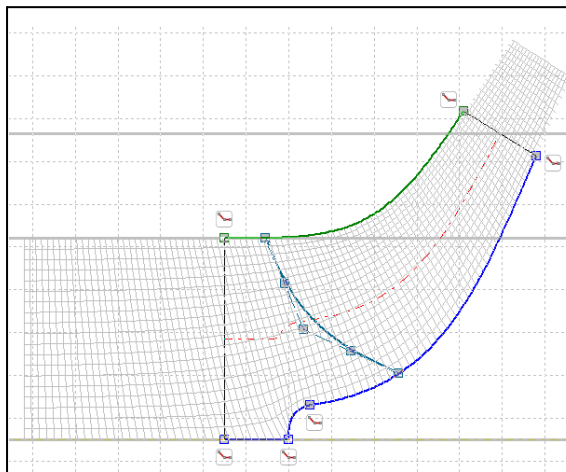
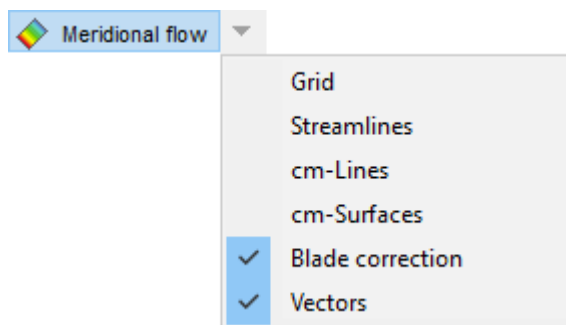
Uncheck this option to detach the leading/trailing edge from meridional inlet/outlet and design its position and shape independently.

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.



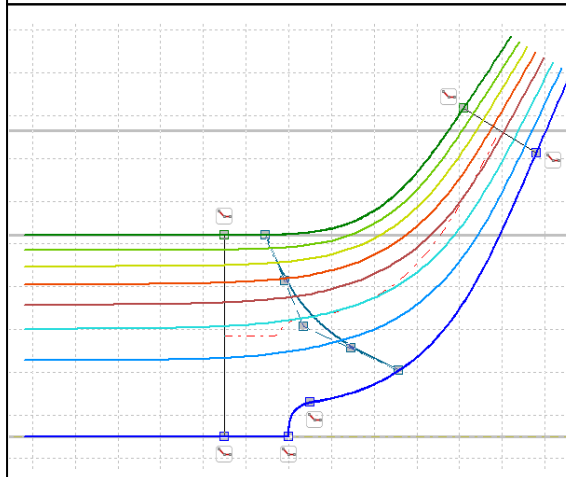
The display can be configured using the corresponding menu (all options can be combined).



Grid

After each change of the meridional contour a new computational grid is calculated.

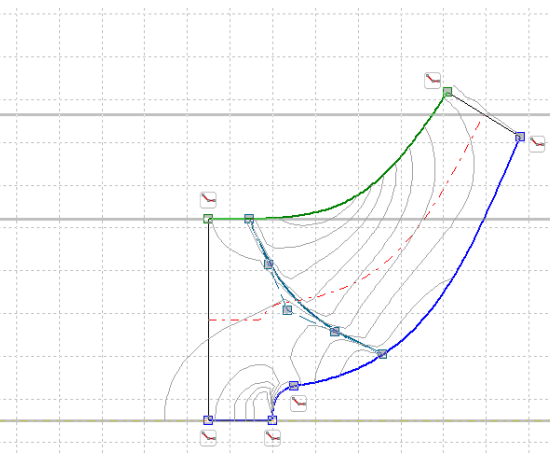
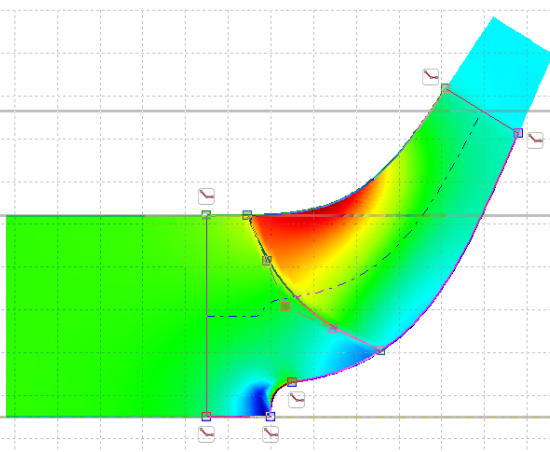
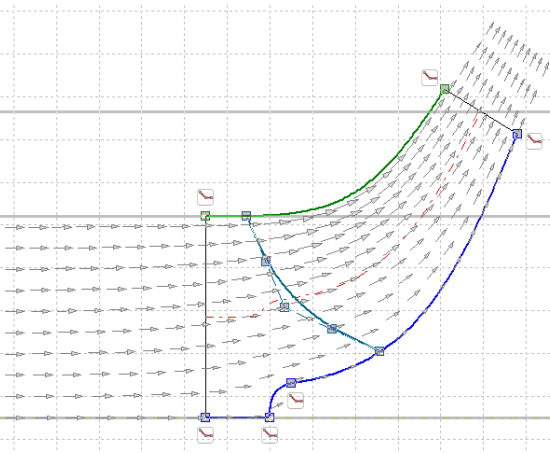
Extensions are added to the inlet and outlet in order to ease the setup of the boundary conditions.



Streamlines

meridional streamlines (lines with constant values of the stream function)

equal mass flow fraction between neighboring streamlines

	<p>cm-lines</p> <p>iso lines of const. meridional velocity c_m</p>
	<p>cm-surfaces</p> <p>iso surfaces of const. meridional velocity c_m</p> <p>(scaling is displayed below the diagram)</p>
	<p>Blade correction</p> <p>If activated, the blockage effect of blade thickness is considered for flow calculation.</p>
	<p>Vectors</p> <p>vectors of meridional velocity c_m</p>

Stream function

Within the meridian the equation for stream function 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:

$$\left[1 - \frac{1}{a^2} \left(\frac{\partial \psi}{\partial z} \right)^2 \right] \frac{\partial^2 \psi}{\partial z^2} + \left[1 - \frac{1}{a^2} \left(\frac{\partial \psi}{\partial r} \right)^2 \right] \frac{\partial^2 \psi}{\partial r^2} + \left[-\frac{2}{a^2} \frac{\partial \psi}{\partial z} \frac{\partial \psi}{\partial r} \right] - \frac{1}{r} \frac{\partial \psi}{\partial r} = 0,$$

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 as 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](#)^[86].

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](#)^[569].

Results

The meridional velocity component can be calculated by the axial velocity component:

$$c_z = \frac{r_R}{r} \frac{\rho_R}{\rho} \frac{\partial \psi}{\partial r},$$

by the radial velocity component:

with:

$$c_m = \sqrt{c_z^2 + c_r^2}.$$

r_R and ρ_R 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

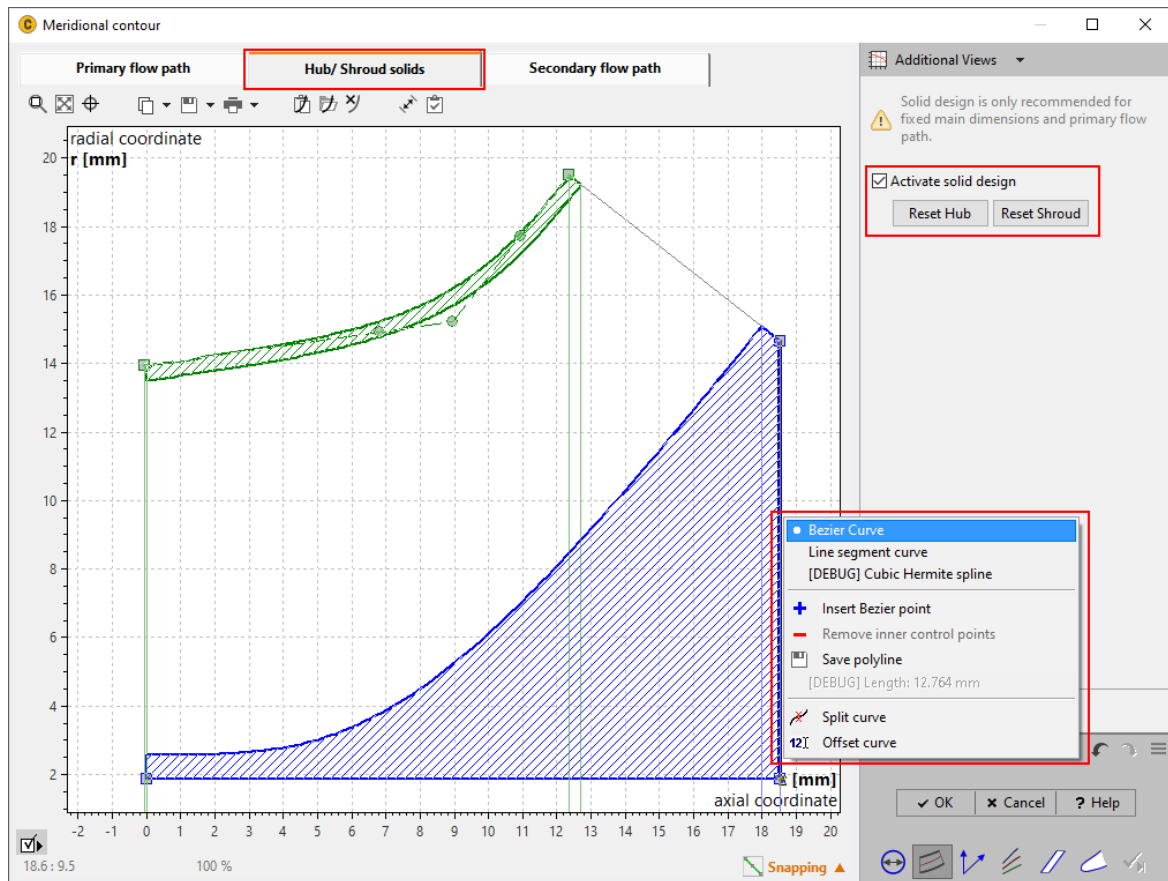
Due to the potential flow theory the given solution is only a rough estimation of the real meridional flow. One has to bear in mind that friction is not considered as well as the no slip boundary condition at hub and shroud. For detailed flow analysis CFD-techniques for solving the entire set of Navier-Stokes-Equations 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.

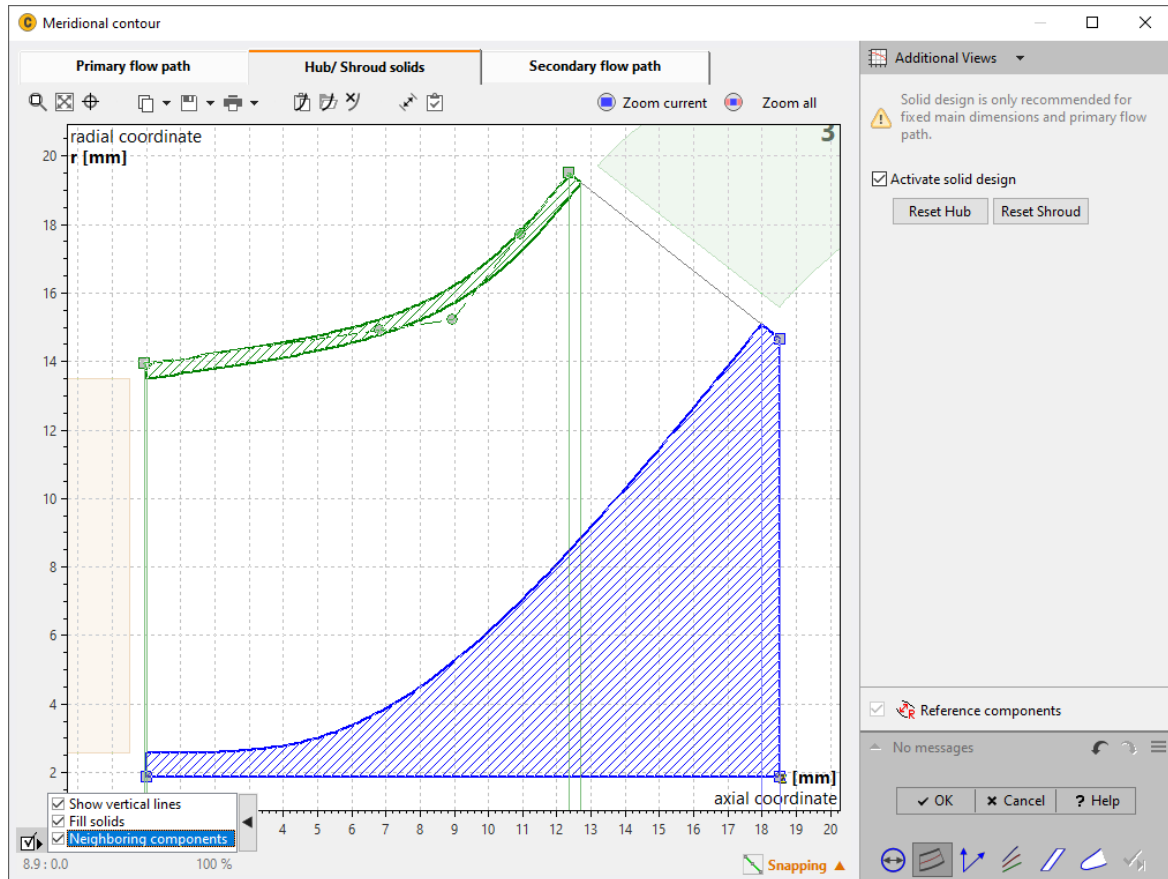
For compressible fluids it is necessary that the flow regime in the entire domain has to be far away from transonic conditions. Otherwise the equation will not have solution.

7.2.2 Hub/Shroud solids

Meridional solid can be designed by selecting the "Hub/ Shroud solids" 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.



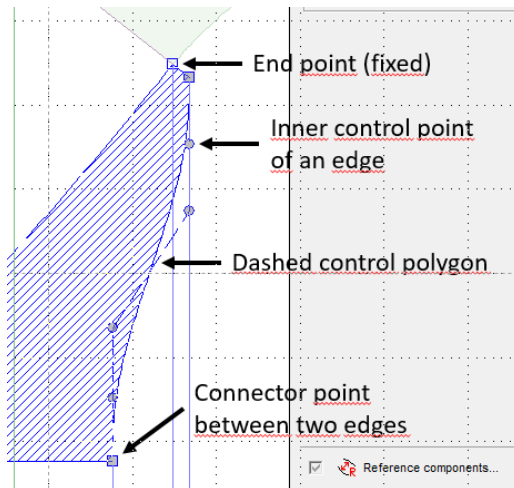
After the [model finishing](#) ⁴⁸⁷ the meridional solid parts and the blade solid are connected to a single solid geometry, "Meridian.Material domain".

Solid design

Hub & Shroud solids can be designed by manipulating their meridional contours. The feature is available if the following conditions are fulfilled:

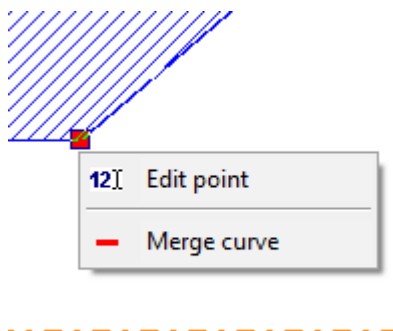
- **Hub:** Only available for existing hub contour
- **Shroud:** Only available for shrouded impellers

By activating one of the solids a default solid 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 solid 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.



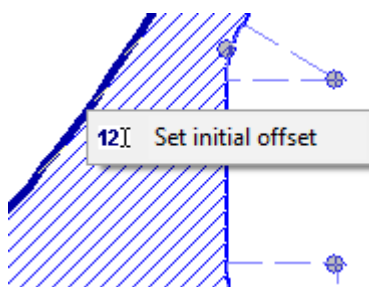
Context menus of contour wire

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.



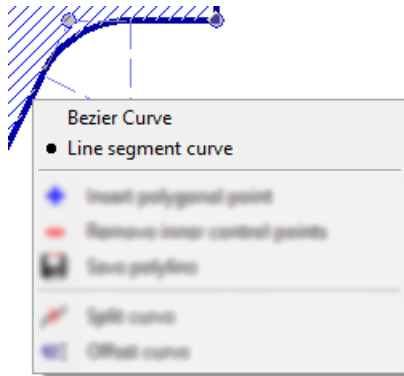
Connector point

- **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.



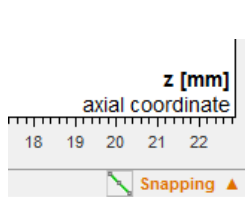
Edge types of contour wire

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](#) ³⁶⁵
- **Line segment curve**
The curve consists of straight lines and rounded corners. (optional)
→ [Details](#) ³⁶⁷

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:



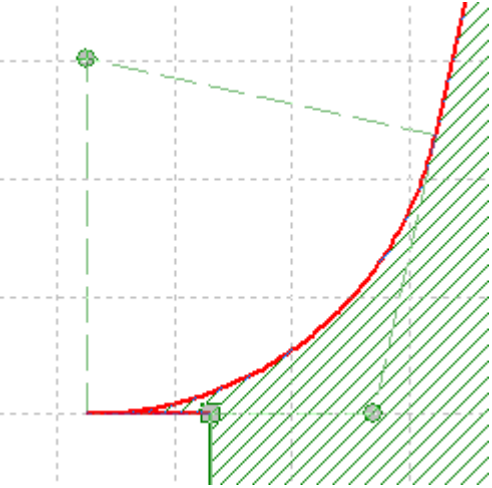
Snapping types

- **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

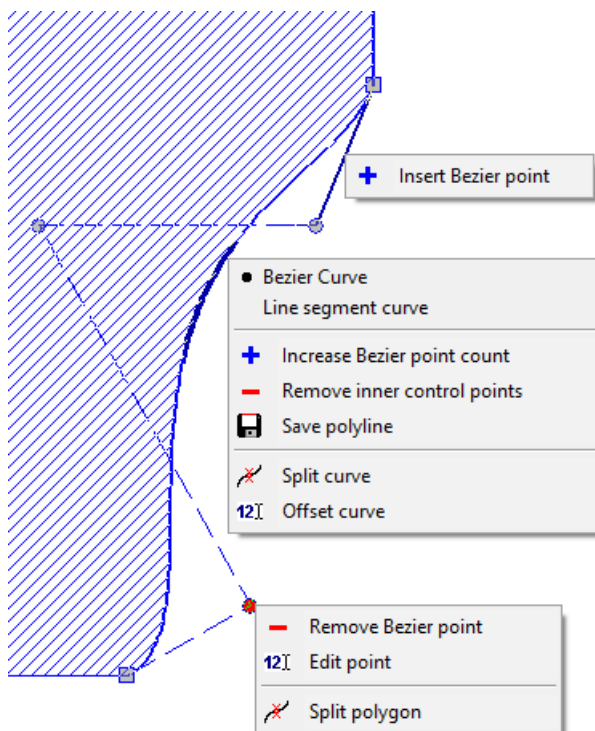
Problem	Possible solution
Hub/ Shroud contour of solid has invalid edges.	
When using Line segment curves with rounded corners the defined radius could be too large. This can result in sharp edges and very small segments.	Corner radius of the invalid (red) curve has to be reduced. This could be achieved by dragging the circle center. The curve is valid when it is no longer drawn in red.

Problem	Possible solution
	

7.2.2.1 Bezier curve

Context menus

Options of the context menu for the graphical object of a **Bezier 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.

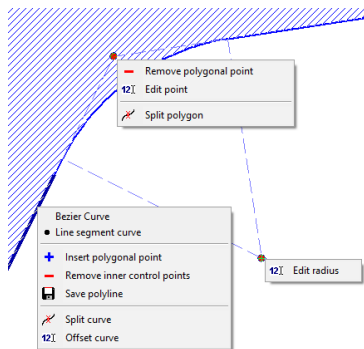
7.2.2.2 Line segment curve

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.

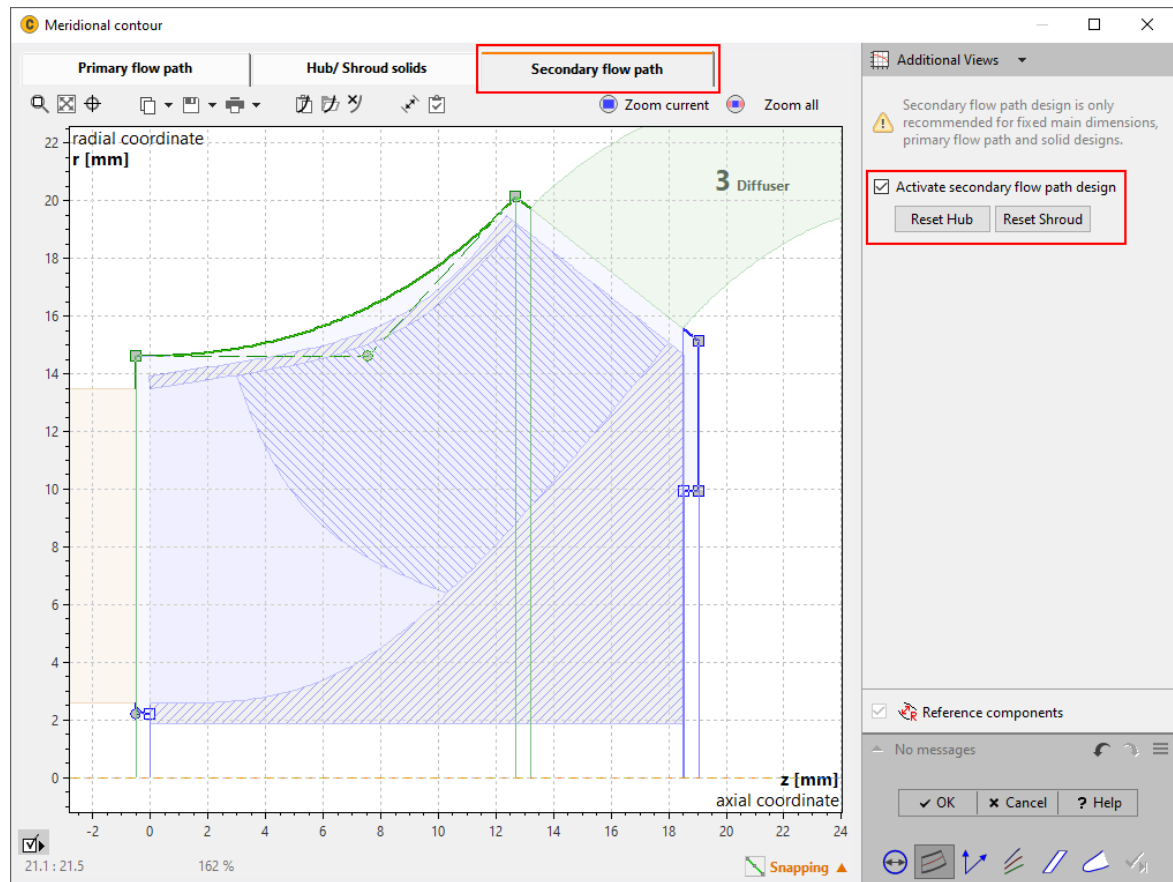
Curve

- **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 solids](#))³⁶⁰.

Endpoints of the "secondary flow path" on the solid are constrained to the solid contour. The context menu of those points provide "Edit z value" and "Edit r value".



After the [model finishing](#) ⁴⁸⁷ the meridional primary and secondary flow paths are connected to a single solid geometry, "Meridian.Fluid domain".

Hub & Shroud casing can be designed by manipulating their meridional contours. The feature is available if the following prerequisites are fulfilled:

- Solids must be activated for this impeller
- Neighboring static components have to exist
- An offset between impeller and neighboring components is required

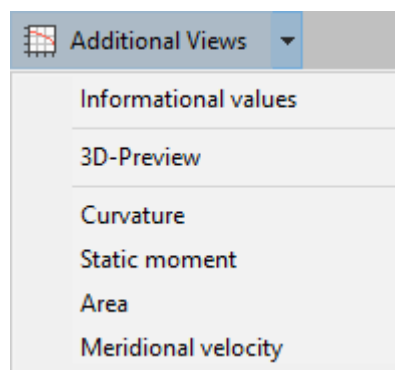
Depending on settings of main dimensions, some parts of the contour can become inactive, e.g. for unshrouded impellers the shroud is already defined by primary flow path.

Possible warnings

Problem	Possible solution
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

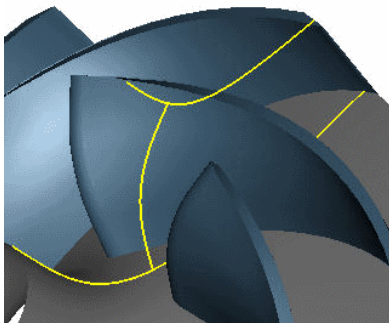
Some additional values are displayed for information:

- 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 α in the hub and shroud end points measured to the horizontal direction
- Angle α_{LE} of leading edge on hub and shroud measured to the horizontal direction
- Axial extension z of hub and shroud
- Radial extension r of hub and shroud
- Angle α_{TE} of trailing edge measured to the horizontal direction
- Default axial extension z_D from inlet shroud to outlet midline (defined for radial impellers only)

- Maximal axial extension z_M of complete meridional shape
- Maximal radial extension 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_2
- LE diameter d_{1ave} 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

The static moment is the integral of the curve length (x) in the blade area multiplied by the radius (r):

$$S = \int_{r_{LE}}^{r_{TE}} r dx$$

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

? IMPELLER | Blade properties

Definition of blade properties is made in two steps:

- (1) [Blade setup](#) 
- (2) [Blade angles](#) 

Specification of number of blades and number of spans

Blades

Number of blades

Number of spans 2 ... 15

Usual number of blades are:

Pump	3 ... 7 Wastewater: 1 ... 3 Inducer: 1 ... 3
Ventilator	6 ... 10 Squirrel cage: 30 ... 60
Compressor	Depending on blade exit angle β_2 : <ul style="list-style-type: none"> • 12 for $\beta_2 \approx 30^\circ$ • 16 for $\beta_2 \approx 45^\circ \dots 60^\circ$ • 20 for $\beta_2 \approx 70^\circ \dots 90^\circ$
Radial turbine	12 ... 20
Axial turbine	30 .. 70 (100)

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 **[for radial & mixed-flow pumps, ventilators, compressors only]**:

$$z = k_z \frac{d_2 + d_1}{d_2 - d_1} \sin \frac{\beta_1 + \beta_2}{2}$$

with $k_z = 6.5 \dots 8.0$ for compressors, else $5.0 \dots 6.5$.

The recommended number of blades using the Zweifel work coefficient is displayed as a hint at the information image **[for axial turbines only]**:

with z the axial chord length and d_{av} the average impeller diameter.

The Zweifel work coefficient is in the range of $\psi = 0.75..1.15$ and is specified in the [approximation functions](#) ¹⁹⁸.

The span positions are illustrated as meridional lines in the **Meridian** diagram in the information area. By default the meridional lines are equally spaced between hub and shroud.

Splitter linked to Main blade

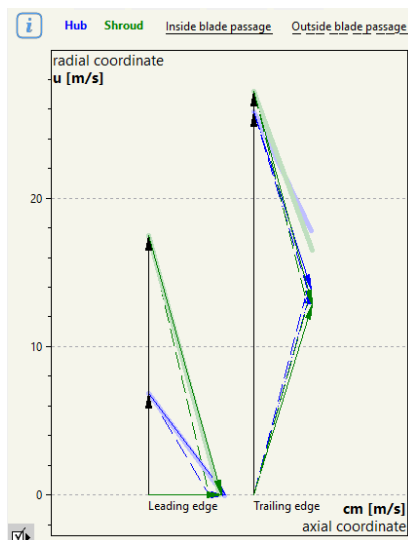
If the impeller has splitter blades then the shape of the splitter can be linked to the main blade optionally. If linked the splitter blades are truncated main blades. Otherwise the splitter blade can be designed completely independent.

Blades

Number of blades	i	16	8 Main 8 Splitter	<input checked="" type="checkbox"/> Splitter linked to Main blade
Number of spans		5	2 ... 11	

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.

Span = 1 (Hub)			Span = 6 (Shroud)		
	Leading edge	Trailing edge	Leading edge	Trailing edge	
z	64.7	94.3	11.95	55	
d	73.5	278.7	188.5	293.3	
αF	90	15.8	90	17.1	
βF	37.5	18.1	15.8	15.3	
u	6.8	25.8	17.5	27.2	
c _m	5.2	3.9	4.9	3.9	
c _u	0	13.9	0	12.8	
c _r	4.5	3.8	0.4	3.7	
c _{ax}	2.7	0.8	4.9	1.4	
c	5.2	14.4	4.9	13.4	
w _u	-6.8	-12	-17.5	-14.4	
w	8.6	12.6	18.1	14.9	
τ	1.28	1.08	1.2	1.09	
i δ	-0.4	8	-0.3	4.9	
w ₂ /w ₁		1.47		0.82	
c ₂ /c ₁		2.76		2.72	
ΔαF		-74.2		-72.9	
ΔβF		-19.4		-0.4	
ΔβB		-11		4.8	
γ		0.85		0.87	
Δ(cu·r)		1.931		1.88	
T		23.75		24.21	
H		36.49		35.52	

(2) Values

Numerical values of velocity components and flow angles are displayed in a table. A short description is at mouse cursor too:

z	Axial position
d	Diameter
F	Angle of absolute flow to circumferential direction
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: $w_u + c$
w	Relative velocity
τ	Obstruction by blades (see below)
i	Incidence angle: $i = \beta_B - \beta_1$
	Deviation angle: $\delta = \beta_B - \beta_2$
w ₂ /w ₁	Deceleration ratio of relative velocity
c ₂ /c ₁	Absolute velocity ratio
F	Abs. deflection angle: $F = F_1 - F_2$
F	Rel. deflection angle: $F = F_1 - F_2$
B	Blade deflection angle: $B = \beta_1 - \beta_2$
	Slip coefficient
(cu·r)	Swirl difference
T	Torque
H/ p _t	Head/Pressure difference (total-total)

Default blade angles βB [°] for main blade using "Free-form 3D" blade shape. Based on:

- Shockless inflow for βB1 at leading edge
- Euler equation for βB2 at trailing edge

	βB1 [°]			βB2 [°]		
	optimal	current	ΔβB [°]	optimal	current	ΔβB [°]
1	37.4	37.1	-0.3	25.3	26.1	0.8
2	32.0	32.8	0.8	24.2	24.9	0.7
3	27.0	28.4	1.4	23.1	23.8	0.7
4	22.4	24.1	1.7	22.1	22.6	0.5
5	18.6	19.8	1.2	21.2	21.4	0.2
6	15.7	15.5	-0.2	20.4	20.2	-0.2

(3) Default βB, mean line design only

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 β_{B1} at blade leading edge
- Euler equation for β_{B2} at blade trailing edge

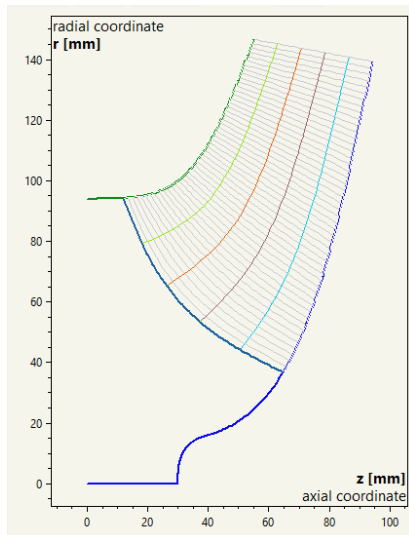


Some blade angle values result from mean line constraints for simple blade shapes.

For some simplified blade shapes the blade angles of some

sections result from the mean line design - see [Blade angles/ "Auto"](#) ³⁹¹.

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.



(4) Meridian

The Meridian with the locations of the spans is displayed in this diagram.

Radial element blades

For Radial element blades the number of spans is fixed to 11. Furthermore a **Distribution exponent** can be specified. This exponent has influence on the distribution of spans and herewith especially on the shape of the leading edge (turbine). For highly spatial curved blades the continuity of the blade surface can be influenced by this parameter.

Distribution exponent (for span positions)

Impact:

Distribution exponent = 1: spans uniformly distributed (default)

Distribution exponent < 1: spans concentrated towards shroud

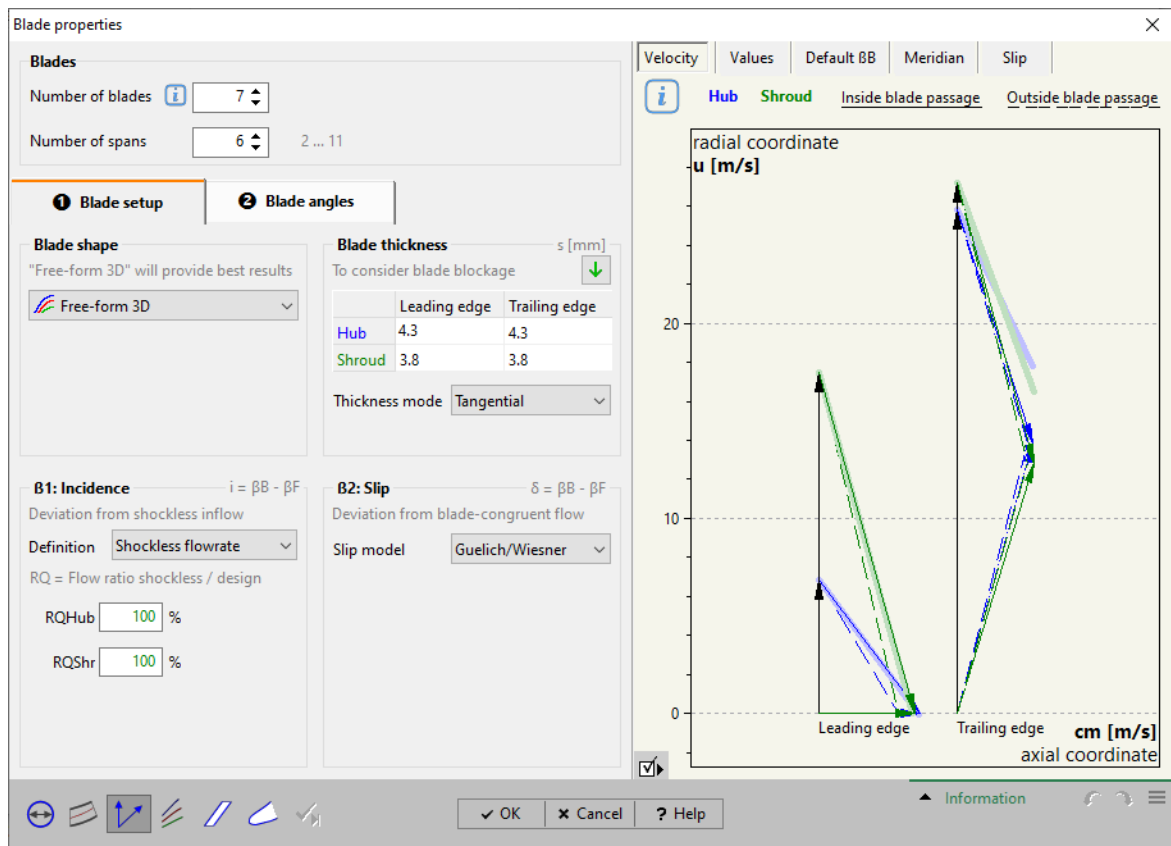
Distribution exponent > 1: spans concentrated towards hub

7.3.1.1 Blade setup

? Impeller | Blade properties



On page **Blade setup** basic blade properties are defined.

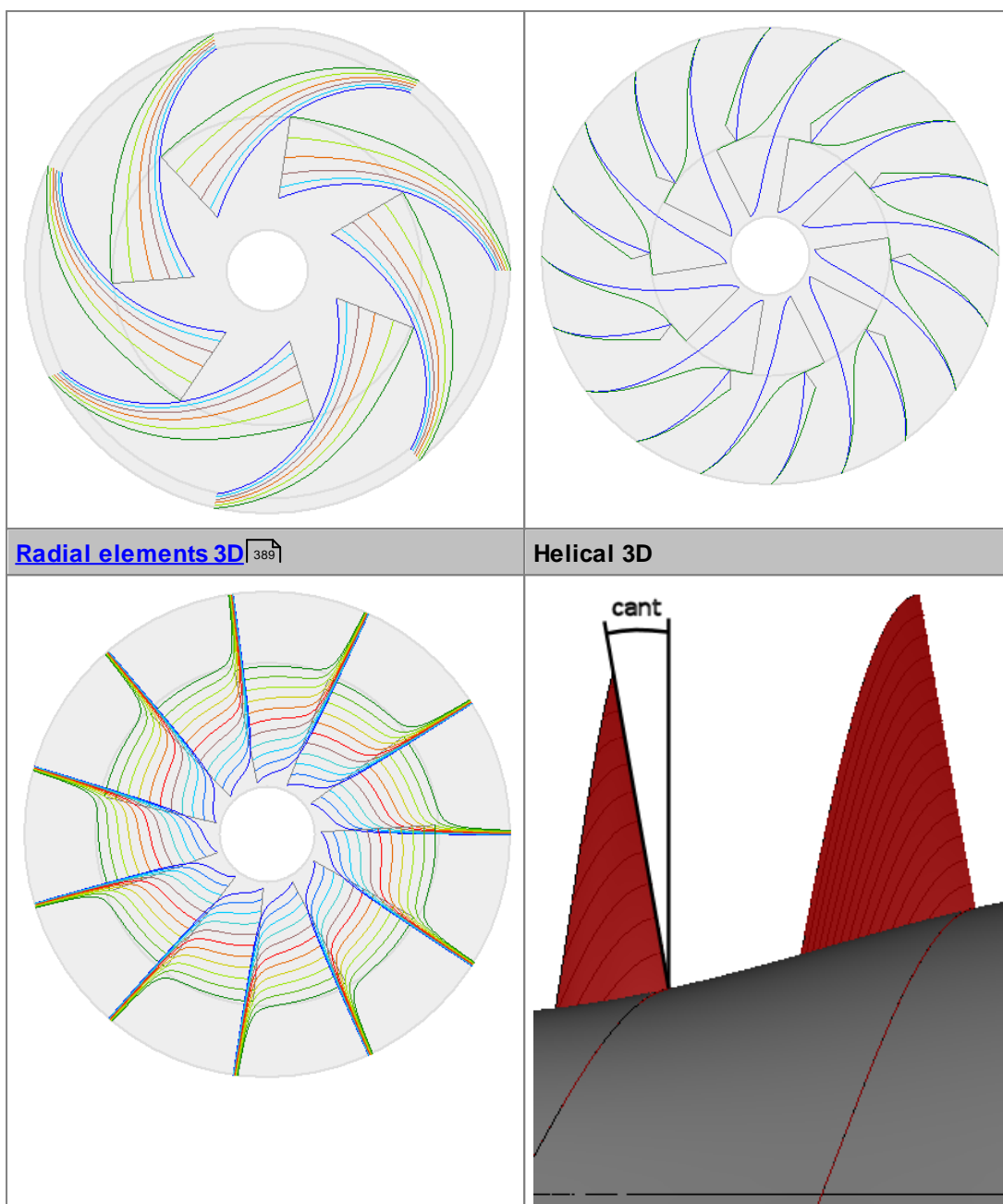


(1) Selection of desired Blade shape

3D types: blade is curved in 3D

Free-form 3D

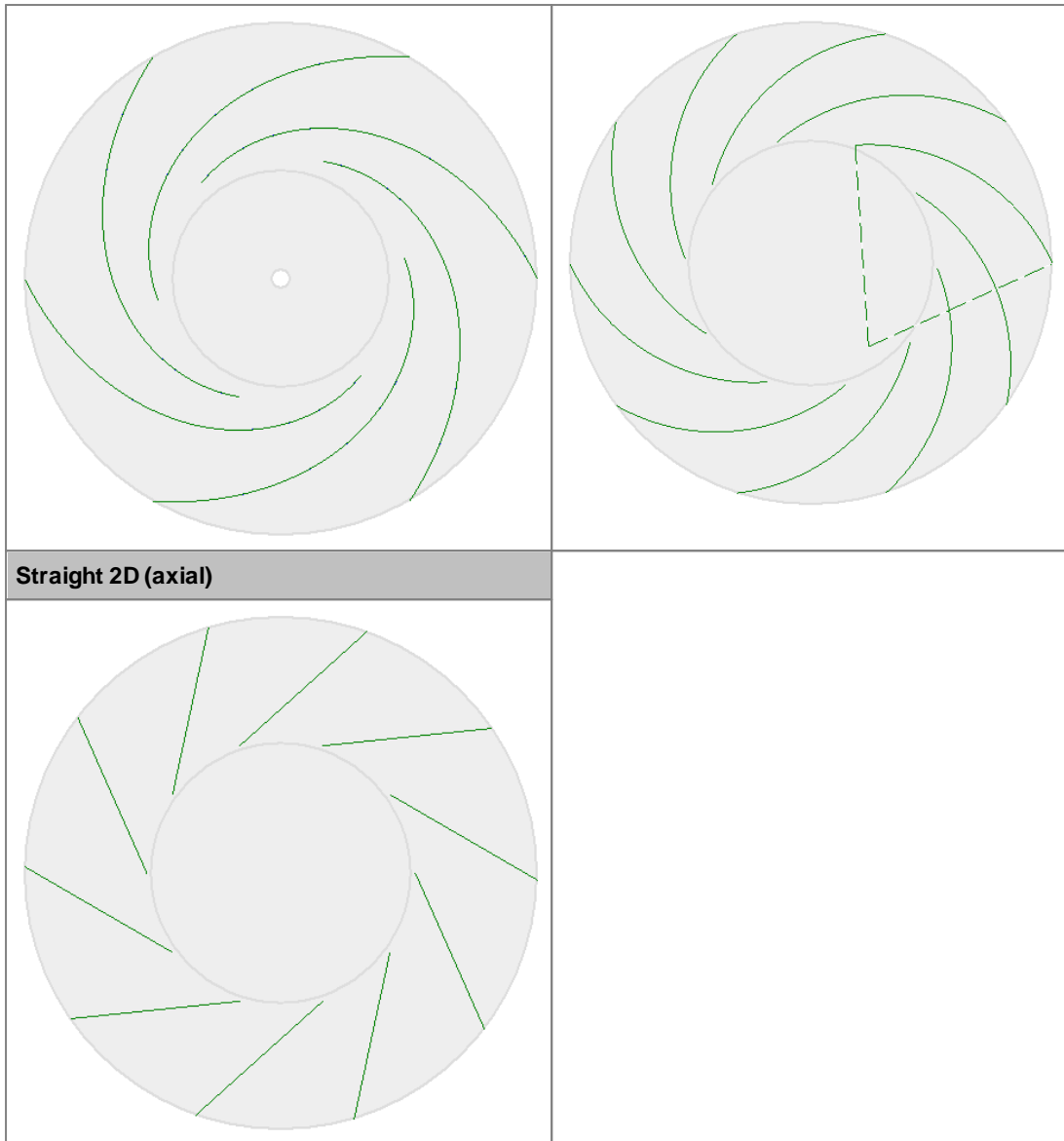
[Ruled surface 3D](#) 386



2D (axial) types: blade is curved in 2D when looking in axial direction

Free-form 2D (axial)

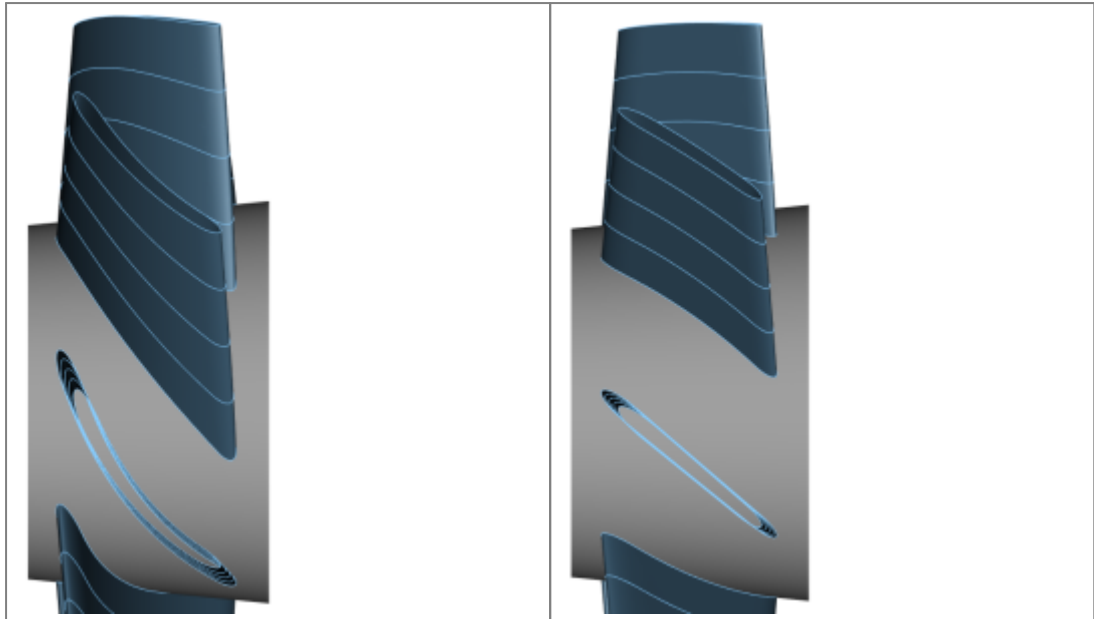
Circular 2D (axial)



2D (radial) types: blade is curved in 2D when looking in radial direction

Free-form 2D (radial)

Straight 2D (radial)



The initial blade shape depends on the machine type and can be customized in the [Impeller preferences](#) ¹⁹⁶.

PUMP	
Radial & Mixed-flow	Free-form 3D
+ Waste water pump	Free-form 2D (axial)
Axial	Free-form 3D
+ Inducer	Helical 3D
VENTILATOR	
Radial & Mixed-flow	Circular 2D (axial)
+ Squirrel cage	Circular 2D (axial)
Axial	Free-form 3D
COMPRESSOR	
Radial & Mixed-flow	Ruled surface 3D
TURBINE	
Radial & Mixed-flow	Radial elements 3D ³⁸⁹

Axial

Free-form 3D

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](#)^[386].

Limitations

	Blade shape	Impeller type	Meridional shape	Splitter blades
3D	Free-form 3D	(no limitations)		
	Ruled surface 3D			
	Radial elements 3D			
	Helical 3D	axial impellers only		not available
2D (axial)	Free-form 2D (axial)	radial & mixed-flow impellers only	available only if the meridional direction is mainly radial: hub must overlap shroud in z-direction about 50% or more hub must not have axial parts within blade area	
	Circular 2D (axial)			
	Straight 2D (axial)			
2D (radial)	Free-form 2D (radial)	axial impellers only	available only if the projection of the shroud mean line in radial direction (relating to leading edge point) hits the hub surface hub must not have radial parts within blade area	not available
	Straight 2D (radial)			

(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](#) ^[198].

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 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 $= s / \sin(\beta_l)$.
- **Orthogonal:** the blade thickness is not projected at all $= s$.
- **None:** the blade thickness is not considered $= 0$.

These options will have an influence on the calculation of the meridional velocity component c_m and therefore on the [blade angle](#) ^[389] calculation when pressing button **Calculate β_B** or if the checkbox **Automatic** is selected. Beyond it, it will influence the [meridional flow calculation](#) ^[356] too.

(3) Specification of incidence angle on blade leading edge (deviation from shockless inflow) on panel β_1 : Incidence

Pump, Ventilator, Compressor		Turbine
from ratio Q for shockless inflow / Q for max. efficiency		fully automatic by theory of WIESNER adapted by Aungier ^[395]
or		or
directly by incidence angle i		directly by incidence angle i

(RQ=100% or $i=0^\circ$ for shockless inflow)		($i=0^\circ$ for shockless inflow)
or		
from ratio of incidence angle i / blade angle β_B	$i_{\text{Rel}} = i / \beta_B$	

For **inducer pumps** there is an additional check if the incidence is $> 1^\circ$ even for high flow rates (overload) to prevent pressure side cavitation.

Squirrel cage ventilators have high incidence typically resulting in blade inlet angles $\beta_{1B} \approx 80^\circ$.

[Pump, Ventilator, Compressor impellers only]

(4) Estimation of slip velocity in panel 2: slip

You have to use one of the following slip models:

Slip model theory	Hints
GÜLICH/ WIESNER ^[399]	closed empirical model, extended Wiesner model
AUNGIER/ WIESNER ^[400]	
VON BACKSTROEM ^[402]	closed empirical model
PFLEIDERER ^[401]	input of coefficient a
User-defined	manual selection of angular deviation $\beta_{2B}-\beta_2$ resp. velocity ratio $c_{u2}/c_{u2,\infty}$
Specific definitions ^[403]	specific slip models for specific impeller types

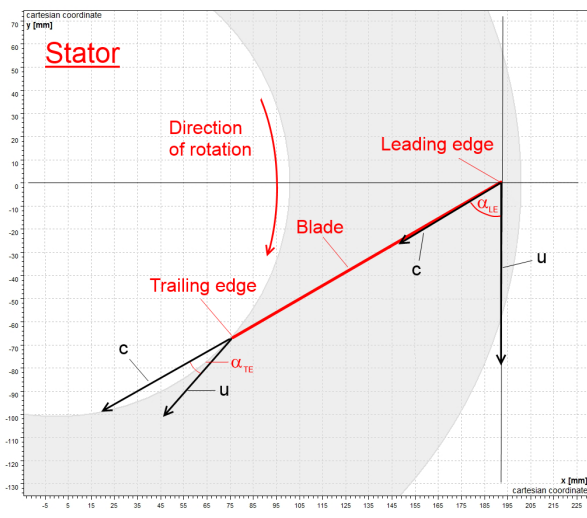
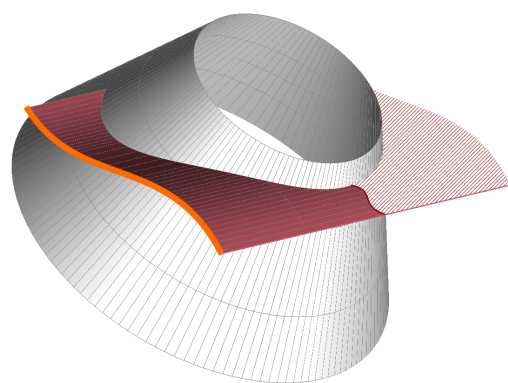
Using the button **Show calculation details** provides [specific information about the B2 calculation](#) ^[403].

Possible warnings

Problem	Possible solutions
Cross section area very small on LE/TE@Hub/Shroud	

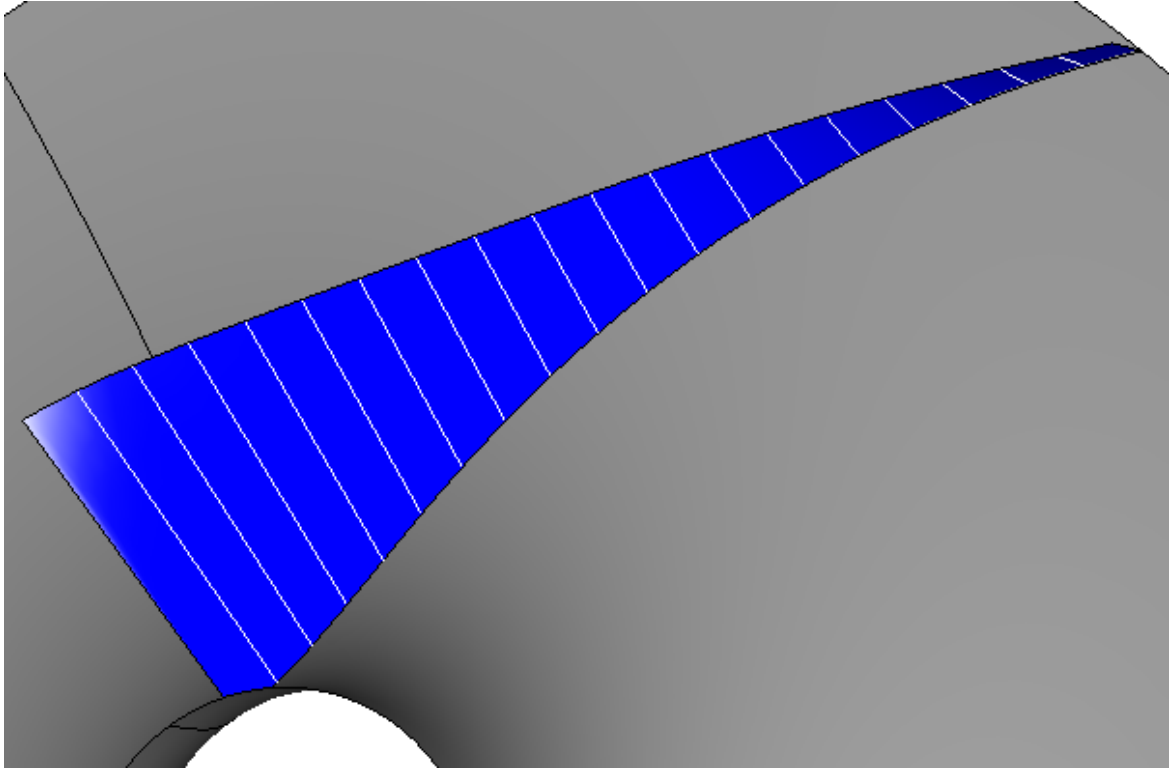
Problem	Possible solutions
Blade thickness is blocking a significant part of the flow passage.	Reduce number of blades and/or blade thickness
Cross section area too small on LE/TE@Hub/Shroud	
Blade thickness is blocking the flow passage completely	Reduce number of blades and/or blade thickness
Blade number different than initially defined. [Wastewater pumps only]	
Number of blades differs from the number that was initially selected in Main dimensions ^[244] used for empirical correlations to calculate the main dimensions. This can result in inconsistent impeller design.	<p>It makes no sense to use other number of blades for main dimension calculation and blade design itself.</p> <p>Before modifying the number of blades here one should adapt the number in Main dimensions^[244], update the empirical parameters and the main dimension.</p>
Mean lines (except hub) may be extrapolated. ("Free-form 2D" blade shape only)	
<p>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.</p> <p>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.</p>	<p>Use axis parallel (const. radius) or slightly sloping meridional leading/ trailing edge.</p> <p>Leading edge: The shroud point should have higher or equal radius than the hub point.</p> <p>Trailing edge: The shroud point should have lower or equal radius than the hub point.</p>
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	Use radial (const. axial position) or sloping meridional leading/ trailing edge.

Problem	Possible solutions
<p>fibers ^[389]. The resulting blade shape is three-dimensional.</p> <p>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.</p>	<p>Leading edge: The shroud leading edge should have a higher or equal axial position compared to the hub.</p> <p>Trailing edge: The shroud trailing edge should have a lower or equal axial position compared to the hub.</p>
<p>"Ruled surface" blades may export low quality surfaces when using two mean lines only. ("Ruled surface 3D" blade shape only)</p>	
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").
<p>"Straight 2D (axial)" blades not possible for selected combination of meridional leading edge contour and blade angle.</p>	
<p>The hub mean line is the master mean line. All other mean lines are adapted automatically in order to overlap the hub mean line if viewing in z-direction.</p> <p>If the other mean lines are extended they will be extrapolated automatically. For specific combinations of meridional leading edge and blade angles B1 ^[389] an extrapolation is impossible.</p>	<p>Leading edge ^[354]: The point on shroud should be moved to a higher radius.</p> <p>B1 ^[389]: Blade angle should be increased.</p>
<p>"Circular + Free-form 2D (axial)" blades not possible for selected distance and angle combination.</p>	
Construction of circular arc is not possible for given parameters. Therefore calculation of blade is blocked	Modify r_3 or a_3 for this blade shape. For further information see blade properties ^[504] for stators.
<p>Extrapolation of "Circular + Free-form 2D (axial)" blades not possible for secondary spans.</p>	
The minimal inner radius for the secondary spans is limited by the circular arc (design curve) defined by r_3 and a_3 .	Try to reduce effect of extrapolation by adjusting meridional Leading edge ^[354] or change parameters defining the circular arc (design curve) of this blade shape. See blade properties ^[504] for stators.

Problem	Possible solutions
"Straight 2D (axial)" blades not possible for selected combination of meridional trailing edge contour and blade angle.	
<p>The blade angle is too small or too large - therefore designing a "Straight 2D" blade shape is impossible.</p> 	<p>Trailing edge^[354]: The edge should be moved to a higher radius.</p> <p>LE/ LE^[389]: Blade angle should be increased.</p>
"Straight 2D (radial)" blades not possible for selected combination of meridional leading edge contour and blade angle	
Blade angles not within a valid range.	
<p>Projection of the designed mean line onto the other spans fails for this blade shape.</p> 	<p>Blade angle should be specified within the recommended range.</p>

7.3.1.1.1 Ruled Surface blade

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.

Blades

Number of blades i

8 Main
8 Splitter

☒ Splitter linked to Main blade

Number of spans

2 ... 11

1 Blade setup

2 Blade angles

Blade shape

"Free-form 3D" will provide best results

≡ Ruled surface 3D

Options for linked splitter

Exact (adjusts main blade)

Blade thickness

s [mm]

To consider blade blockage

Leading edge

Trailing edge

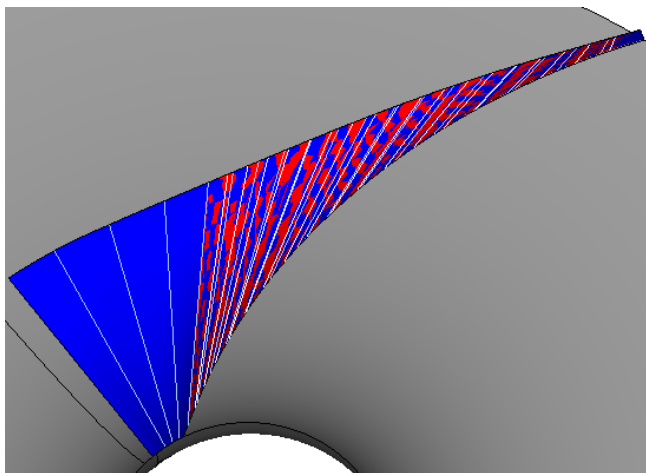
	Main	Splitter	Main	Splitter
Hub	2.6	-	2.6	-
Shroud	2.6	-	2.6	-

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.

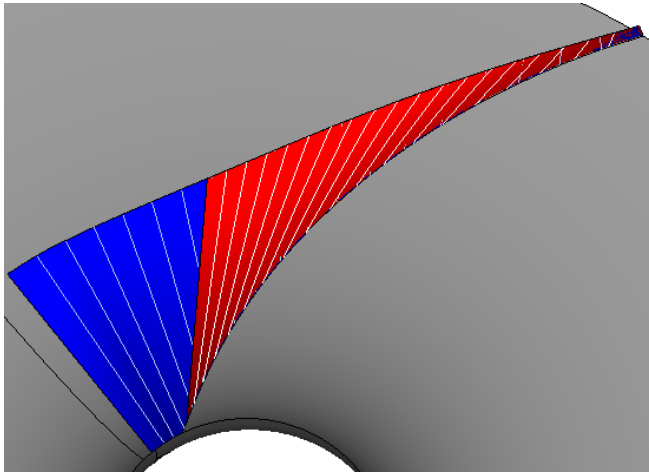
Mean lines only: The blade geometry of the splitter is designed using the mean lines of the main blade. The advantage of this option is a higher flexibility in design of a curved leading edge of the splitter. (depends on the number of mean lines)

The following pictures illustrate the combination of different options (splitter is rotated into the main blade for illustration):



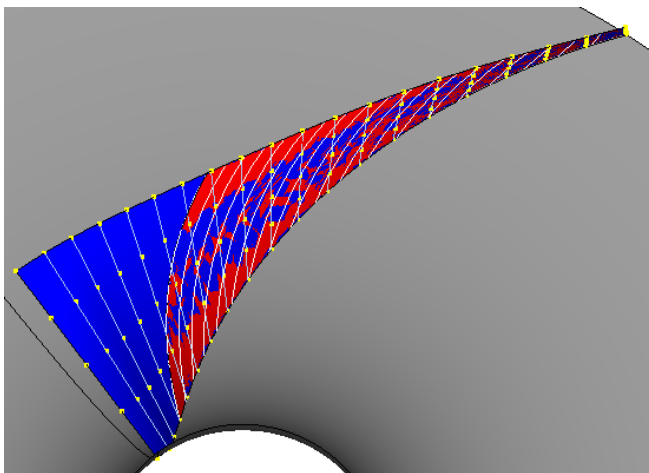
- Splitter linked to Main Blade
- 2 spans
- Exact (adjusts main blade)

Main and Splitter are using identic rulings. The splitter leading edge is influencing the rulings and therefore the main blade.



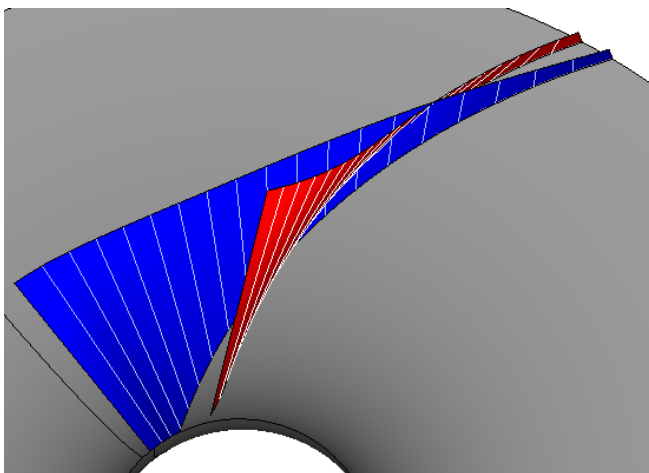
- Splitter linked to Main Blade
- 2 spans
- Mean lines only

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

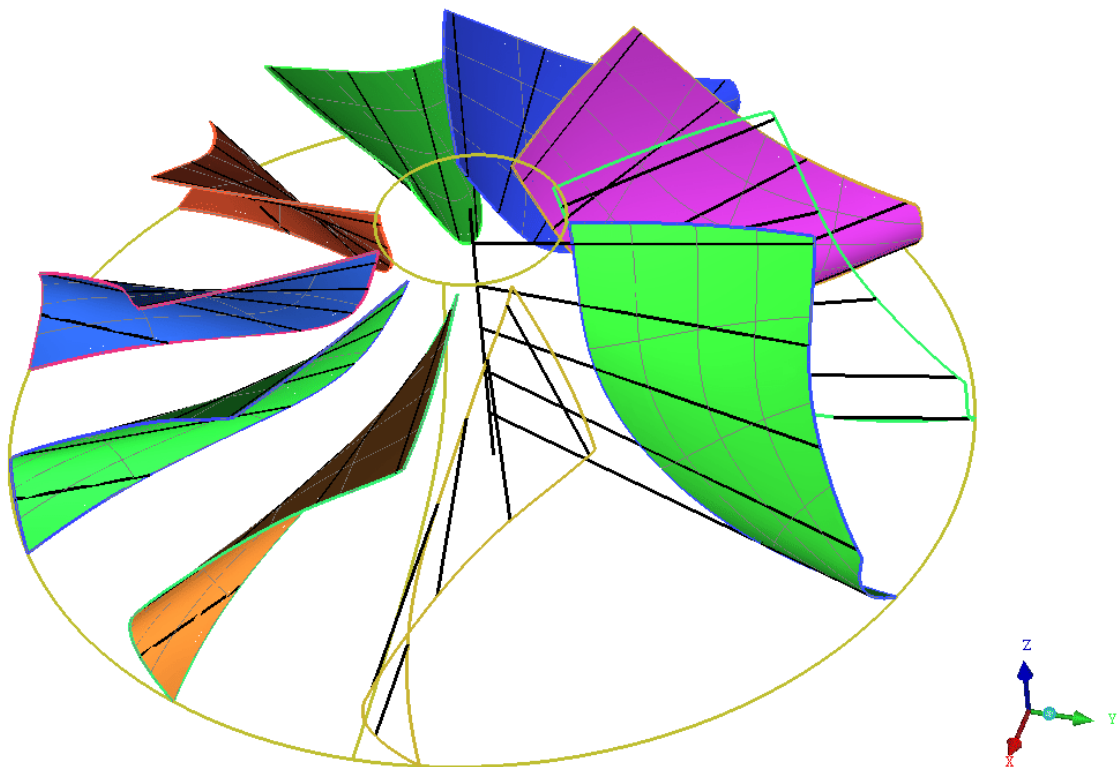
Main and splitter are using their own rulings. There is no coupling between splitter and main blade. The splitter shape can highly deviate from the main blade.

7.3.1.1.2 Radial element 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): $\approx 90^\circ$
- [Inclination angle](#) ^[345] from hub and shroud to the horizontal: $\beta' < 90^\circ$
- Vertical trailing (turbines) or leading edge resp. (all other types) with $z \approx \text{const.}$
- Small wrap angle: $\varphi \approx 360^\circ/\text{number of blades}$

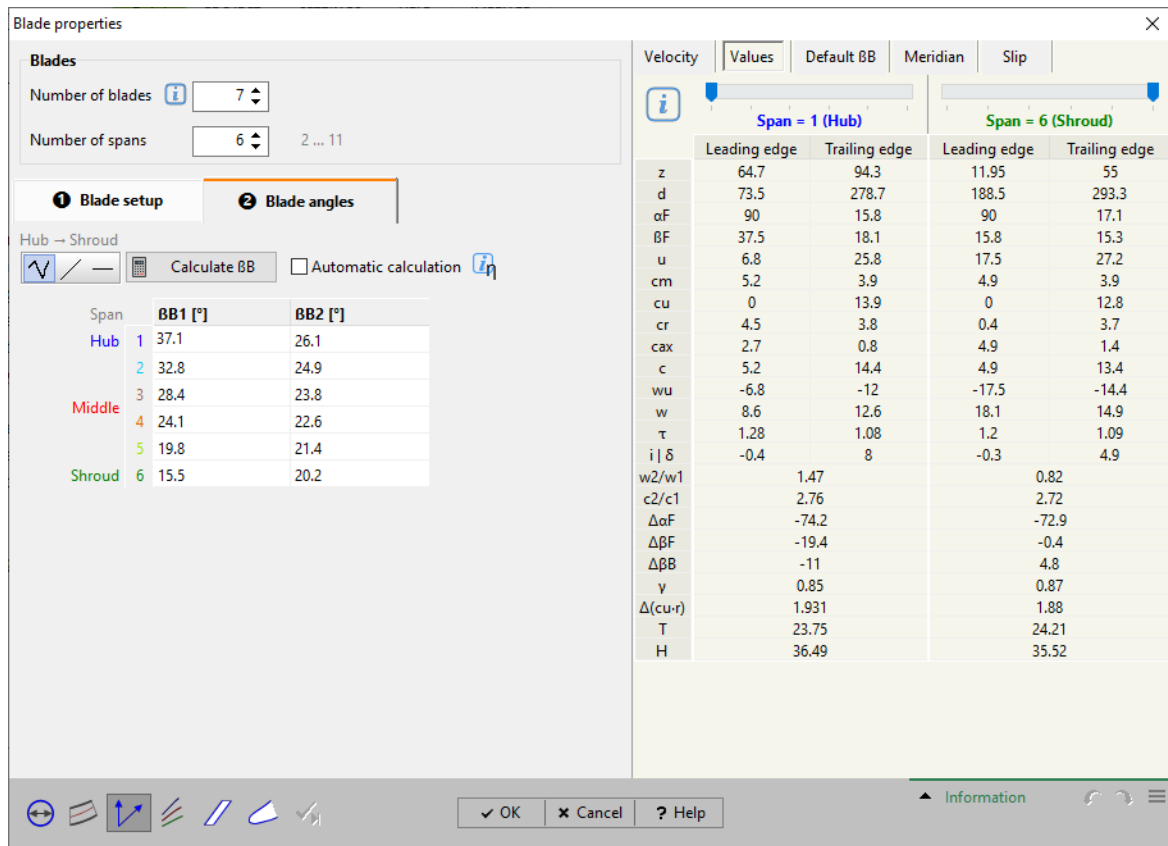


7.3.1.2 Blade angles

? Impeller | Blade properties



On this page the **blade angles** are calculated.



Blade properties

Blades

Number of blades: 7

Number of spans: 6 (2 ... 11)

Blade setup | **Blade angles**

Hub → Shroud

Calculate BB ☐ Automatic calculation

Span	BB1 [°]	BB2 [°]
Hub 1	37.1	26.1
2	32.8	24.9
3	28.4	23.8
Middle 4	24.1	22.6
5	19.8	21.4
Shroud 6	15.5	20.2

Velocity | **Values** | **Default BB** | **Meridian** | **Slip**

Span = 1 (Hub) | Span = 6 (Shroud)

	Leading edge	Trailing edge	Leading edge	Trailing edge
z	64.7	94.3	11.95	55
d	73.5	278.7	188.5	293.3
α_F	90	15.8	90	17.1
β_F	37.5	18.1	15.8	15.3
u	6.8	25.8	17.5	27.2
cm	5.2	3.9	4.9	3.9
cu	0	13.9	0	12.8
cr	4.5	3.8	0.4	3.7
cax	2.7	0.8	4.9	1.4
c	5.2	14.4	4.9	13.4
wu	-6.8	-12	-17.5	-14.4
w	8.6	12.6	18.1	14.9
τ	1.28	1.08	1.2	1.09
i δ	-0.4	8	-0.3	4.9
w2/w1	1.47		0.82	
c2/c1	2.76		2.72	
$\Delta\alpha_F$	-74.2		-72.9	
$\Delta\beta_F$	-19.4		-0.4	
$\Delta\beta_B$	-11		4.8	
γ	0.85		0.87	
$\Delta(cu-r)$	1.931		1.88	
T	23.75		24.21	
H	36.49		35.52	

OK Cancel Help

Later designed mean lines depend on the number and the meridional position of profile sections as well as the blade angles. Blade angles β_{B1} and β_{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](#)^[405] 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 B

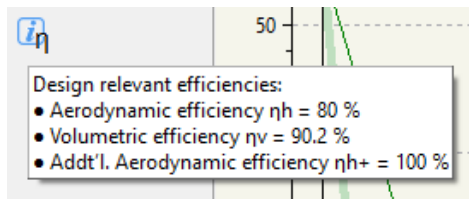
- Calculation of blade angles using values from [Blade setup](#)^[375] by pressing button **Calculate B**

- 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 ventilators 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 β_B** . 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](#)^[393]

→ 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](#)^[405] 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"](#)^[412] 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 r_1 , r_2 and blade angles β_{B1} , β_{B2} at leading and trailing edge. If the calculation of the circular blade is not possible a warning symbol is displayed.

Possible warnings

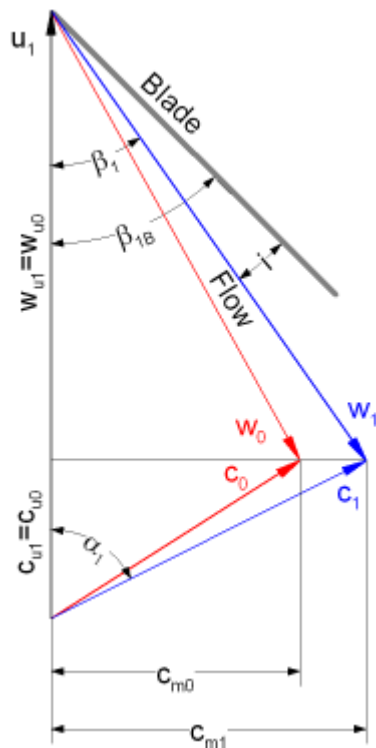
Problem	Possible solutions
Blade angles are updated automatically. Therefore geometry modifications are possible.	
Blade angles are updated automatically if any input parameters are modified.	To fix the blade angles you could uncheck the "Automatic" calculation. Then you have to manually start the calculation if required.
Blade angles are not updated automatically. Therefore the design could be not up-to-date.	
Blade angles 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.
Swirl gradient violates Euler equation. Check blade angles and velocity triangles.	
$c_{u2} \cdot r_2$ is lower than $c_{u1} \cdot r_1$ (turbines: $c_{u2} \cdot r_2$ is higher than $c_{u1} \cdot r_1$) resulting in energy transmission in the wrong direction (Euler equation of turbomachinery).	Recalculate and/or check blade angles β and flow angles α at leading and trailing edge.
B1/2 (leading/trailing edge) is larger than warning level.	
Blade angle difference (highest - lowest value) at all spans exceeds the warning level. The resulting blade could be highly twisted.	Check the resulting blade shape and avoid high blade angle differences on spans if possible.
Blade angles B1/2 cannot be determined. Thermodynamic state could not be calculated. Check main dimensions, meridional shape or global setup. [for compressors and turbines only]	
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 ^[86] (e.g. decrease mass flow).
B1/2 (leading/trailing edge) is larger than error level.	
Blade angle difference (highest - lowest value) at all spans exceeds the error level. Blade design	Decrease the blade angle differences on spans.

Problem	Possible solutions
based on these extreme values makes no sense.	

7.3.1.2.1 Inlet triangle

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](#)^[244]) are different only due to blockage of the flow channel by blades (τ_1) in section 1. This results in an increased meridional velocity c_m .



$$\tan \beta_1 = \frac{c_{m1}}{w_{u1}}$$

$$c_{m1} = c_{m0} \tau_1$$

$$\tau_1 = \frac{t_1}{t_1 - \sigma_1} \quad \text{with} \quad t_1 = \frac{\pi d_1}{z}, \quad \sigma_1 = \frac{s_1}{\sin \beta_{1B}}$$

$$c_{m0} = Q / (\pi d_1 b_1)$$

$$w_{u1} = u_1 - c_{u1}$$

$$u_1 = \pi d_1 n$$

$$c_{u1} = c_{uS} \frac{r_s}{r_1} = u_s (1 - \delta_r) \frac{r_s}{r_1} \quad (\text{const. inflow swirl})$$

Selected blade angle β_{1B} does only indirectly influence the velocity triangle due to blade blockage.

Differences between selected blade angle β_{1B} and flow angle β_1 is referred as the incidence angle: $i = \beta_{1B} - \beta_1$

$$i = \beta_{1B} - \beta_1$$

In general an inflow without any incidence is intended ($i=0$). If $i \neq 0$ the flow around the leading edge shows high local velocities and low static pressure:

$i > 0$: $\alpha_1 < \alpha_{1B} \rightarrow$ stagnation point on pressure side

$i < 0$: $\alpha_1 > \alpha_{1B} \rightarrow$ stagnation point on suction side

A small incidence angle i can be profitable for best efficiency point. Calculation of α_{1B} inside CFturbo gives inflow without incidence.

Typical inlet blade angles are:

Pumps, Ventilators	$\alpha_{1B} < 40^\circ$ due to best efficiency
Pumps	α_{1B} as small as possible due to cavitation; with regard to efficiency not smaller than $15 \dots 18^\circ$
Compressors	optimal blade angle α_{1B} is about 30°

If the radius of leading edge varies from hub to shroud the blade angle α_{1B} does not remain constant. A higher radius on shroud results in a lower value for α_{1B} - the blade is curved on leading edge.

Possible warnings

Problem	Possible solutions
Leading edge blade angle $\alpha_{B1} > xx^\circ$ [pumps, ventilators only]	
Unusual high inlet blade angles. Small inlet angles are typical for pumps and ventilators.	Too high values indicate too small inlet cross section. Increase leading edge dimensions (Main dimensions ^[244])
Leading edge blade angle $\alpha_{B1} < xx^\circ$	
Unusual low inlet blade angles.	Too small inlet angles indicate too high inlet cross section. Decrease leading edge dimensions (Main dimensions ^[244])
Vaned Stator downstream swirl differs significantly from defined value at cu,cm specification [axial turbines only]	

Problem	Possible solutions
If a vaned stator is located prior the rotor, its blade angles might yield circumferential velocities that are significantly different from those defined by cu, cm specification ^[460] .	Adjust the precursor stator trailing edge blade angles manually or by using the soft button "Set TE" in the blade properties ^[503] of the stator.
A reasonable thermodynamic state could not be calculated @LE. Consider change of blade angles or thickness, main dimensions or global setup. [for compressors and turbines only]	
The geometry does not allow for the establishment of a physically valid state. E.g. the mass flow is too high.	Adjust the leading edge blade angles or thickness values or main dimensions ^[244] or the global setup ^[86] (e.g. mass flow or inlet conditions).
The blade angles are not within the valid range.	
Usage of CFturbo is limited to inlet angles between 0° and 180°.	Blade angle calculation is impossible (see below) or adjust unsuitable user input for blade angles.
βB indeterminate. It's not possible to determine blade angle βB.	
Blade angle calculation failed.	Check input values and geometry.

[\[Turbine rotors only \]](#)

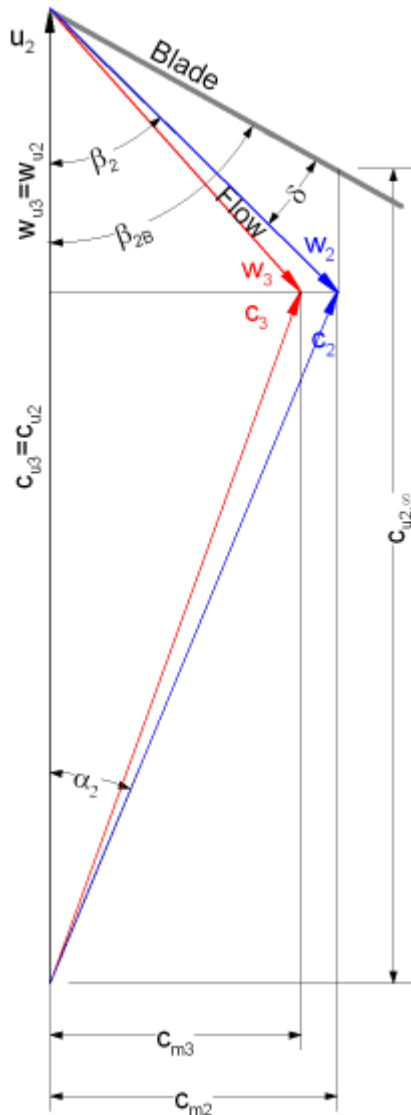
In case of **turbines** the calculation of the incidence by [Aungier](#)^[400] can be used.

According to decreased energy transmission the slip coefficient γ is defined:

$$\gamma = 1 - \frac{c_{u1\infty} - c_{u1}}{u_2}$$

7.3.1.2.2 Outlet triangle

The outlet triangle is determined by geometrical dimensions of flow channel and selected blade angle β_B . The blade angle β_B strongly affects the transmission of energy in the impeller therefore it has to be chosen very carefully.



Similar to the inlet the velocity triangles in cross sections 2 and 3 are different due to blockage of the flow channel by blades τ_2 in section 2.

$$\tan \beta_2 = \frac{c_{m2}}{w_{u2}}$$

$$c_{m2} = c_{m3} \tau_2$$

$$\tau_2 = \frac{t_2}{t_2 - \sigma_2} \quad \text{with} \quad t_2 = \frac{\pi d_2}{z}, \quad \sigma_2 = \frac{s_2}{\sin \beta_{2B}}$$

$$c_{m3} = Q / (\pi d_2 b_2)$$

$$w_{u2} = u_2 - c_{u2}$$

$$u_2 = \pi d_2 n$$

$$c_{u2} = \frac{Y / \eta_h + u_1^2 (1 - \delta_r)}{u_2}$$

$$\text{from: } \tilde{Y} = \frac{Y}{\eta_h} = u_2 c_{u3} - u_1 c_{u0}$$

For determination of β_{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 β_{2B} . The flow angle β_2 is always smaller than blade angle β_{2B} due to the slip velocity. This difference is called deviation angle α_2 :

The deviation angle should not exceed $10^\circ \dots 14^\circ$, in order to limit increased turbulence losses by asymmetric flow distribution.

A reduced flow angle β_2 results in smaller circumferential component of absolute speed c_{u2} , which is - according to Euler's equation - dominant for the transmission of energy. Blade angle β_B is estimated by $c_{u2,\infty}$ 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 CFturbo (not for **Turbines**):

- [GÜLICH/ WIESNER](#) ³⁹⁹
- [AUNGIER/ WIESNER](#) ⁴⁰⁰
- [PFLEIDERER](#) ⁴⁰¹
- [VON BACKSTROEM](#) ⁴⁰²
- [Specific definitions](#) ⁴⁰³

Blade angle β_B 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 β_B exist due to optimal efficiency:

Pumps	15°...45°, commonly used 20°...27°
Ventilators	not higher than 50°
Compressors	35°...50°, unshrouded impellers up to 70°...90°
Turbines	radius dependent, see sine rule ⁴²⁹

Radial machines - except for turbines - with low specific speed n_q usually have similar values for β_B . The blades for this type of impellers are often designed with a straight trailing edge ($\beta_B = \text{const.}$). For turbine rotors the radii along the trailing edge from hub to shroud are very different, resulting in very different values for β_B and twisted blades.

Possible warnings

Problem	Possible solutions
Trailing edge blade angle $\beta_{B2} < \alpha^\circ$	

Problem	Possible solutions
Unusual low outlet blade angles	Too small outlet angles indicate too high outlet cross section. Decrease trailing edge dimensions (Main dimensions ^[244])
Deviation (slip) between blade and flow is pretty high. (pumps, ventilators, compressors only)	
Unusual high deviation (slip) between blade and flow direction at outlet. This indicates too high blade loading.	Possible solutions could be: increase the impeller diameter (Main dimensions ^[244]), increase the number of blades, increase meridional blade length (Meridional contour ^[338]), select a different slip model
Trailing edge blade angle $\beta_{B2} > xx^\circ$.	
Unusual high blade angles at trailing edge. This can be caused by overloading the impeller.	Increase trailing edge dimensions (Main dimensions ^[244]) and/or the slip coefficient .
A reasonable thermodynamic state could not be calculated @TE. Consider change of blade angles or thickness, main dimensions or global setup. [for compressors and turbines only]	
The geometry does not allow for the establishment of a physically valid state. E.g. the mass flow is too high.	Adjust the trailing edge blade angles or thickness values or main dimensions ^[244] or the global setup ^[86] (e.g. mass flow or inlet conditions).
Blade angles are not within the valid range.	
Usage of CFturbo is limited to blade angles between 0° and 180°.	Blade angle calculation is impossible (see below) or adjust unsuitable user input for blade angles.
No possibility to determine Blade angles β_B.	
Blade angle calculation failed.	Try to increase the impeller diameter d_2 or outlet width b_2 and/or the slip coefficient .
Deviation (slip) between blade and flow is too high.	

Problem	Possible solutions
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.	Possible solutions could be: increase the impeller diameter (Main dimensions ^[244]), increase the number of blades, increase meridional blade length (Meridional contour ^[338]), select a different slip model

7.3.1.2.2.1 Slip coefficient by GÜLICH/ WIESNER

Outflow (slip) coefficient 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.

Wiesner developed an empirical equation for the estimation of outflow coefficient:

$$\gamma = 1 - \frac{\sqrt{\sin \beta_{2B}}}{z^{0.7}}$$

Gulich modified this formula by two additional correction factors:

$$\gamma = f_1 \left(1 - \frac{\sqrt{\sin \beta_{2B}}}{z^{0.7}} \right) k_w$$

with the correction factors:

$$f_1 = \begin{cases} 0.98 & \text{for radial impellers} \\ 1.02 + 1.2 \cdot 10^{-3} (\eta_q - 50) & \text{for mixed-flow impellers} \end{cases}$$

$$\varepsilon_{\text{Lim}} = \exp\left(-\frac{8.16 \sin \beta_{2B}}{z}\right)$$

Circumferential component of blade congruent flow can be calculated as follows:

$$c_{u2\infty} = c_{u2} + (1-\gamma)u_2$$

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:

$$\gamma = 0.5(\gamma_{\text{Hub}} + \gamma_{\text{Shroud}})$$

The switch between radial and mixed-flow calculation of the correction factor f_1 is done by:

$$f_1 = \max\left(0.98, 1.02 + 1.2 \cdot 10^{-3} (n_q - 50)\right)$$

7.3.1.2.2.2 Slip coefficient by AUNGIER/ WIESNER

Outflow (slip) coefficient γ 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.

Aungier adjusted [Wiesner's](#)^[399] original empirical equation for the estimation of outflow coefficient:

$$\gamma = 1 - \frac{\sqrt{\sin \beta_{2B}}}{z^{0.7}}$$

The limiting radius ratio r_{Lim} is given by:

The slip factor is corrected for radius ratios $r/r_2 > r_{\text{Lim}}$ with:

$$\gamma_{\text{cor}} = \gamma \left(1 - \left(\frac{\varepsilon - \varepsilon_{\text{Lim}}}{1 - \varepsilon_{\text{Lim}}} \right)^{\sqrt{\beta_{2B/10}}} \right)$$

[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.

$$z_{\text{cor}} = z_{\text{mB}} + z_{\text{sB}} \frac{l_{\text{sB}}}{L_{\text{mB}}}$$

Circumferential component of blade congruent flow can be calculated as follows:

$$c_{u2\infty} = c_{u2} + (1 - \gamma)u_2$$

7.3.1.2.2.3 Slip coefficient by PFLEIDERER

Reduced energy transmission is expressed by decreased output coefficient p:

$$p = \frac{\tilde{\gamma}_{\infty}}{\tilde{\gamma}} - 1$$

This coefficient can be empirically calculated in dependence of experience number ψ' :

$$p = \psi' \frac{r_2^2}{zS}$$

$$S = \int_{r_1}^{r_2} r dx$$

static moment from leading to trailing edge

experience number

experience number a:

Radial impeller with guided vanes a = 0.6

with volute $a = 0.65 \dots 0.85$

with plain diffuser $a = 0.85 \dots 1.0$

Mixed flow/axial impeller $a = 1.0 \dots 1.2$

(the numbers are valid for sufficiently high Re; ψ' strongly grows with small Re)

More descriptive is the decreased output factor k_L :

$$k_L = \frac{1}{1+p} = \frac{\tilde{Y}}{\tilde{Y}_\infty} = \frac{\Delta(rc_u)}{\Delta(rc_u)_\infty} \quad (k_L=1: \text{ for flow congruent to blade})$$

Circumferential component of the flow, which is congruent to blade, can be calculated as follows:

$$c_{u2\infty} = \frac{c_{u2}}{k_L} - \frac{r_1^2}{r_2} \left(\frac{1}{k_L} - 1 \right) 2\pi n (1 - \delta_r)$$

Now the outflow (slip) coefficient γ according to [Wiesner](#)^[399] can be calculated:

$$\gamma = 1 - \frac{c_{u2\infty} - c_{u2}}{u_2}$$

7.3.1.2.2.4 Slip coefficient by VON BACKSTROEM

Outflow (slip) coefficient γ 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](#)^[569] 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 sol is the solidity defined by:

$$\text{sol} = \frac{1 - \varepsilon \cdot z}{2\pi \cdot \sin(\beta_{2B})}$$

The limiting radius ratio $r_{1\text{Lim}}/r_2 = 0.5$, the radius ratio $r_1/r_2 = \text{Max}(r_1/r_2, r_{1\text{Lim}}/r_2)$. The constant $F_0 = 5$.

7.3.1.2.2.5 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 :

number of blades	slip coefficient
1	0.48 ... 0.6
2	0.53 ... 0.65
3	0.67 ... 0.75

Inducer pumps (GÜLICH)

For inducer pumps the deviation angle depends on the blade angles on leading and trailing edge and the solidity.

$$\delta [^\circ] = \left(2^\circ + \frac{\beta_{2B} [^\circ] - \beta_{1B} [^\circ]}{3} \right) \cdot \left(\frac{t}{L} \right)^{1/3}$$

7.3.1.2.2.6 B2 calculation details

Circumferential component of blade congruent flow can be calculated as follows:

$$c_{u2\infty} = c_{u2} + (1 - \gamma)u_2$$

The slip coefficient is a function of blade angle, number of blades and the meridional geometry (see e.g. [GÜLICH/WIESNER](#)^[399] etc.):

$$\gamma = f(\beta_{2B}, z, \dots)$$

The relation of the blade angle to the velocity components is:

$$\tan(\beta_{2B}) = \frac{c_{m2}}{u_2 - c_{u2\infty}}$$

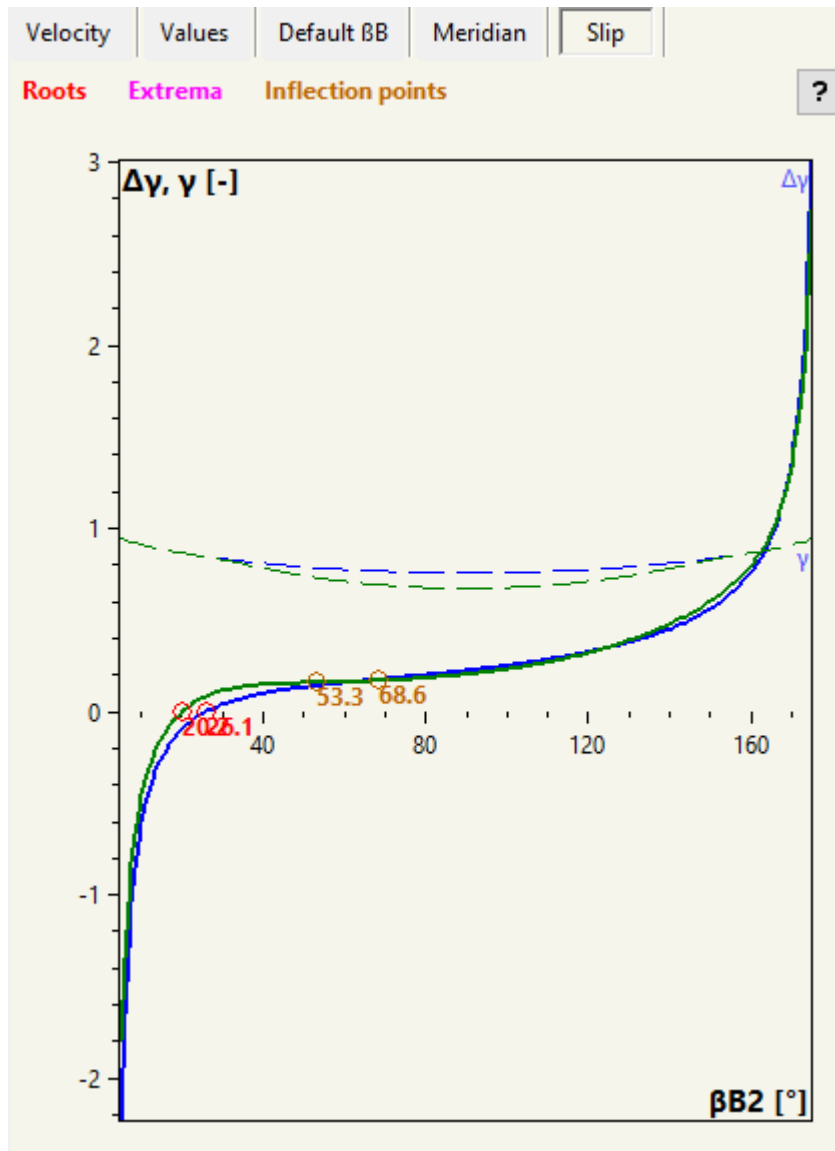
A swirl has to be produced by the impeller in accordance to the design specific work (here without pre-swirl) with the Euler equation:

$$Y = c_{u2} \cdot u_2$$

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:

$$0 = \Delta\gamma = \frac{c_{m2}}{u_2 \tan(\beta_{2B})} - \frac{Y}{u_2} - f(\beta_{2B}, z, \dots)$$

One can test this equation with different values of β_{2B} and will get a function of the form $\gamma = f(\beta_{2B})$. 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

? IMPELLER | Blade mean lines

The blade mean lines are designed on the number of meridional flow surfaces which were determined in [Blade properties](#) ^[371].

Depending on the selected blade shape (see [Blade properties](#) ^[371]) 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](#)^[389]. 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](#)^[412] [Views](#)^[412]

Design mode

Select the currently available mode to design the blade mean lines.

→ See [Design mode](#)^[432]

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 (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](#)^[198]. 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 polyline at which the stacking position is applied is determined by the - 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](#)^[70] (buttons above the columns).

Angular positions

Wrap angle $\Delta\varphi$ 109.0 °

Overlapping $\Delta\varphi/t =$ 2.119

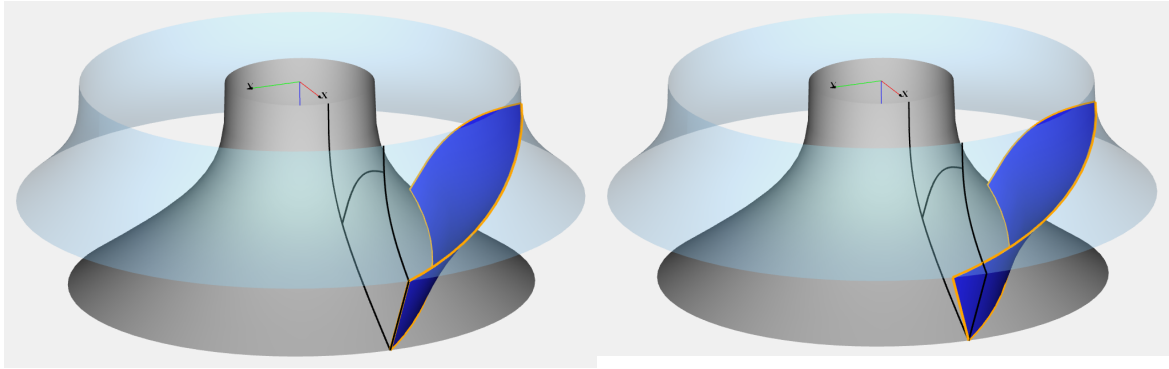
Stacking position 0 100
 ↓ 100 %
 LE TE

<< Set progression >>

	Stacking angle	Wrap angle
j	φ [°]	$\Delta\varphi$ [°]
1	0.0	137.4
2	2.0	121.5
3	4.0	111.6
4	6.0	104.0
5	8.0	95.7
6	10.0	83.7

Rake angle

The rake angle 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 $= -20^\circ$ (right).

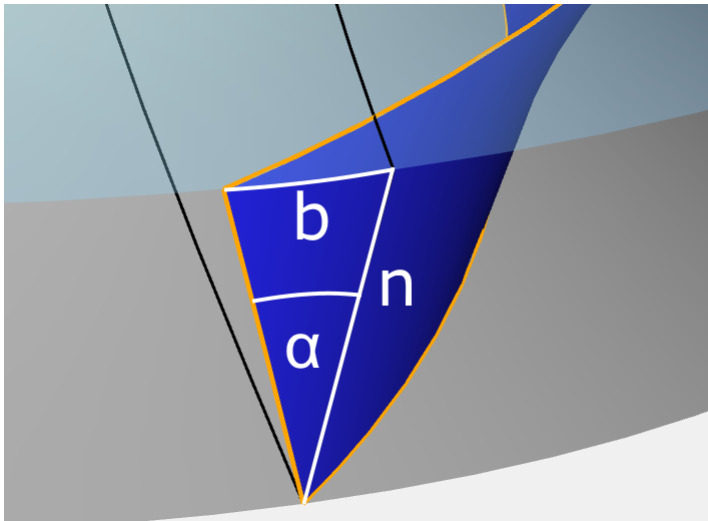


Blade with rake angle = 0°

Blade with rake angle = -20°

The rake is determined by (see picture below):

$$\tan(\alpha) = \frac{b}{n}$$



The rake angle can be set directly when the [blade shape](#)^[375] is either Free-form 3D or Ruled Surface 3D. In case of [Design mode](#)^[432] = **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.

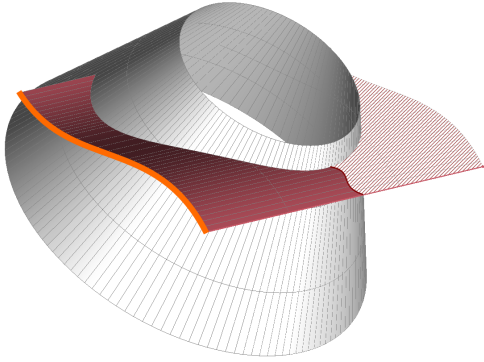
Rake angle

☐ at leading edge r °

☒ at trailing edge r °

Possible warnings

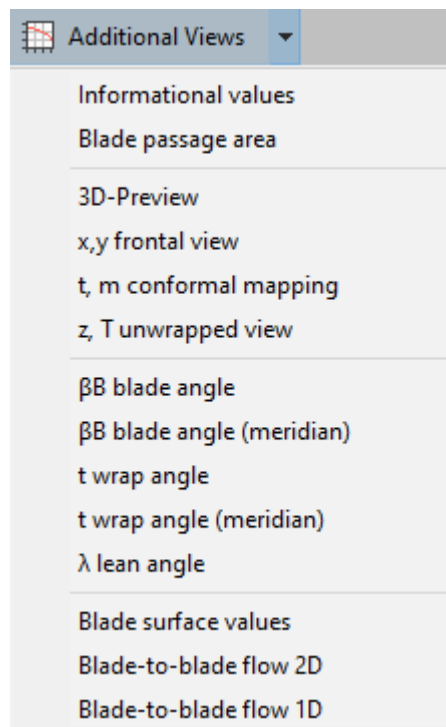
Problem	Possible solutions
Blade angles and blade extensions could lead to unusual blade shapes.	
Blade angles and blade extensions could lead to non-feasible blade shapes.	
<p>The values of the blade angles B_1, B_2 and the meridional and tangential blade extension most likely result in an abnormal or strange blade shape.</p> <p>To avoid any subsequent problems such mean line shapes are blocked.</p>	<p>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.</p> <p>You can</p> <ul style="list-style-type: none"> a) modify the blade wrap angle (checking the blade overlapping) or b) modify the blade angles B_1 and B_2 (probably the main dimensions have to be adapted)
$B1/2$ (leading/trailing edge) is higher than warning level	
Blade angle difference (highest - lowest value) at all spans exceeds the warning level. The resulting blade could be highly twisted.	Check the resulting blade shape and avoid high blade angle differences on spans if possible.
$B1/2$ (leading/trailing edge) is higher than error level	
Blade angle difference (highest - lowest value) at all spans exceeds the error level. Blade design based on these extreme values makes no sense.	Decrease the blade angle differences on spans.
Blade calculation failed due to boundary conditions and constraints.	
Projection of the design mean line onto the other spans fails for this blade shape.	Decrease wrap angle.


Problem	Possible solutions
	
High tangential leading edge sweep angle requires high number of spans.	
<p>Leading edge sweep angle (tangential difference between hub and shroud meanline at LE) is high. This curved shape requires a minimal number of spans to avoid abnormal or strange blade shape.</p> <p>A warning and an error level exist for this test.</p>	<p>Increase the number of spans - see Blade angles ^[389].</p>
Blade edge exceeds the meridional boundaries.	
<p>The meanlines of inner blade spans are crossing the meridional extents at leading or trailing edge.</p> <p>This is only possible for ruled surface blades with more than 2 spans.</p>	<p>Change meridional position of leading/ trailing edge or reduce number of spans to 2.</p>
Blade angles B1 and/or B2 different compared to blade properties.	
<p>Current blade angle values deviate from the specified values in Blade properties ^[371]. This is possible for imported geometry only.</p>	<p>Check imported m,t-curves or -curves and compare with specified values at leading and trailing edge. The values resulting from the current meanlines are displayed in "Additional views/ Informational values/ Blade angle B".</p>
<p>Overlapping of adjacent blades might be too low. Overlapping of adjacent blades might be too high.</p>	
<p>Min. wrap angle = Pitch (2 / number of blades)</p>	<p>Modify the blade wrap angle and/ or the number of blades (see Blade angles ^[389]).</p>


Problem	Possible solutions
Max. wrap angle = $F \cdot \text{Pitch}$ with $F=3.0$ for compressors, else $F=2.5$	
Coupling partially deactivated. Blade surface deformation can occur.	
<p>The mean lines are currently not linearly coupled, which can result in deformed blade surfaces.</p> <p>Either linear coupling has been deactivated or it is impossible because of highly deviating blade angle values.</p> <p>The warning occurs because the intersection of B_2 line and intersection line for one or more mean lines cannot be determined. Usually this has one of the following causes:</p> <ul style="list-style-type: none"> a) It is geometrically impossible to determine this intersection (approximate parallel lines). b) The intersection is not between the points of hub and shroud mean line. c) The point of intersection is too close to the endpoints of the mean line (lower than 5%). 	<p>Activate linear coupling if it is deactivated.</p> <p>Homogenize B_2 blade angle values (see Blade properties ^[371]).</p>
Curvature very high. Swirl change might not be as high as intended.	
<p>One or more mean lines do not change their direction smoothly enough in the m,t coordinate system resulting in partially high curvature (B gradient).</p> <p>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.</p>	<p>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.</p> <p>Very high difference between B_1 and B_2 can make the solution of the problem more difficult and could require a higher wrap angle.</p>


7.3.2.1 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 ⁴¹⁴ .
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](#)^[86] 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](#)^[501] and [outlet](#)^[502] of the main dimensions.

[Only compressible fluids and stators]

3D-Preview

[3D model](#)^[225] 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_H 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](#)^[416]

z, T unwrapped view

Peripheral projection of all mean lines (z = axial coordinate, T = circumferential coordinate).

[Axial impellers only]

B blade angle

B progression along every mean line.

Too high local extreme values should be avoided if possible.

Additionally 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](#)^[424].

B blade angle (meridian)

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.
lean angle	<p>Distribution of the lean angle . The blade lean angle can be manipulated only indirectly.</p> <p>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^[389] 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.</p> <p>→ See Blade lean angle^[417]</p>
Blade surface values	→ See Blade surface values ^[419]
Blade-to-blade flow 1D	→ See Blade-to-blade flow 1D ^[424]
Blade-to-blade flow 2D	→ See Blade-to-blade flow 2D ^[426]

7.3.2.1.1 Informational values

The tables contains additional values for information:

Radial diffuser [Stator type "Radial diffuser" only]

Various values to verify the quality of the diffuser design.

→ see [Mean line](#)^[507] 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 (α_1) and trailing edge (α_2).

→ see [Blade lean angle](#) ^[417]

Blade loading [Pump impeller only]

Blade loading estimation with lift coefficient (Guelich):

$$\zeta_a = \frac{\pi \psi}{z L_{Bl}/d_2 \sqrt{\varphi_2^2 + [1 - \psi/4 (1 - \sin \varepsilon/z)]^2}} \quad \zeta_{a,max} = 0.9$$

and with the effective blade loading (Gülich):

$$\xi_{eff} = \frac{2\pi \cdot \psi \cdot u_2}{\eta_H \cdot z \cdot L_{Bl}/d_2 (w_1 + w_2)} \quad \xi_{range} = \left(\frac{40}{nq} \right)^{0.77} \pm 15\%$$

Blade angle

Table with the blade angles β calculated in the [Blade properties](#) ^[371] dialog or computed due to simple blade shapes.

Blade angle in x-y

Table with the blade angles of the frontal view β_{xy} .

In case of strictly radial impellers these values are consistent with the blade angles β .

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](#) ⁴²⁹

Blade solidity

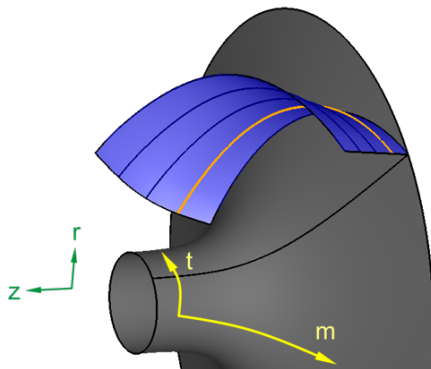
Ratio between blade length (in different definitions) and pitch ($\pi d_2/z$).

Other information

Table with:

- chord length l
- mean line length l_{ML}
- resulting angles of overlapping φ_B of 2 neighboring blades
- incidence angle i for hub and shroud

7.3.2.1.2 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.

Both quantities are created by the reference of absolute distances in meridional (M) and tangential direction (T) to the local radius r :

$$dm = \frac{dM}{r} \quad dt = \frac{dT}{r}$$

$$\tan\beta = \frac{dm}{dt}$$

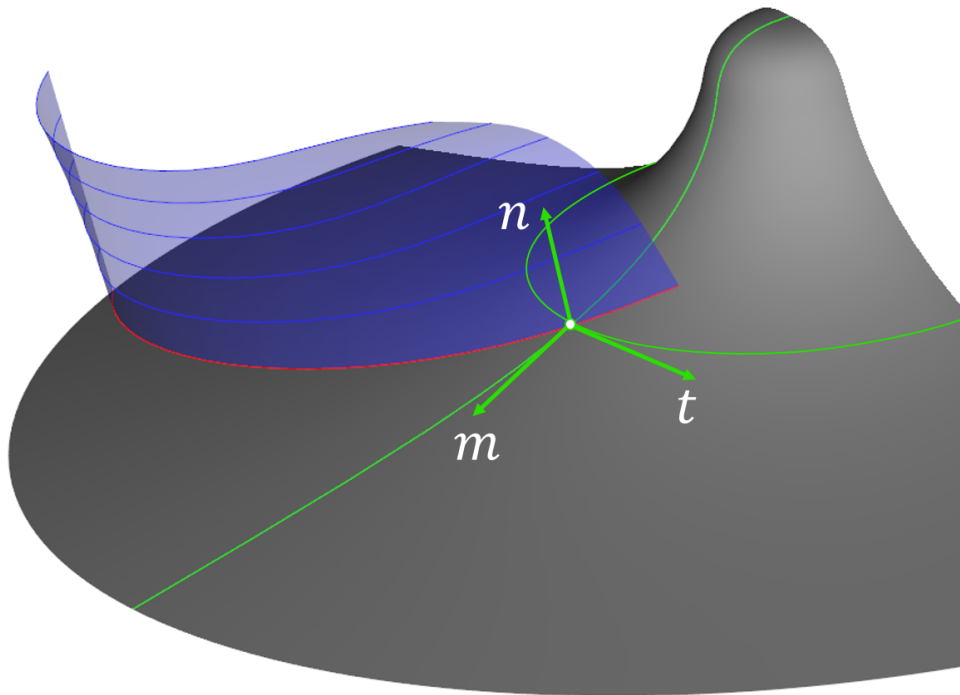


Special display option for splitter blades:

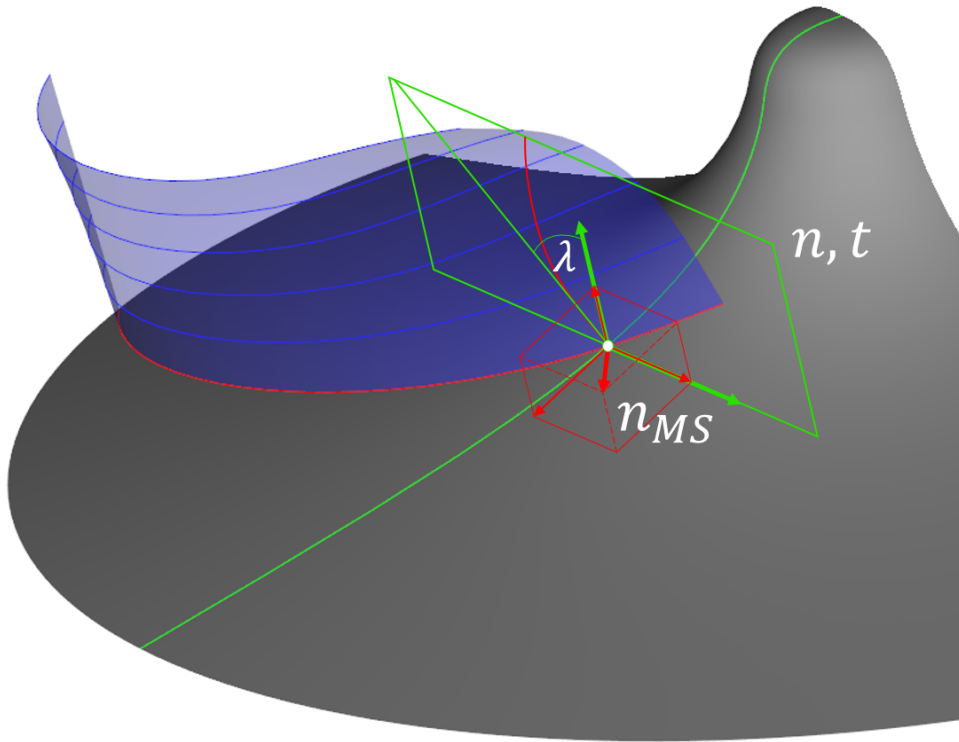
With "**Splitter blade relative to main blade**" checked, corresponding mean lines (splitter and main) have the same maximum m -value. Otherwise all mean lines have the same maximum m -value as the main blade's hub mean line.

7.3.2.1.3 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 **blade angle** β 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** β only depends on the mean line itself, **blade lean angle** is measured in n,t -plane and gives an information about the slope between mean lines in circumferential direction.

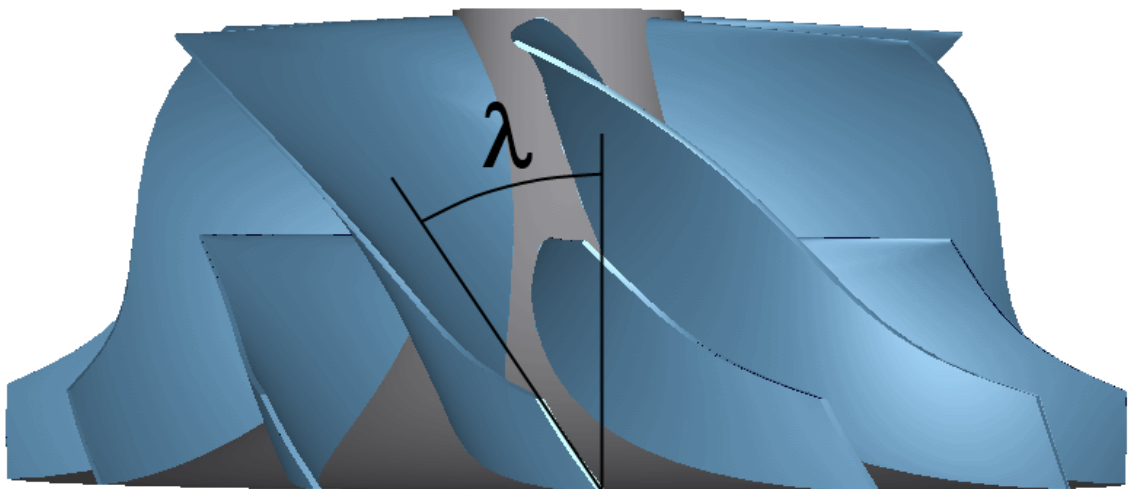


Using the surface normal direction n_{MS} on mean surface the **blade lean angle** is calculated from the ratio of the portion in n and t :

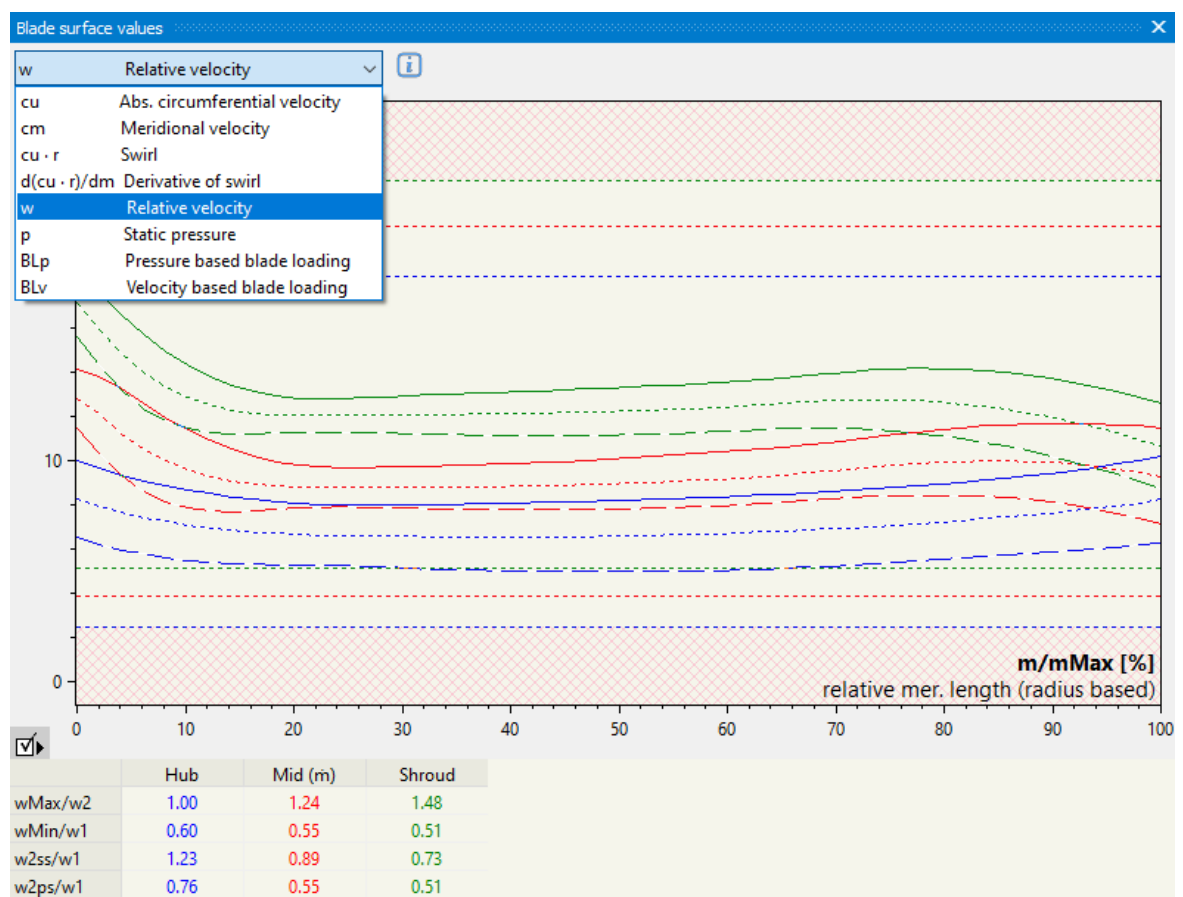
$$\lambda = \arctan\left(-\frac{n_{MS,n}}{n_{MS,t}}\right)$$

With an example of a compressor some means for the manipulation of the blade lean angle are given:

- $_1 \uparrow$: [blade angle](#)^[389] $B_1 \downarrow$
- $_1 \uparrow \downarrow$: move [second Bezier point](#)^[406] at leading edge
- $_1 \uparrow$: [wrap angle](#)^[406] \uparrow
- $_1 \uparrow$ and enlargement of the curvature: reduction of the meridional extension of the [meridional contour](#)



7.3.2.1.4 Blade surface values

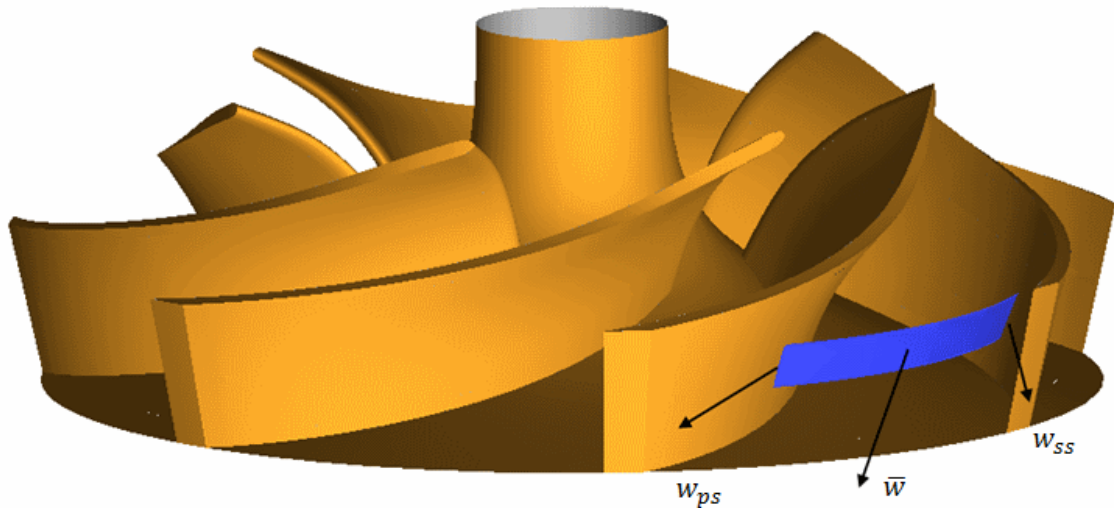


Determination of velocity distribution on impeller blades by Stanitz & Prian ⁵⁶⁸

Stream lines must be known a-priori (see [Meridional flow calculation](#) ³⁵⁶). 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 calculated 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:

$$\bar{w} = \frac{\dot{m}}{\bar{\rho} \cdot A}$$

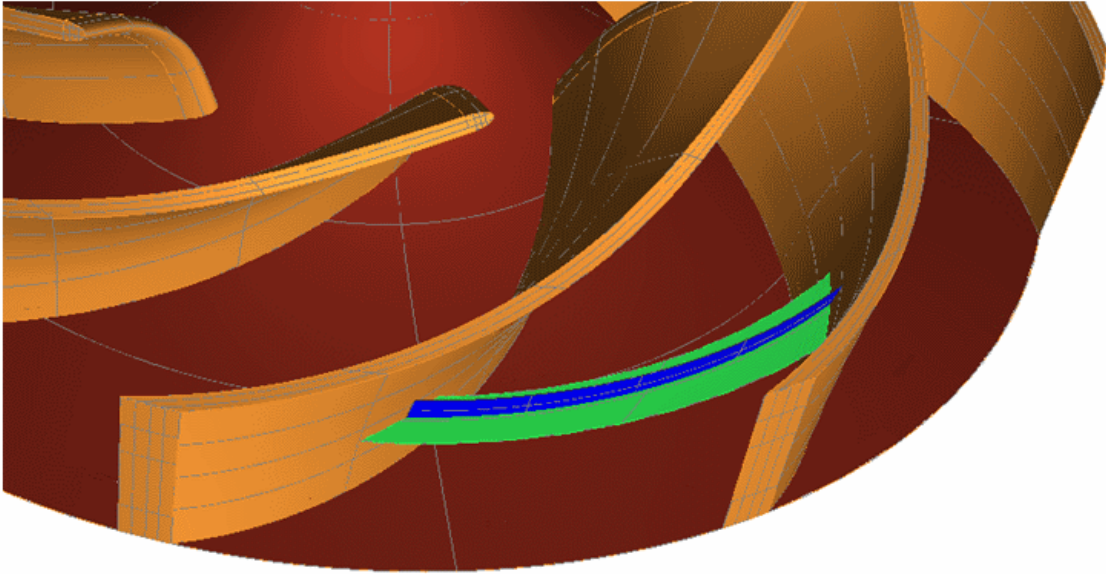
The part mass flow is a function of the entire mass flow, number of blades and number of stream lines. Between two adjacent stream surfaces there is always the same mass flow.



The cross section is determined by stream line distance h , the radius r , the tangential distance between pressure and suction side of two neighboring blades t and by a mean relative flow angle:

$$A = r \cdot \Delta t \cdot \Delta h \cdot \sin(\beta)$$

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:

$$w_{ss} = \frac{\sin(\beta_{ps}) \sin(\beta_{ss})}{\sin(\beta_{ps}) + \sin(\beta_{ss})} \left(\frac{2\bar{w}}{\sin(\beta_{ps})} + u \cdot (\cot(\beta_{ps}) - \cot(\beta_{ss})) + \frac{\partial(c_u \cdot r \cdot \Delta t)}{\partial m} \right),$$

here u is the local circumferential velocity, c_u is the circumferential component of the absolute velocity, β_{ss} and β_{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:

$$w_{ps} = 2 \cdot \bar{w} - w_{ss}.$$

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:

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 be geometric constellations where the cross section (blue surface in the images above) is too small for the mass flow specified in the [global setup](#)^[86]. 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:

$$h_{ps} - h_{ss} = \frac{2\pi}{n} \cdot c_m \frac{\partial(r \cdot \bar{c}_u)}{\partial m}.$$

The blade loading can be expressed in terms of the pressure difference between suction and pressure side and divided by the total inlet pressure:

$$BLp = \frac{p_{ps} - p_{ss}}{p_{in,total}} = \frac{p_{av}}{p_{in,total}} \left(\frac{2\pi}{n} \cdot c_m \frac{\partial(r \cdot \bar{c}_u)}{\partial m} - c_v (T_{ps} - T_{ss}) \right).$$

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 c_u as well as the average swirl B can also be displayed. Those quantities are determined by:

$$\bar{c}_u = u - \bar{w} \cdot \cos(\beta)$$

Also the [Ackeret](#)^[568] 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 $1.8 \cdot w_2$, whereas the minimum relative velocity shall not be smaller than $0.3 \cdot w_1$ (w_1 and w_2 are the average relative velocities at LE and TE resp.). Those limits (max. and min. velocities) are not displayed for splitter blades.

$$\text{Ackeret} = w_2 / w_1,$$

$$\text{Ackeret}_{\max} = 1.8 = w_{\max \text{SS}} / w_2,$$

$$\text{Ackeret}_{\min} = 0.3 = w_{\min \text{PS}} / w_1.$$

[Compressors and Turbine rotors only]

Mach-number

Mach-number can be displayed both relative as well as absolute.

$$\text{Ma}_w = w / a,$$

$$\text{Ma}_c = c / a.$$

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:

$$p_{\text{cr}} = \pi_{\text{cr}} \cdot p_{\text{tin}},$$

$$T_{\text{cr}} = T_t \cdot \pi_{\text{cr}}^{R/\text{cp}},$$

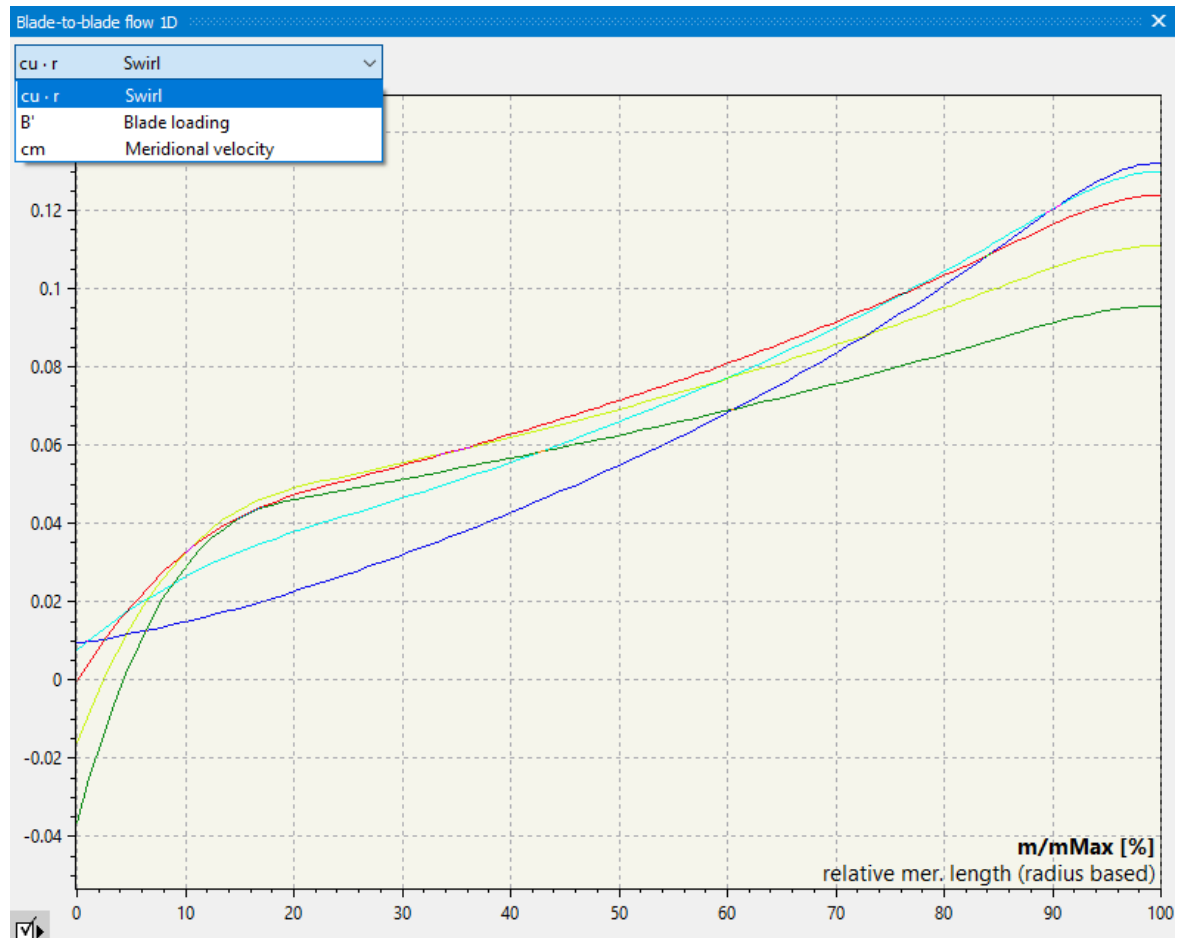
Here π_{cr} is the pressure ratio at which the flow is at sonic speed in the smallest cross section:

$$\pi_{cr} = \left(\frac{2}{\kappa + 1} \right)^{\frac{\kappa}{\kappa - 1}}.$$

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.

7.3.2.1.5 Blade-to-blade flow 1D

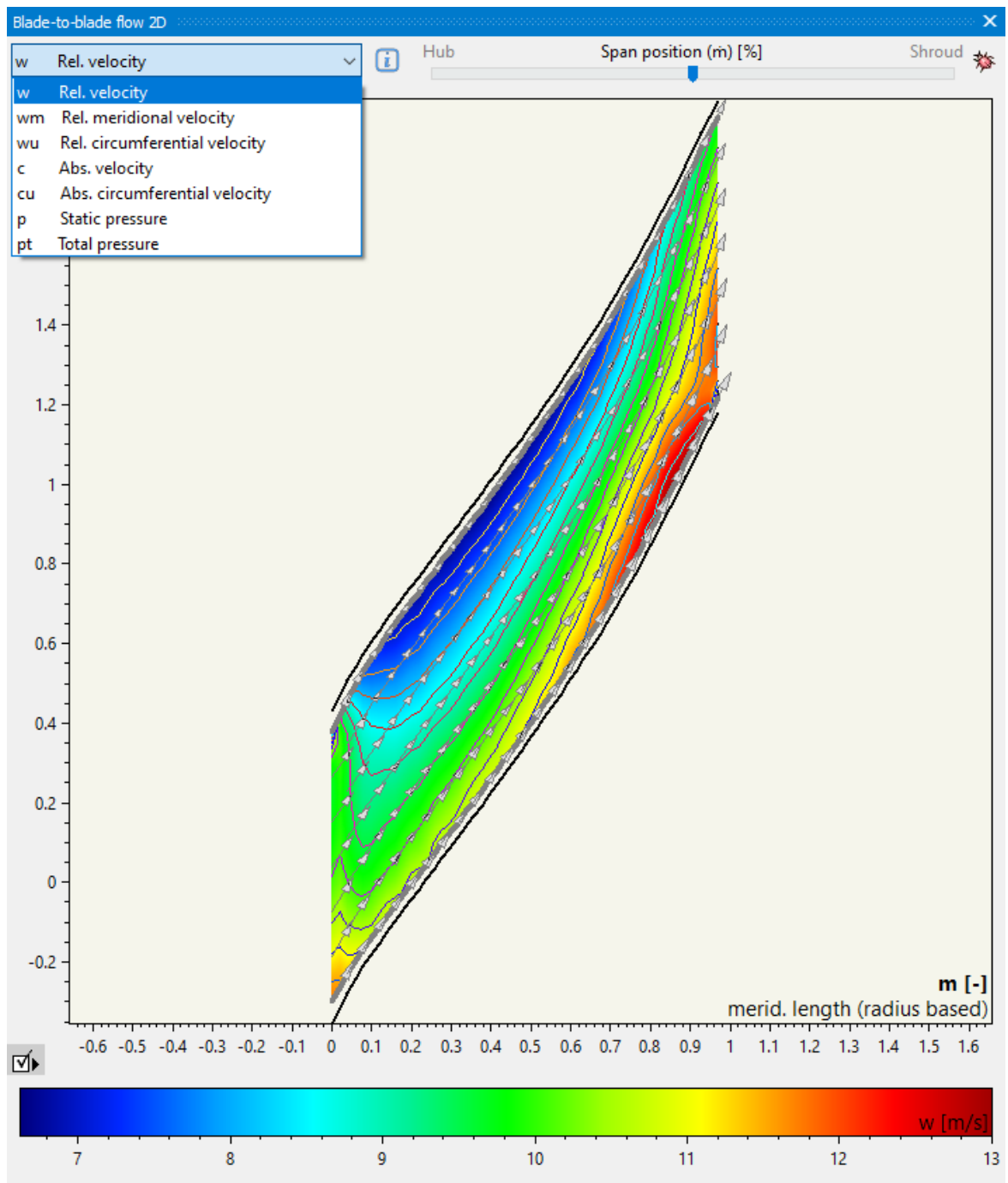


Swirl $c_{u,r}$ and its derivative

c_u -values are calculated based on the assumption that the flow follows the direction given by the blade, i.e. by the blade angles. For radial and mixed-flow impeller this assumption is applied until the Stanitz-radius (see [Stanitz & Prian](#)^[568]). At radii bigger than the [Stanitz-radius](#)^[421] the [slip](#)^[396] 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](#)^[393]) the relative flow is assumed to be blade congruent latest as $m/m_{\max} = 30\%$, depending on the incidence angle. The meridional velocity component c_m is taken from the [meridional flow calculation](#)^[356].

In case the [meridional flow calculation](#)^[356] failed the blade-to-blade flow 1D results cannot be calculated and the diagram will not be available.

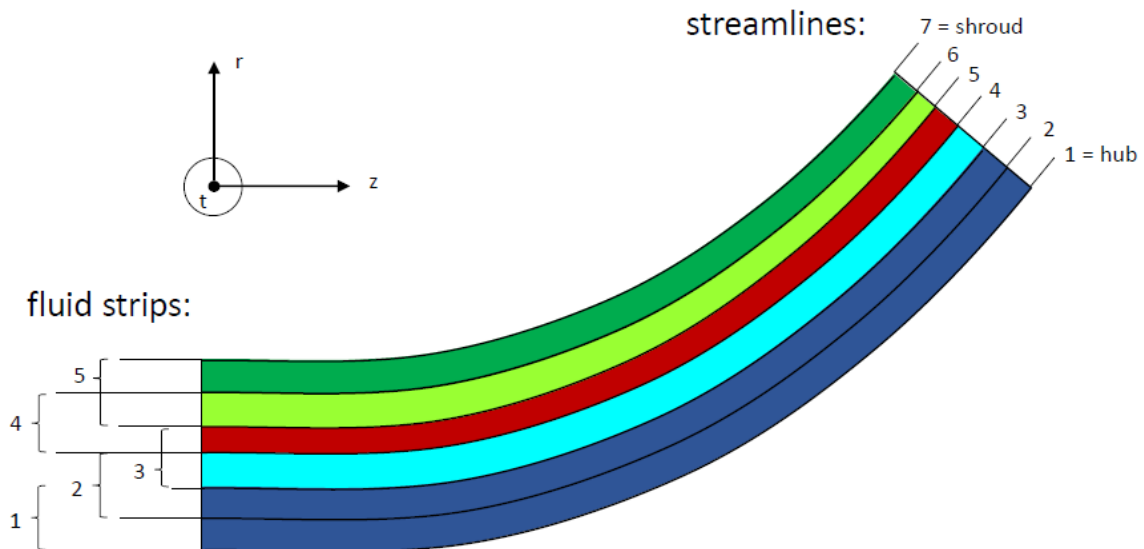
7.3.2.1.6 Blade-to-blade flow 2D



Stream function

Stream lines must be known a-priori (see [Meridional flow calculation](#)^[356]). 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 calculated 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&Prrian](#)^[419] approach here a two-dimensional relative flow is calculated. This equation in m-t co-ordinates reads as:

$$\frac{\partial^2 \psi}{\partial t^2} - \frac{1}{\rho} \frac{\partial \rho}{\partial t} \frac{\partial \psi}{\partial t} + \frac{\partial^2 \psi}{\partial m^2} - \frac{1}{\rho \Delta n} \frac{\partial(\rho \Delta n)}{\partial m} \frac{\partial \psi}{\partial m} = \rho \Delta n \cdot 2\omega \cdot r \frac{\partial r}{\partial m}$$

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:

$$\nabla \times \vec{w} = -2\vec{\omega},$$

$$\nabla \cdot (\rho \vec{w}) = 0$$

Here w is the relative velocity, ω is the rotational speed and ρ is the fluid density. In the equation above 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&Prrian](#)^[419] 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](#)^[86] mass flow.

$$\Psi_{ss} = 0$$

$$\Psi_{ps} = \frac{2}{\text{No. fluidstrips}} \cdot \dot{m}$$

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](#)^[569].

Results

The tangential and the meridional relative velocity component resp. can be calculated by:

$$w_u = \frac{1}{\rho \Delta n \cdot r} \frac{\partial \Psi}{\partial m}$$

$$w_m = \frac{1}{\rho \Delta n \cdot r} \frac{\partial \Psi}{\partial t}$$

The static pressure p_i can be determined by using the constancy of the rothalpy. For incompressible fluids that reads:

$$\frac{p_1}{\rho} + \frac{1}{2} (w_1^2 - u_1^2) = \frac{p_i}{\rho} + \frac{1}{2} (w_i^2 - u_i^2)$$

For compressible fluids the same principle is applied for the specific enthalpy and temperature resp. with the assumption of [perfect gas behavior](#)^[201]. Since the density is already known the static pressure can be calculated using the equation of state $p=f(T, \rho)$.

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.

$$\text{Ma}_w = w/a,$$

$$\text{Ma}_c = c/a.$$

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.

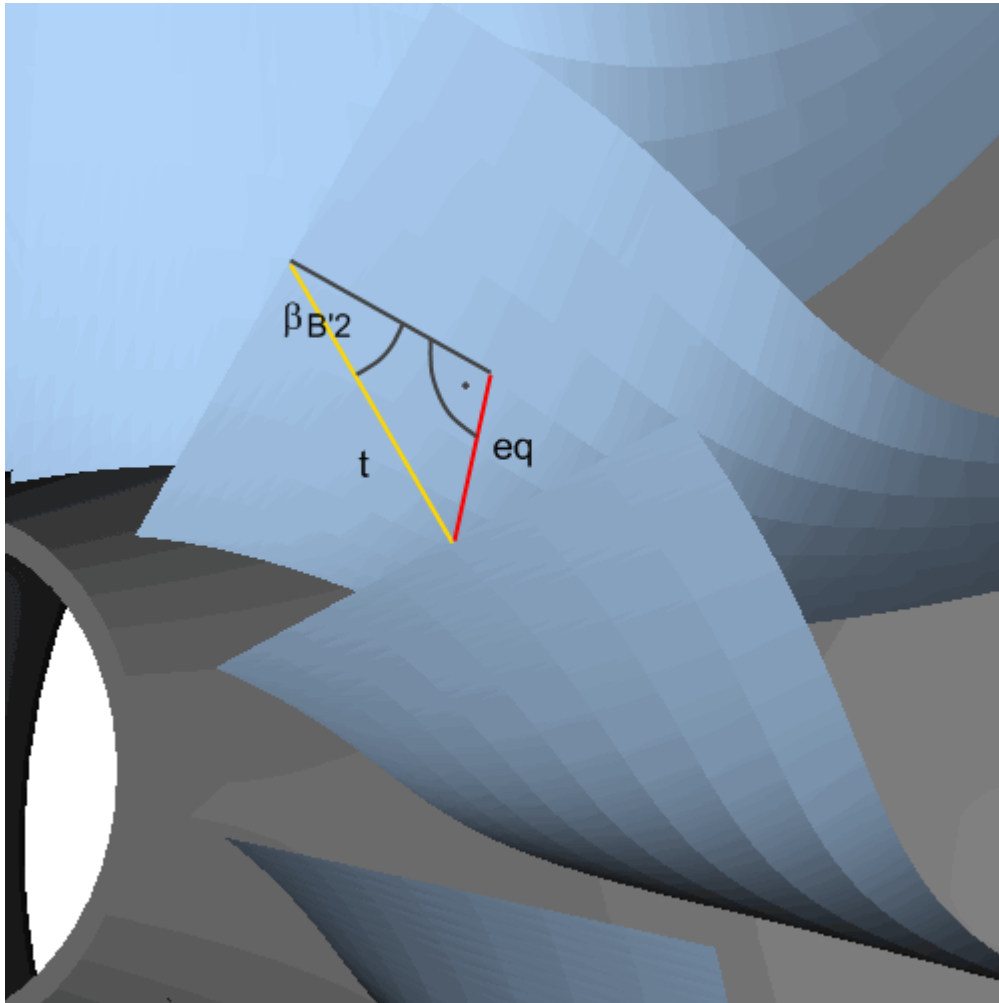
7.3.2.1.7 Sine rule

[Turbine rotors only]

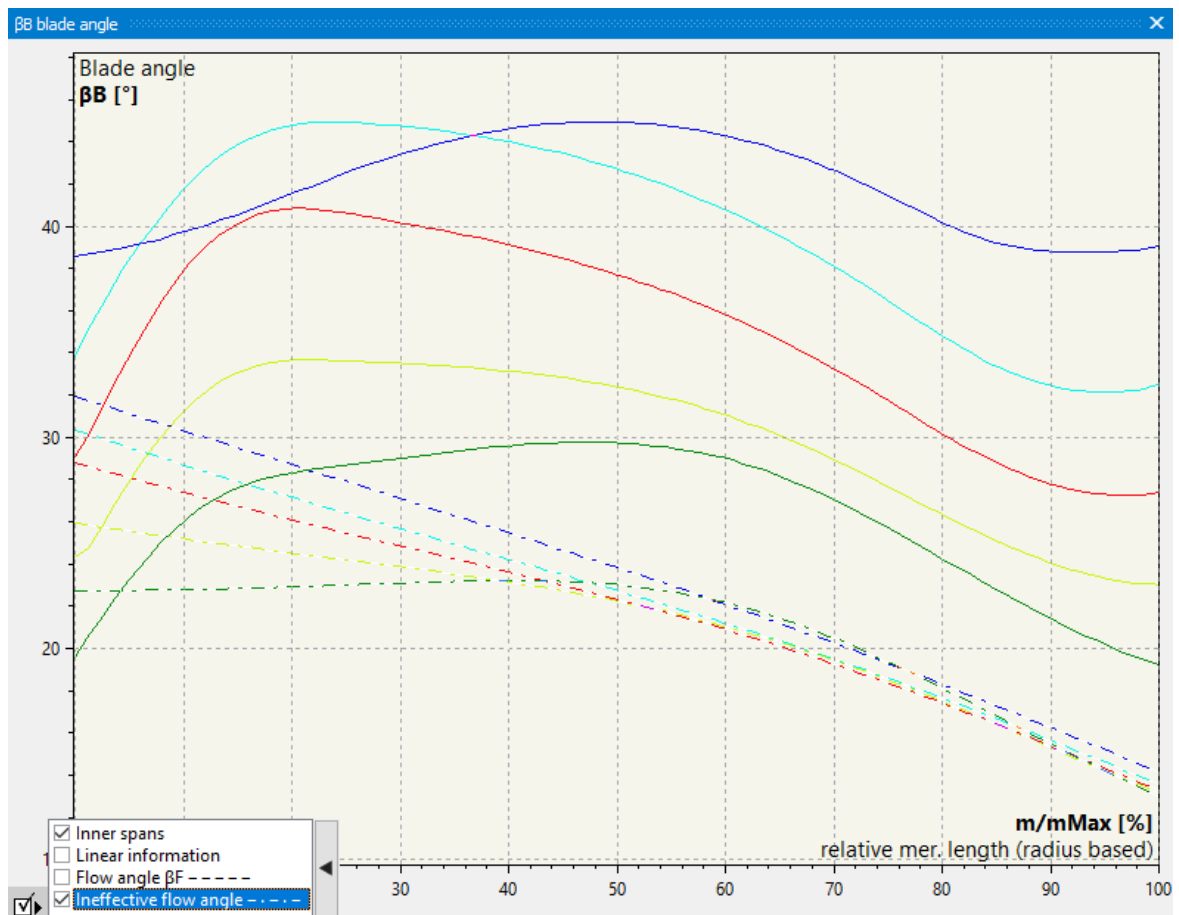
With the help of the sine rule blade angles at the outlet can be evaluated. In accordance to this rule blade angles at the outlet should have almost the same size as the angle that is built by a hypotenuse being the pitch t , and a cathetus (opposite leg) being the smallest distance between two neighboring mean lines eq at a flow surface. If this is the case the outflow can be regarded as almost tangential to the trailing edge.

$$\sin \beta_{B'2} = \frac{eq}{t}$$

This is shown in a picture for a single mean line.



7.3.2.1.8 Blade angles



Blade angles and relative angles

Three different angle distributions can be displayed for each span:

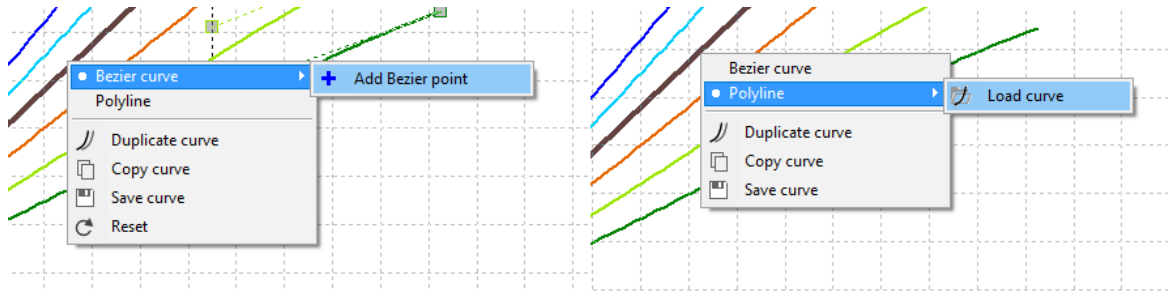
- Blade angles β_B —
- Relative flow angle β_F - - -
- Ineffective relative flow angle β_{FI} - · -

With the specified incidence and deviation angles (see [blade properties](#)^[371]) and the attachment and detachment location (the latter is the [Stanitz-Radius](#)^[421]) 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.

7.3.2.2 Design mode

The mean lines can be designed by alternative methods, which can be selected on panel **Design mode**.

In general the mean lines are represented by 3rd order Bezier curves. The point count can be changed by using the context menu of the mean lines and Bezier curves can be fitted from polylines. Moreover, the curve mode can be switched to polyline to use a user-defined polyline directly. Also, a mean line can be loaded from the [profile manager](#)^[210]. 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.

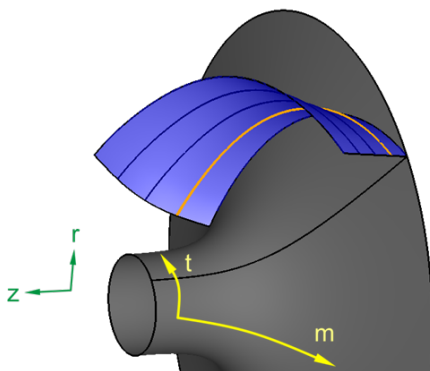
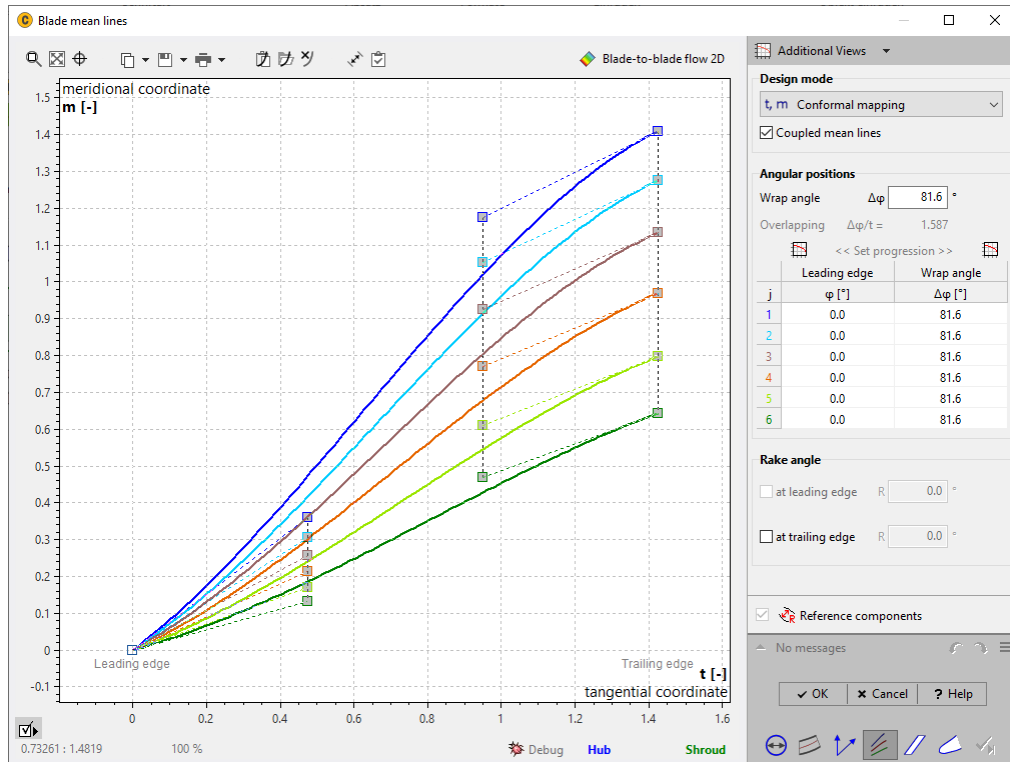


The visibility of the inner mean lines can be toggled via "Inner spans".

t, m Conformal mapping

The blade is designed in the conformal t, m mapping by Bezier curves.

CFturbo's primary design is fixing point 0 (leading edge) for all cross sections at tangential coordinate $t=0$ and meridional coordinate $m=0$, while point 3 is determined by the meridian coordinate of the trailing edge m and the wrap angle θ . The initial wrap angle θ_0 is based on [empirical functions](#)^[198].



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 :

$$dm = \frac{dM}{r} \quad dt = \frac{dT}{r}$$

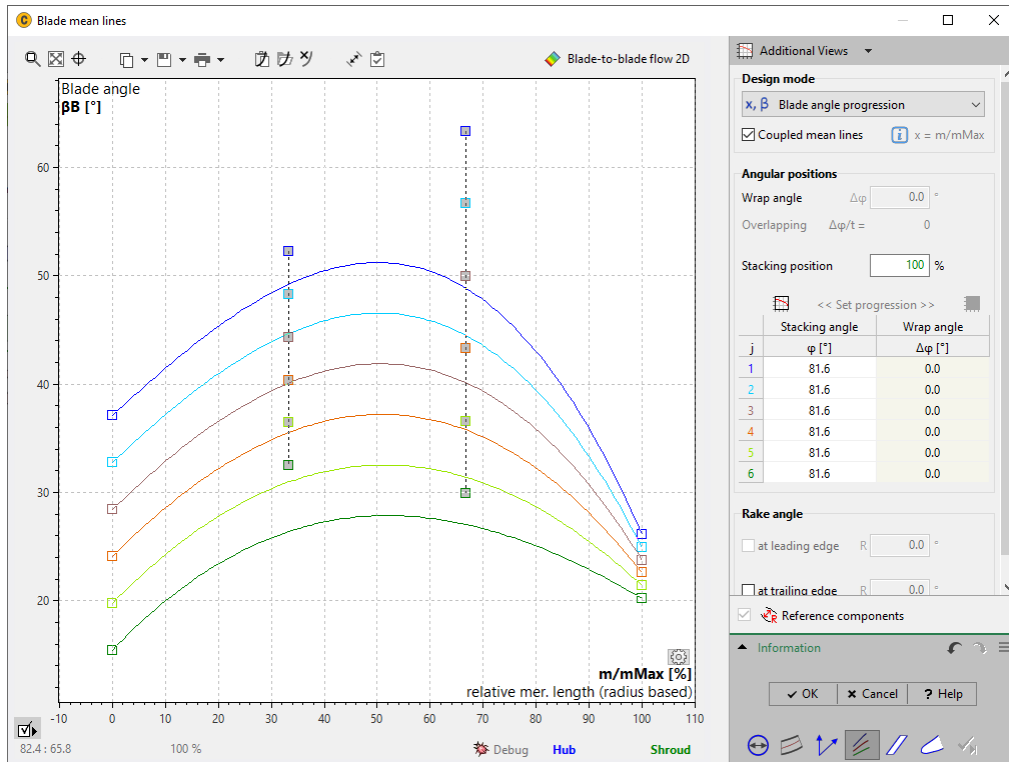
$$\tan\beta = \frac{dm}{dt}$$

If you select any point of the inner or outer cross sections, you can move this point along the related straight line. This line is given by B_1 or B_2 (rotation of the connecting line). Points 0 (leading edge) and 3 (trailing edge) can only be moved horizontally ($m=\text{const}$). Points 3 can be moved interactively (move/ rotate trailing edge). Points 0 (leading edge) can moved only by modifying wrap angles in table [Angular positions](#) ⁴⁰⁶.

x, Blade angle progression

The blade is designed via the Beta distribution by Bezier curves.

x is a flexible variable, which can be selected in [Preferences](#) ¹⁹⁶ or directly within this dialog by pressing the button above the x-axis caption.



x, 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')
	angular speed

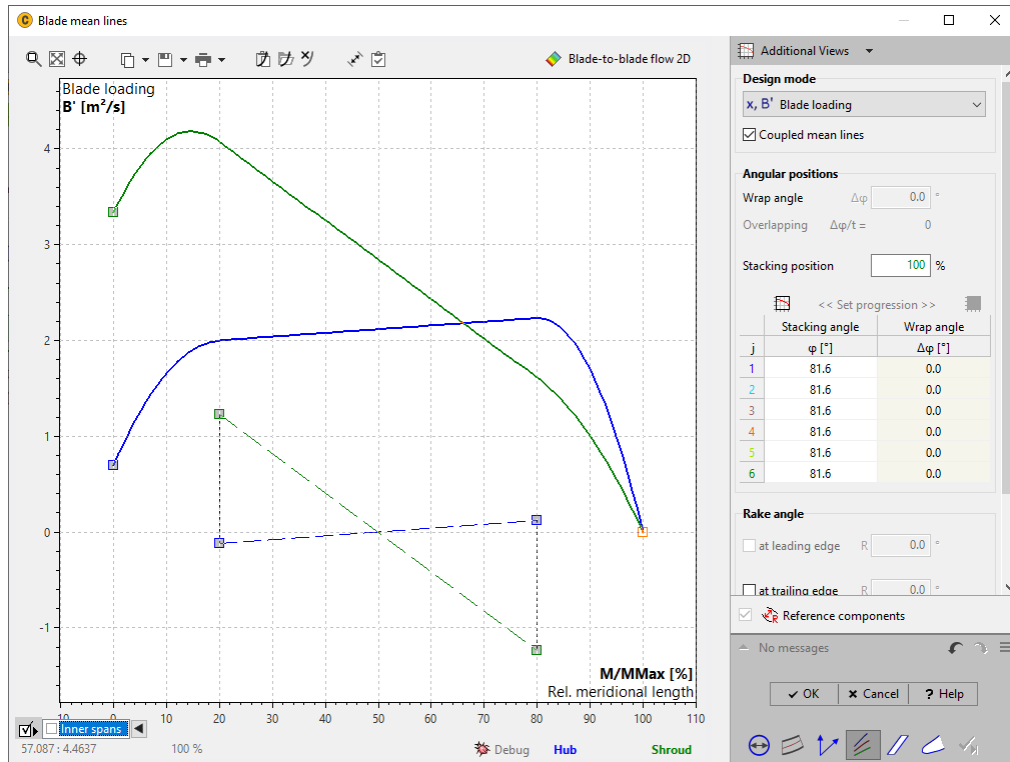
By integrating B' w.r.t M/M_{\max} one gets the swirl $c_{u2}\hat{r}_2$ that is to say the Euler work. The B' curve is consisting of 3 parabola. The inner parabola's third coefficient is zero which results in a linear piece of the curve. Parameters of the curve are the absolute values of B' at LE and 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](#)^[356]. The result of the definition of the B' curve is a certain c_u -distribution that will be used for the calculate the relative β_F (impeller) and the absolute flow angle α_F (stator) resp.:

$$\tan(\beta_F) = \frac{c_m}{\omega \cdot r - c_u},$$

$$\tan(\alpha_F) = \frac{c_u}{c_m}.$$

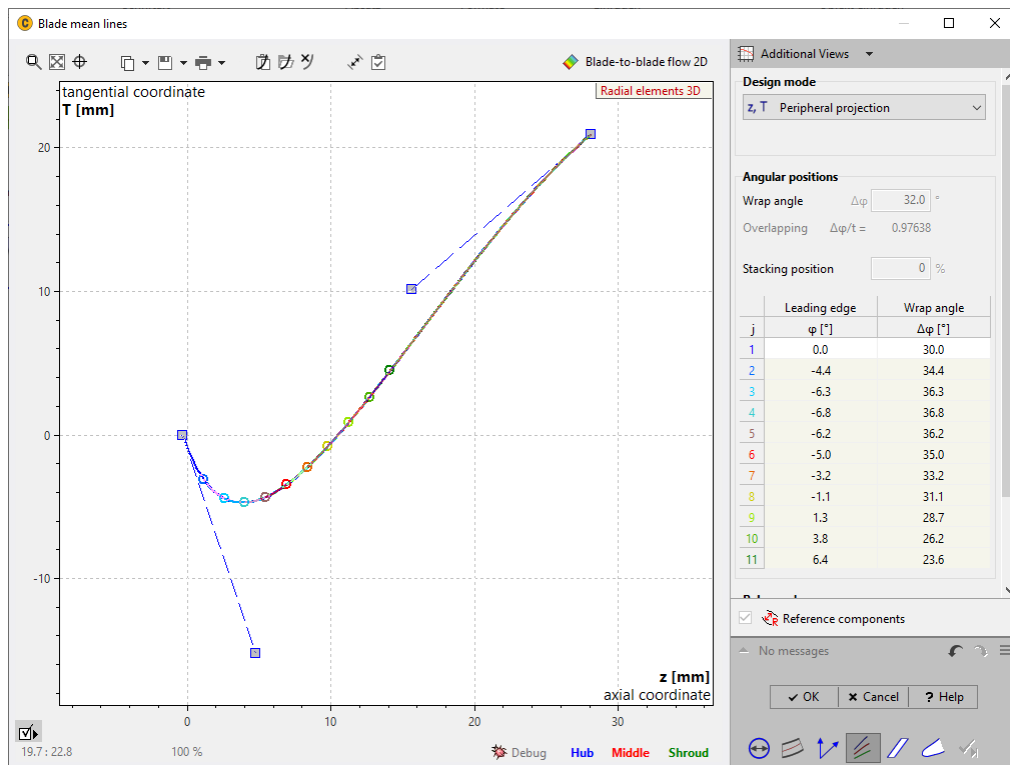
Using the information of incidence and deviation from the [blade properties](#)^[371] the absolute flow angle α_B (or α_F for stators) is determined.

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](#)^[424]). That is why deviations from the Kutta-condition may occur.



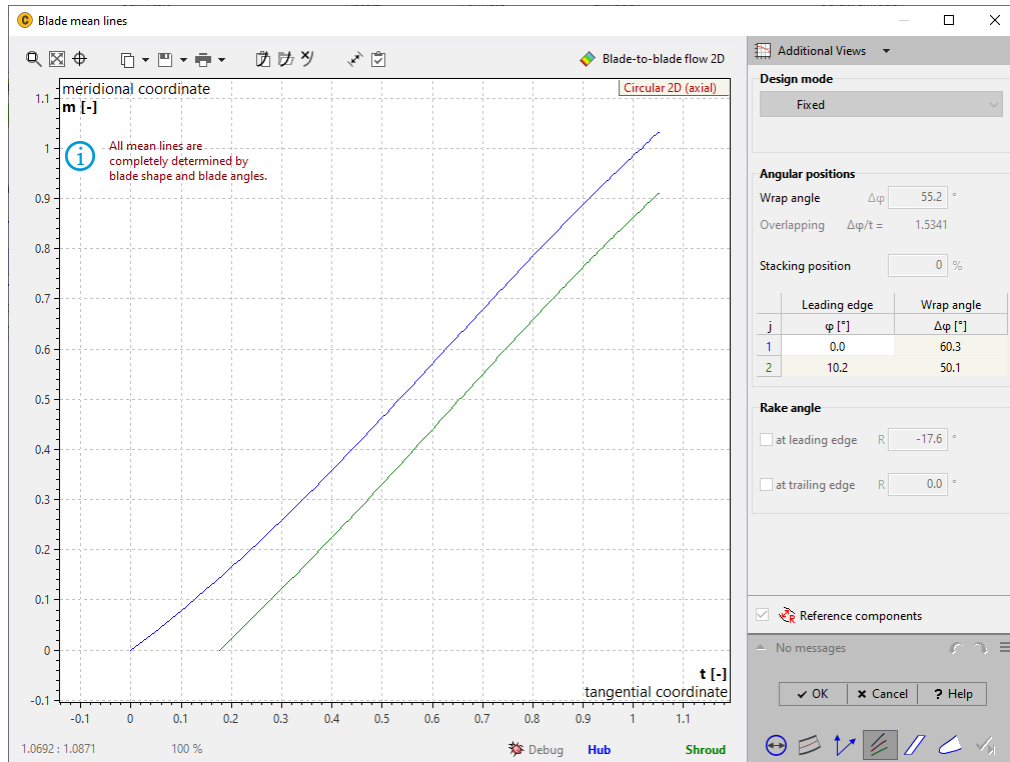
z, T Peripheral projection [for Axial impellers & Radial element blades ³⁸⁹ only]

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.



Fixed

All mean lines are completely determined by blade shape and blade angles. (see [Blade properties](#) ³⁷¹)



7.3.3 Blade profiles

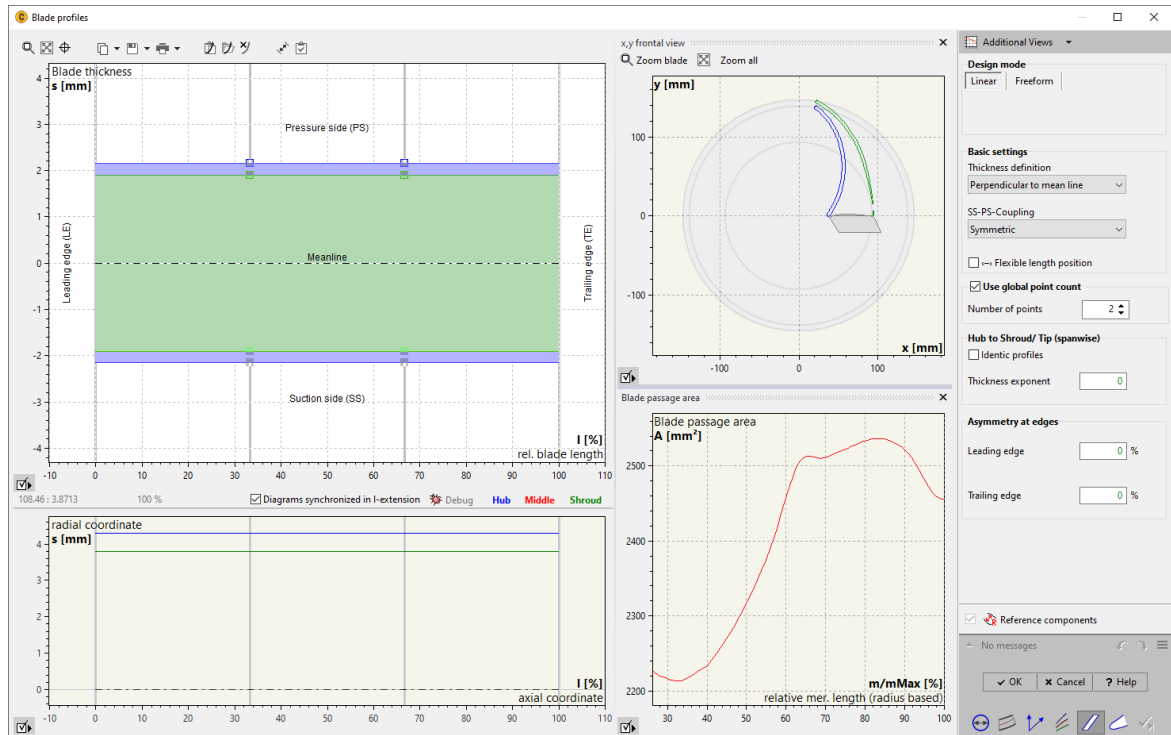
? IMPELLER | Blade profiles

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](#)^[198]).

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 ($t = \pi d / n_{Bl}$) 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).



The following properties for the profile design can be specified:

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

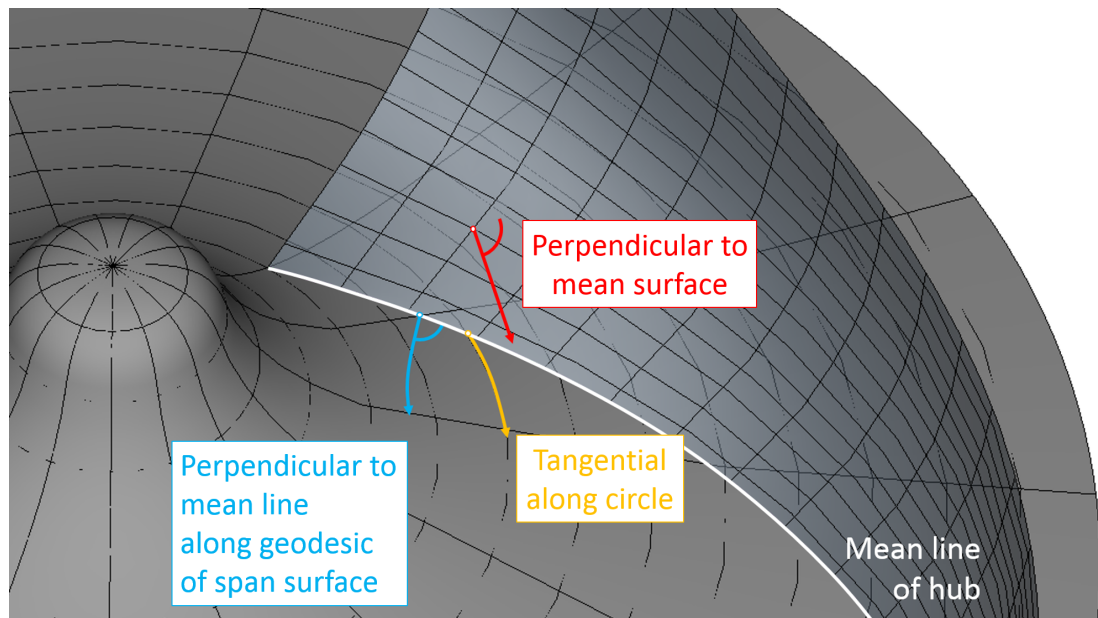
After creating the mean surface from all mean lines, the thickness values are added along surface normal. Naturally, this method depends on all mean lines.

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.



SS-PS-Coupling

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

Global point count

Global number of control points

Flexible length position

Shifting control points in horizontal direction

Hub to Shroud/Tip (spanwise)

Identic profiles

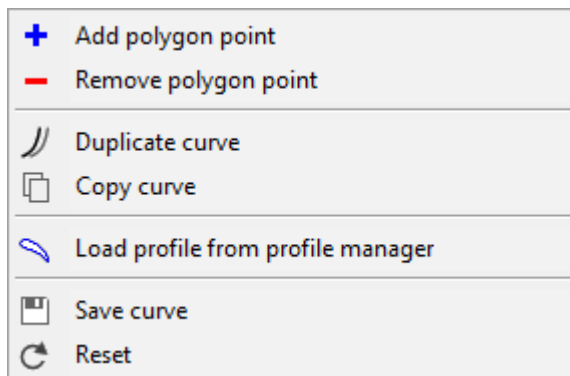
All profiles have the same thickness distribution

Thickness exponent

Adjusts the morphing of hub/shroud-thickness for inner profiles. Default is linear.

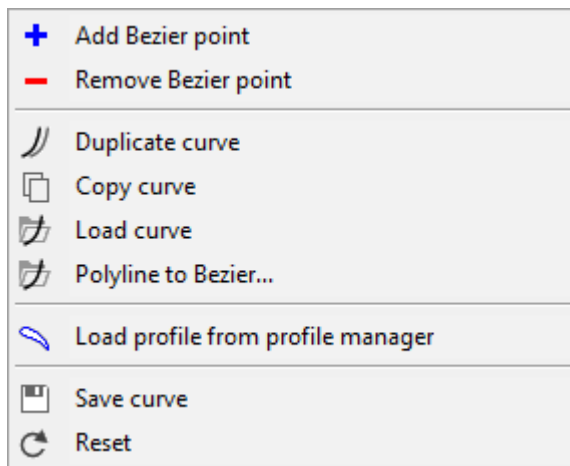
Asymmetry at edges (hub)

Provides adjustments of asymmetry at leading / trailing edge relative to first inner control points respectively.



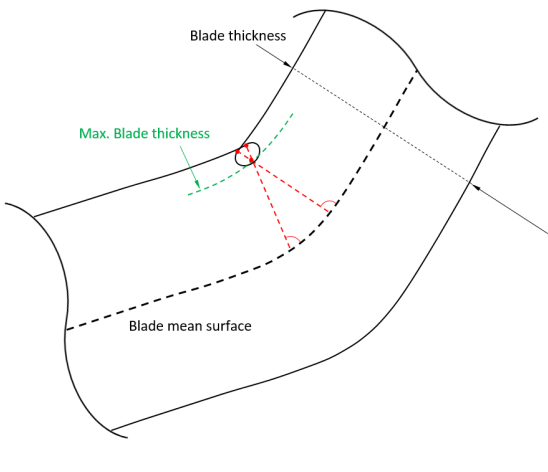
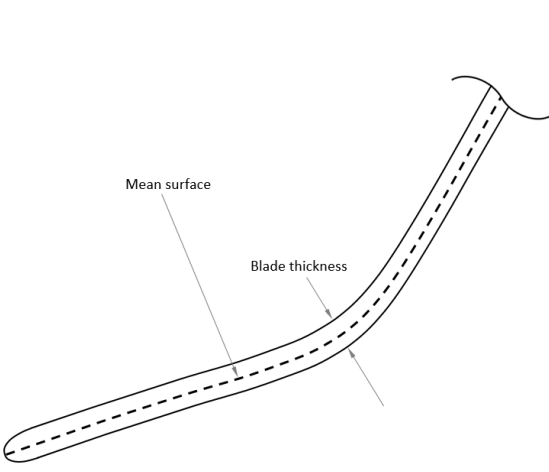
Each thickness curve has a popup menu to add/ remove polygon/ Bezier points, to load or save the curve and to reset the distribution to default.

For Bezier curves a [Polyline to Bezier](#)⁴⁴⁶ conversion is available as well as using a thickness distribution from a pre-defined profile from [profile manager](#)²¹⁰.



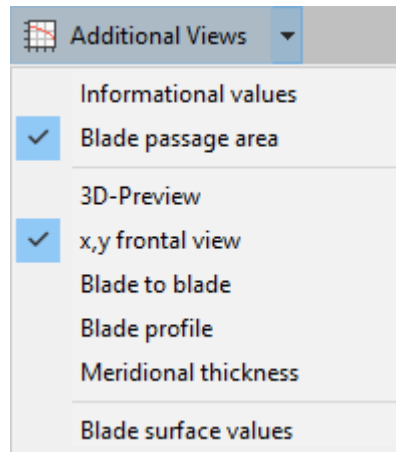
Possible warnings

Problem	Possible solutions
Pressure and suction side (...) are intersecting or swapped.	
The blade sides are intersecting or they are on the opposite position. Normally this can occur only when loading profiles from file.	Check the imported profile data if a) pressure and suction side are not intersecting b) pressure side is always above suction side
Profile of Main/ Splitter blade exceeds its valid range.	
Profile is defined for a relative blade length smaller 0% or greater 100%.	Check the imported profile data or correct the Beziér control points to lie between 0% and 100%.
Loaded profiles do not correspond to settings of design mode	
Profile properties defined by context menu in the design dialog do not match Design mode settings.	May occur if thickness distribution is loaded from profile manager ²¹⁰ : 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
Blade thickness values don't match target thickness on LE/TE.	

Problem	Possible solutions
Current profile thickness on leading- / trailing edge deviate from the specifications of the Blade properties ³⁷¹ dialog.	Check the imported profile data if the values for leading and trailing edge match those of the Blade properties ³⁷¹ dialog.
<p align="center">Pressure/Suction side at Hub/Shroud: max. thickness seems too high to get smooth surface.</p>	
The combination of of high blade thickness and high meanline curvature results in degenerated blade profiles and prevents creating smooth blade surface.	Either blade thickness at the specified profile side or meanline curvature at the specified span position has to be reduced .
 <p>The diagram shows a curved blade profile. A dashed line represents the 'Blade mean surface'. A solid line represents the 'Blade thickness'. A red circle highlights a point where the thickness is high, labeled 'Max. Blade thickness' with a green arrow. The high curvature of the mean surface and the high thickness at that point lead to a degenerated profile.</p>	 <p>The diagram shows a curved blade profile. A dashed line represents the 'Mean surface'. A solid line represents the 'Blade thickness'. The thickness is controlled along the mean surface, resulting in a smooth blade surface.</p>
<p align="center">Internal blade thickness is lower than specified in Blade properties dialog.</p>	
After changing the blade thickness on leading or trailing edge in the Blade properties ³⁷¹ 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.	Adjust the inner control points

7.3.3.1 Additional views

The following information can be displayed in the blade profile dialog using the "Additional views" button:



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

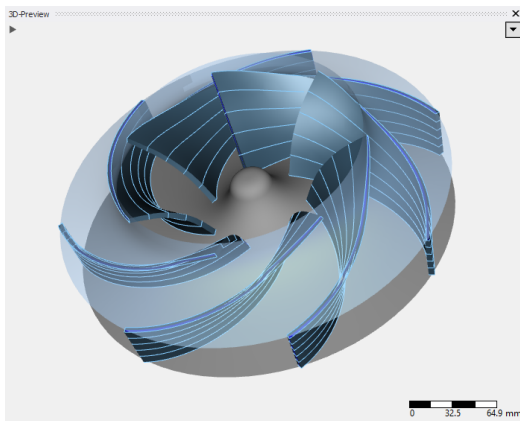
If the cells are colored red, then the thickness on leading/trailing edge is differing from the **Target thickness**.

Target thickness

Orthogonal blade thickness values for hub and shroud profiles at leading edge and at trailing edge as defined in the [Blade properties](#)^[371] dialog.

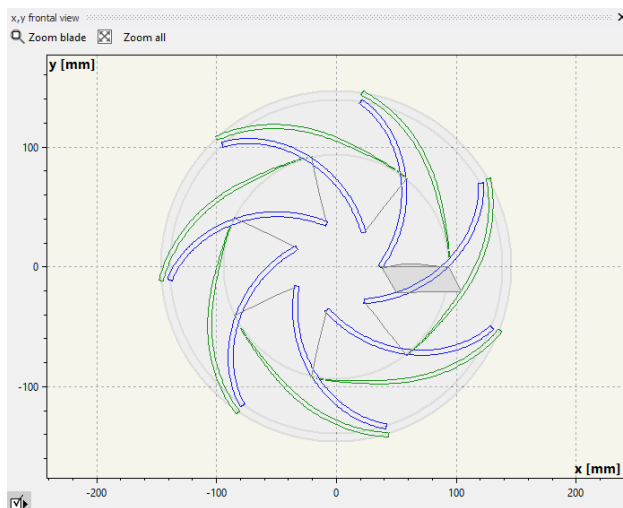
Please note that the blade thickness on leading and trailing edge should be modified in the [Blade properties](#)^[371] dialog only. In this case the blade angle calculation should be updated due to the blade blockage.

3D-Preview



[3D model](#) ²²⁵ 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.

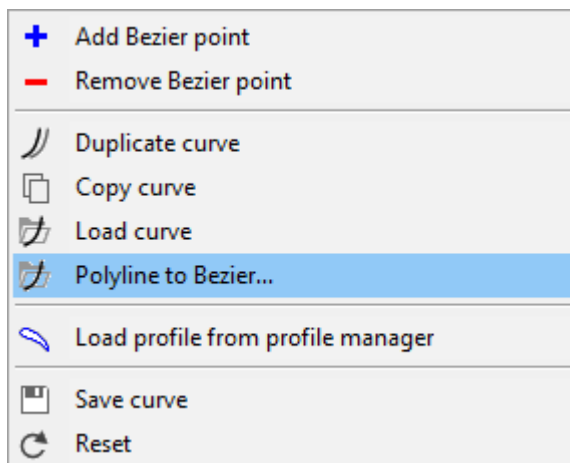
Meridional thickness

Thickness of blade in z-r-co-ordinates. In display options  the definition of thickness can be switched.

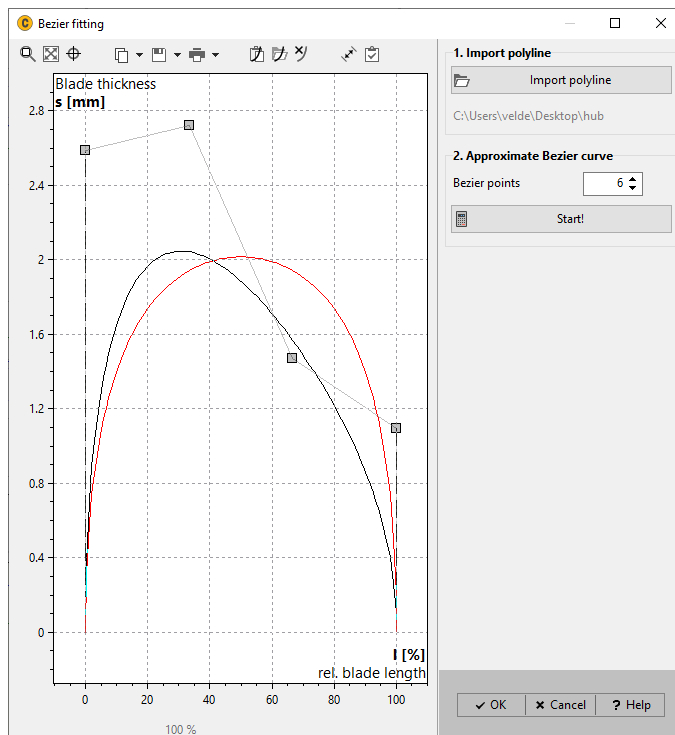
Blade surface values

See [blade surface values](#)  ⁴¹⁹.

7.3.3.2 Converting Polyline / Bezier



Any existing thickness distribution can be converted to a Bezier curve for further modifications.



First the desired polyline is imported via Import from file. The imported curve is displayed red, the original curve blue.

By pressing the **Start!** button the position of the Bezier points is calculated in such a way that the imported polyline is approximated roughly.

The existing and via context menu added Bezier points can be moved for better matching the imported curve.

7.3.4 Blade edges

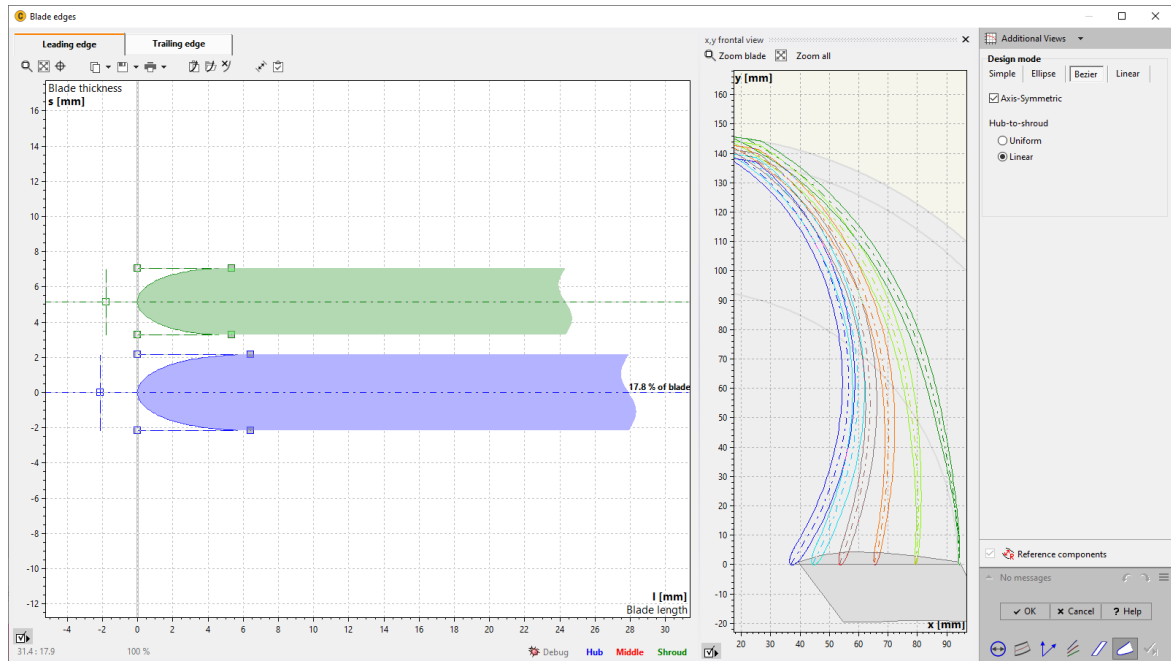
? IMPELLER | Blade edge



The previously designed blade has a blunt leading and trailing edge (connection line between endpoints of suction and pressure side).

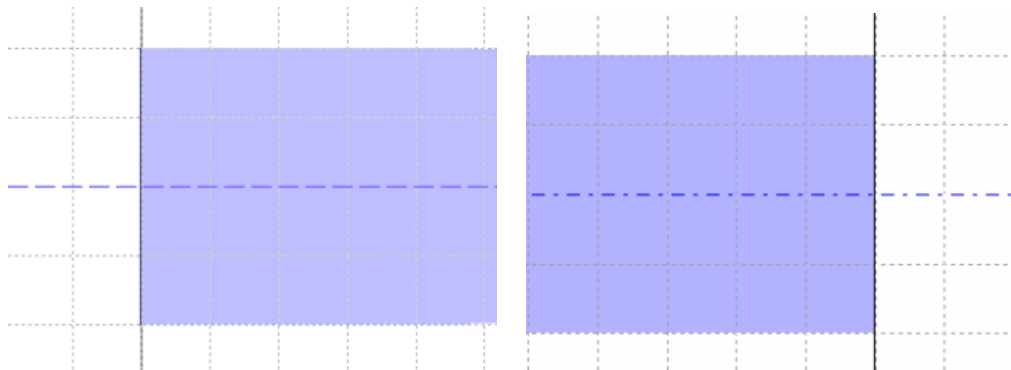
The blade edges are designed by specifying its thickness distribution. The representation of the blade thickness s is made on 15% of the straight blade length l on **leading** and **trailing edge**.

If the complete thickness distribution including leading or trailing edge was already designed in the [Blade profile](#) ^[438] dialog, then the [Edge position](#) ^[455] (transition from blade edge to blade suction/pressure side) has to be defined only.



In panel **Design mode** the blade edge shape can be selected:

(1) Simple

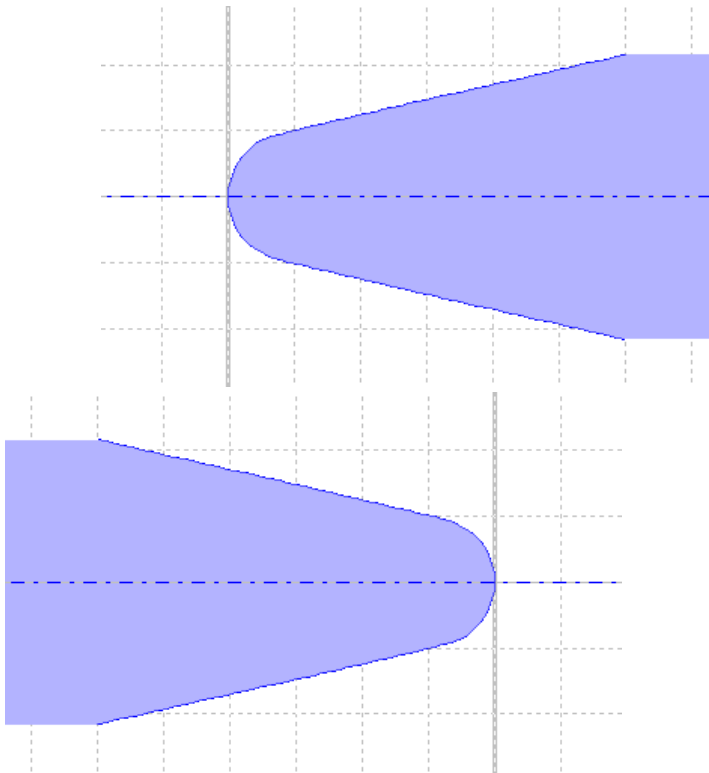


The blade edge has a blunt end. A straight line is calculated from the endpoint of suction side perpendicular to the mean line.

Trim on inlet/outlet effects trimming the blade on the corresponding inlet or outlet surface.

(2) Linear

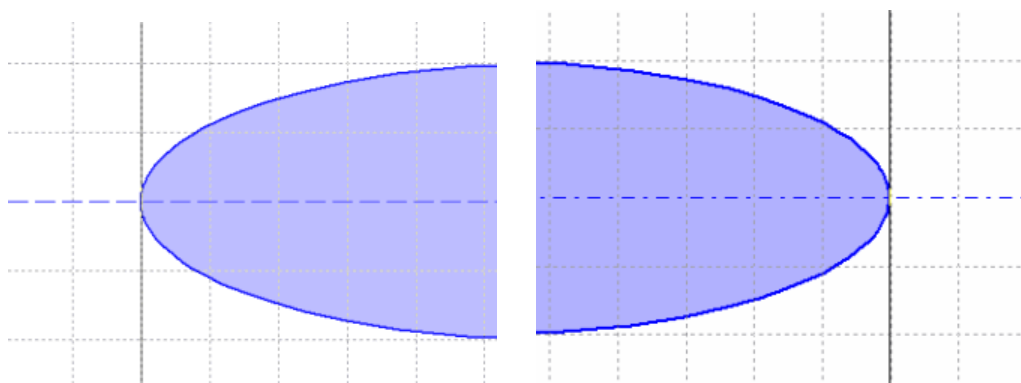
The blade thickness is changing linear, with an elliptic rounding at the end.



The edge is defined by the overall length L , the radius of the end rounding R and the axis ratio of the end rounding.

Furthermore an asymmetry can be specified.

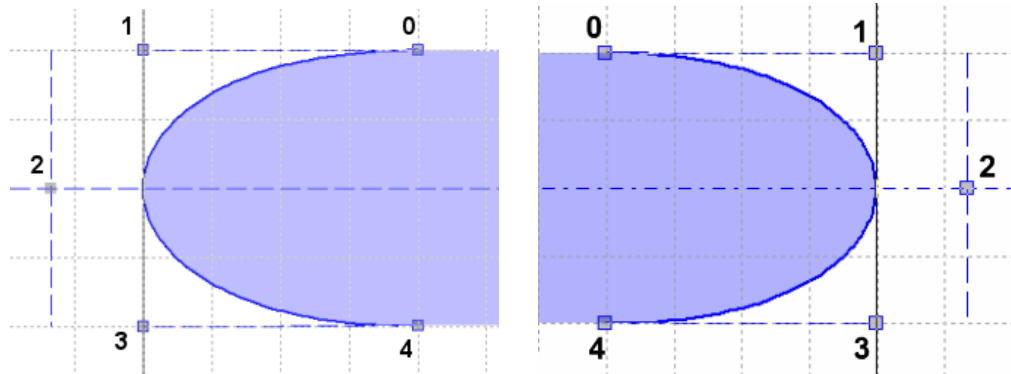
(3) Ellipse



The blade edge is rounded elliptically.

The **axis ratio** can be defined. One axis runs on the mean line, the other perpendicular.

(4) Bezier



For this purpose 4th order Bezier curves are used.

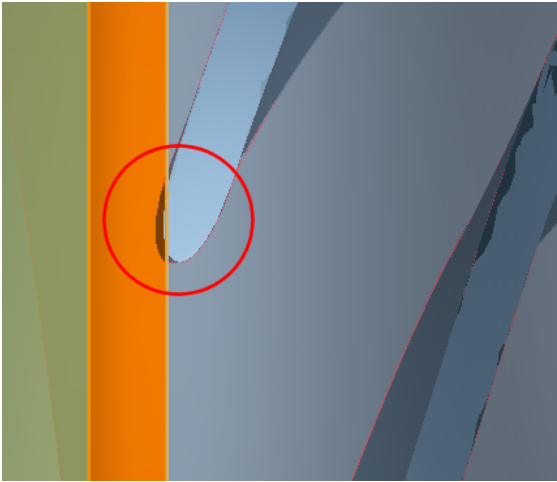
Points 0 and 4 representing the transition between the blade sides and the rounded blade edge. You can move these points only along the corresponding blade side. Bezier points 1 and 3 can only be moved on straight lines which correspond to the gradient of the curve in points 0 or 4, respectively in order 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.

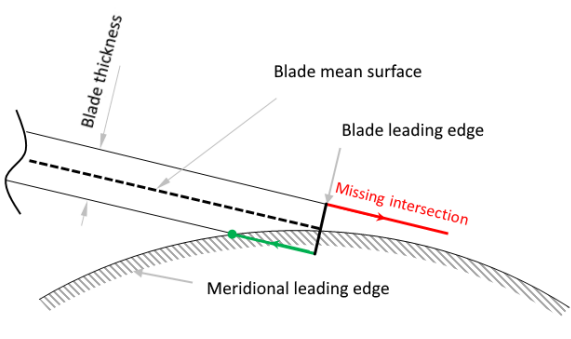
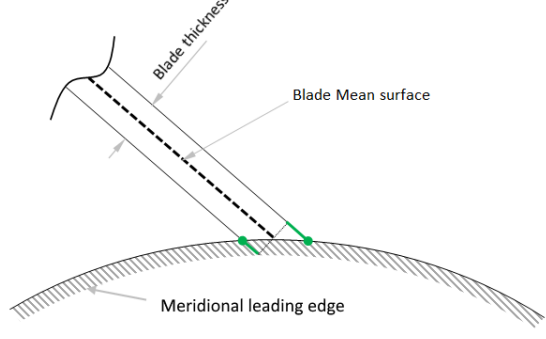
Axis-Symmetric results in symmetric geometry, i.e. points 0/4 and 1/3 have the same horizontal position and point 2 is on the middle line.

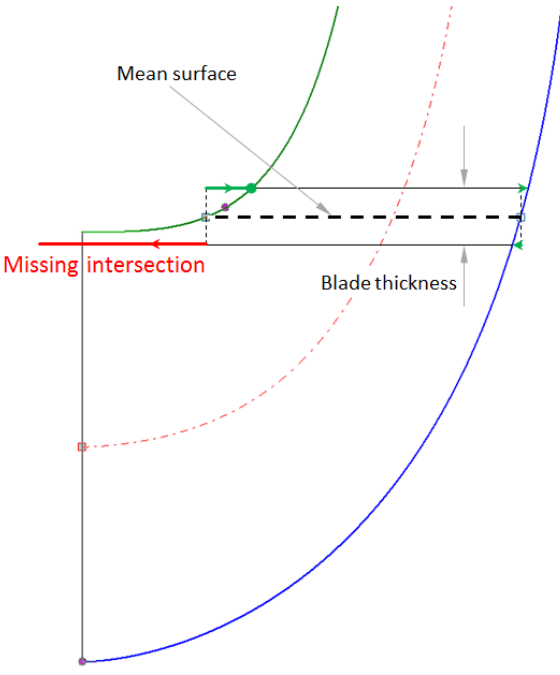
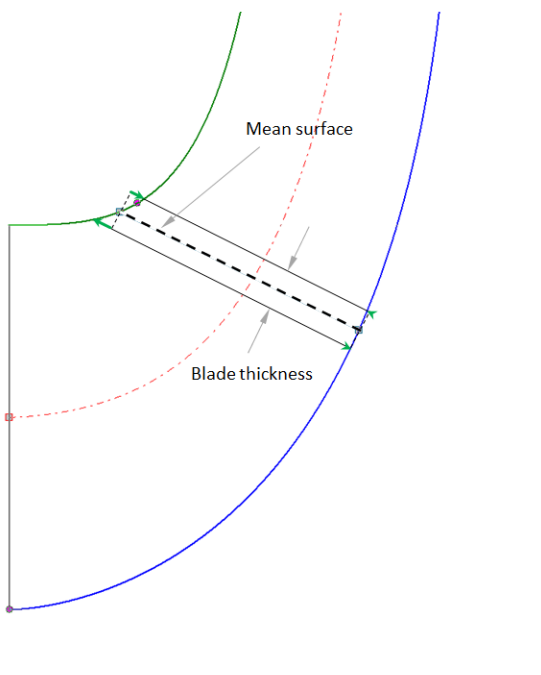
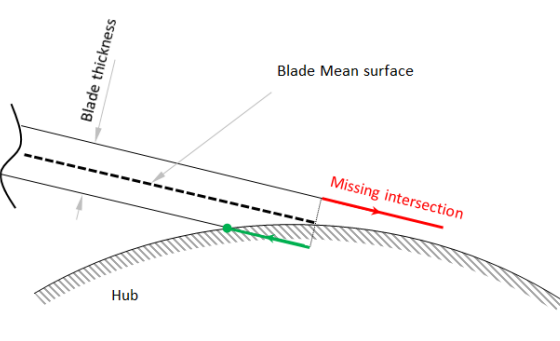
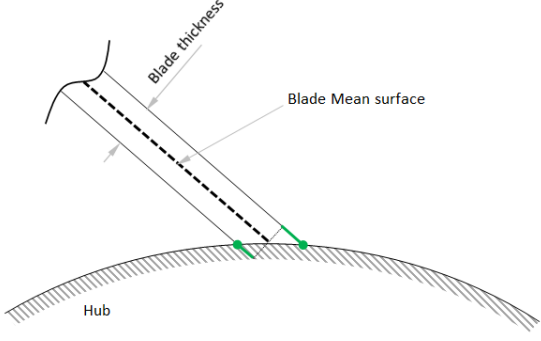
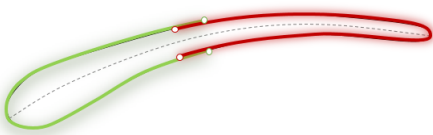
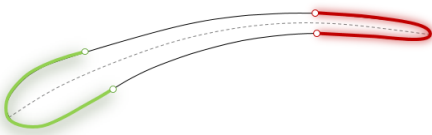
There are two different possibilities to determine the shape of the blade edge. In the Bezier curve option panel you can select between:

- **Coupled linear:**
only blade edges of hub and shroud will be fixed, while anything between will be interpolated linearly
- **Uniform:**
when designing blade edge on hub or shroud then Bezier points of all other leading edges have the same relative positions

Possible warnings

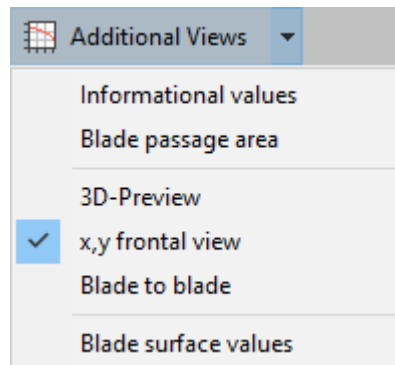
Problem	Possible solutions
Blades exceed meridional boundaries due to specified blade edge geometry. Check meridional leading and trailing edge position.	
<p>The warning indicates that some parts of the blade leading edge are outside the meridional dimensions of the component.</p>  <p>The orthogonal application of thickness on the mean lines can result in some blade position outside the meridional boundaries. Therefore, the model finishing⁴⁸⁷ option 'solid trimming' will not be available.</p>	<p>Dependent upon the location of these areas one has to modify leading or trailing edge.</p> <p>If the leading edge (or the trailing edge of turbines) exceeds the meridional boundaries you can adjust it in the Meridional contour³³⁸ dialog only.</p> <p>Exceeding trailing edge (or leading edge of turbines) can be corrected by trim on in/outlet.</p>
It is impossible to trim blade at leading/trailing edge.	
<p>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.</p>	<p>Meridional contour³³⁸, Mean line⁴⁰⁵: The angle between the mean line and the meridional leading/ trailing edge should be high.</p> <p>Blade profile⁴³⁸: Reduce blade thickness.</p>

Problem	Possible solutions
	
<p>Error when extrapolating Blade to reach Hub/Shroud surface. Check meridional geometry, blade angles and thickness.</p>	
<p>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.</p> <p>For the below illustrated configurations of meridional contour and blade geometry the extrapolation fails.</p>	<p>Meridional contour^[338]: Account for blade thickness during leading edge positioning or align leading edge towards the direction of the shroud normal (see images below).</p> <p>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.</p> <p>Blade properties^[371]: Increase the number of spans.</p> <p>Blade profile^[438]: Reduce blade thickness or change thickness definition to "Perpendicular to mean line".</p> <p>Mean line^[405]: Check mean line shape and keep lean angle on a low level.</p>

Problem	Possible solutions
	
	
Impossible blade edge design: overlapping leading and trailing edge blade.	
Overlapping of Leading and trailing edge, impossible to design pressure and suction side.	Reduce the edge positions of one or both edges.
	

7.3.4.1 Additional views

The following information can be displayed in the blade edges dialog using the "Additional views" button:



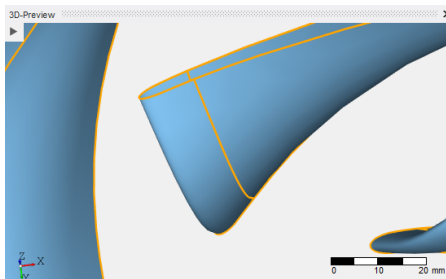
Informational values

The Info panel represents information of the designed blade profile:

Throat area

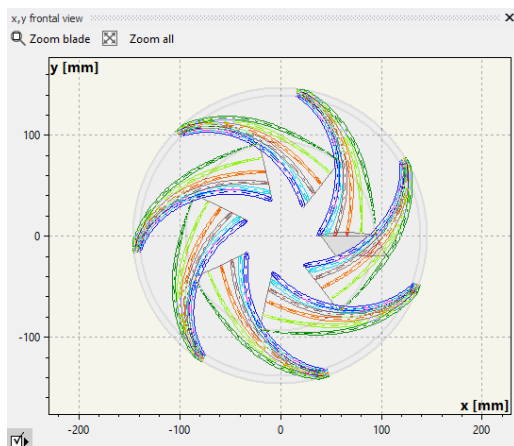
Smallest cross section between 2 neighboring blades

3D-Preview



[3D model](#)²²⁵ of the currently designed blades as well as surfaces of hub and shroud and mean surfaces.

Frontal view



The **Frontal view** represents the designed blades in a frontal view, including diameters d_H und d_2 .

Furthermore the smallest cross section between 2 neighboring blades is displayed.

Blade passage area

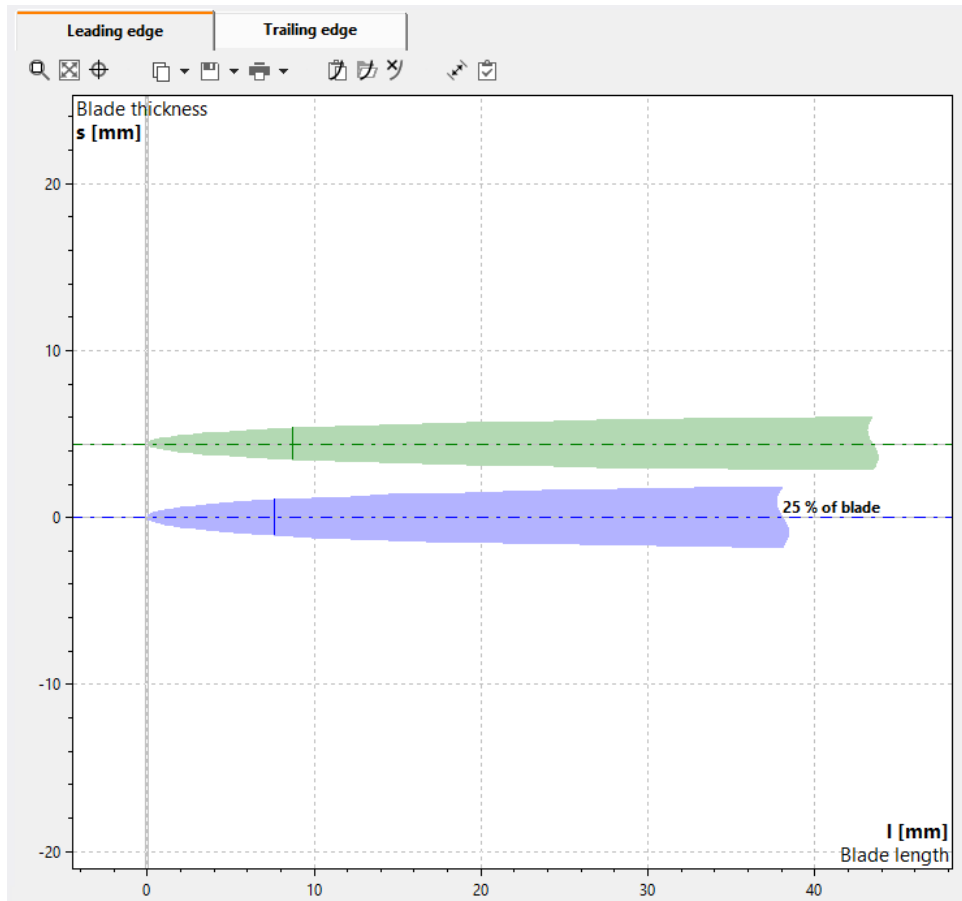
Area that is approximately perpendicularly flown through and formed by hub, shroud and two neighboring blades.

Blade surface values

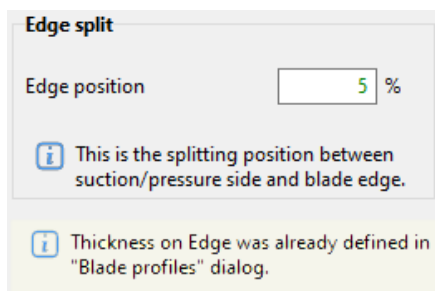
See [blade surface values](#) ⁴¹⁹.

7.3.4.2 Edge position

If the complete thickness distribution including leading or trailing edge was already designed in the [Blade profile](#) ⁴³⁸ dialog, then the [Edge position](#) ⁴⁵⁵ (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](#) ⁴⁵⁷

- (2) [Blade profiles](#)  473
- (3) [Blade sweeping](#)  475

7.4.1 Blade properties

? IMPELLER | Blade properties

Definition of blade properties is made in three steps:

- (1) [Cu-specification](#)  460
- (2) [Blade profile selection](#)  463
- (3) [Kinematics](#)  466

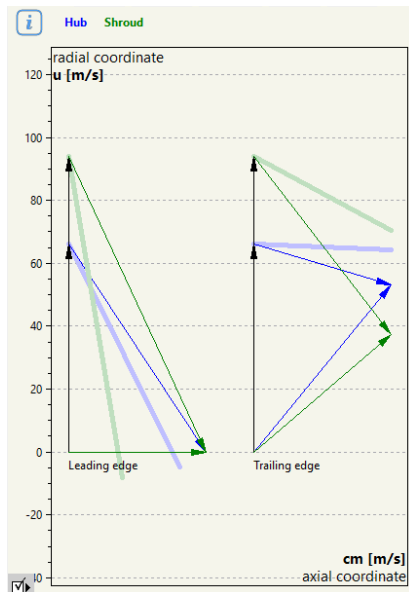
Specification of number of blades and number of spans

Blades

Number of blades	<input type="text" value="20"/>	
Number of spans	<input type="text" value="15"/>	2 ... 15

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.

	Span = 1 (Hub)		Span = 15 (Shroud)	
	Leading edge	Trailing edge	Leading edge	Trailing edge
z	3	57	3	57
d	280	280	399	399
oF	90	39.4	90	49.5
BF	33.5	73.7	24.9	37.6
u	66	66	94	94
cm	43.7	43.7	43.7	43.7
cu	0	53.2	0	37.3
cr	0	0	0	0
cax	43.7	43.7	43.7	43.7
c	43.7	68.8	43.7	57.5
wu	-66	-12.8	-94	-56.7
w	79.1	45.5	103.7	71.6
τ	1	1	1	1
i δ	-7	14	-15.4	24.1
w ₂ /w ₁		0.58		0.69
c ₂ /c ₁		1.57		1.31
$\Delta\alpha_F$		-50.6		-40.5
$\Delta\beta_F$		40.1		12.7
$\Delta\beta_B$		61.1		52.1
v		0.83		0.65
$\Delta(cu-r)$		7.44		7.44
T		0.702		0.988
Δ_{pt}		0.04208		0.04208

(2) Values

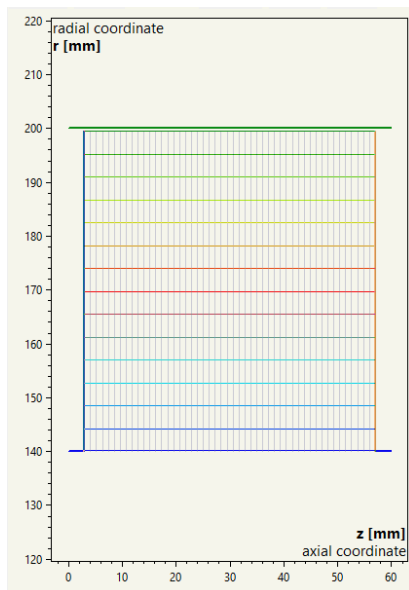
Numerical values of velocity components and flow angles are displayed in a table. The track bar on top of table can be used to get the values at any span. A short description is at mouse cursor too:

- z Axial co-ordinate
- d Diameter
- F Angle of absolute flow to circumferential direction
- 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: $w_u + c$
- w Relative velocity
- τ Obstruction by blades (see below)
- i Incidence angle: $i = B_1 - 1$
- Deviation angle: $= B_2 - 2$
- w_2/w_1 Deceleration ratio of relative velocity
- F Absolute deflection angle
- F Relative deflection angle
- B Blade deflection angle
- Slip coefficient

($cu \cdot r$) Swirl difference
 M Torque
 p_t Pressure difference (total-total)

(3) Meridian

The Meridian with locations of the spans is displayed in this diagram.



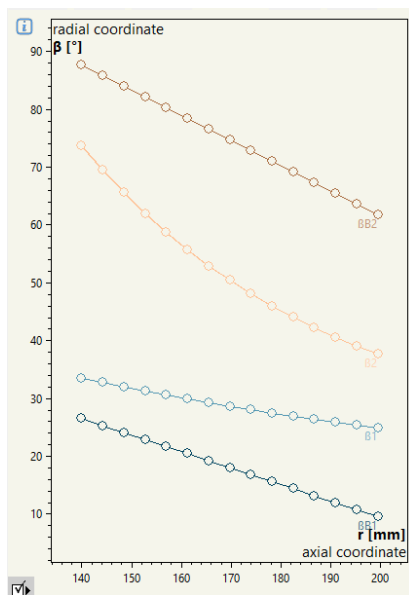
(4) Current β

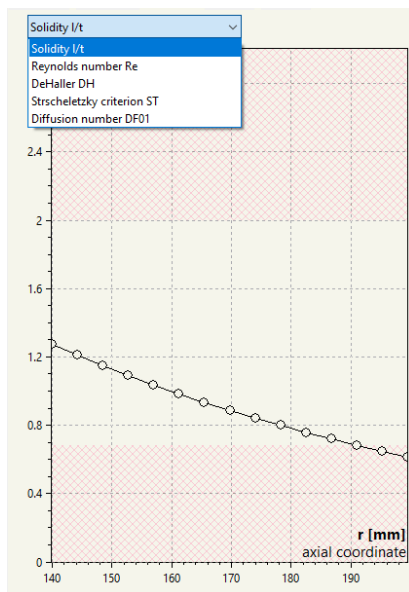
Here blade angles as well as relative flow angles are displayed versus span.

Progressions of geometric parameters (angles):

1/2 Angle of relative flow to circumferential direction

B1/2 Blade angles at leading and trailing edge





(5) Criteria

Progressions of aerodynamic and airfoil parameters:

Re	Reynolds-number
l/t	solidity
DH	DeHaller criterion
ST	Strscheletzky criterion
DF01	diffusion number

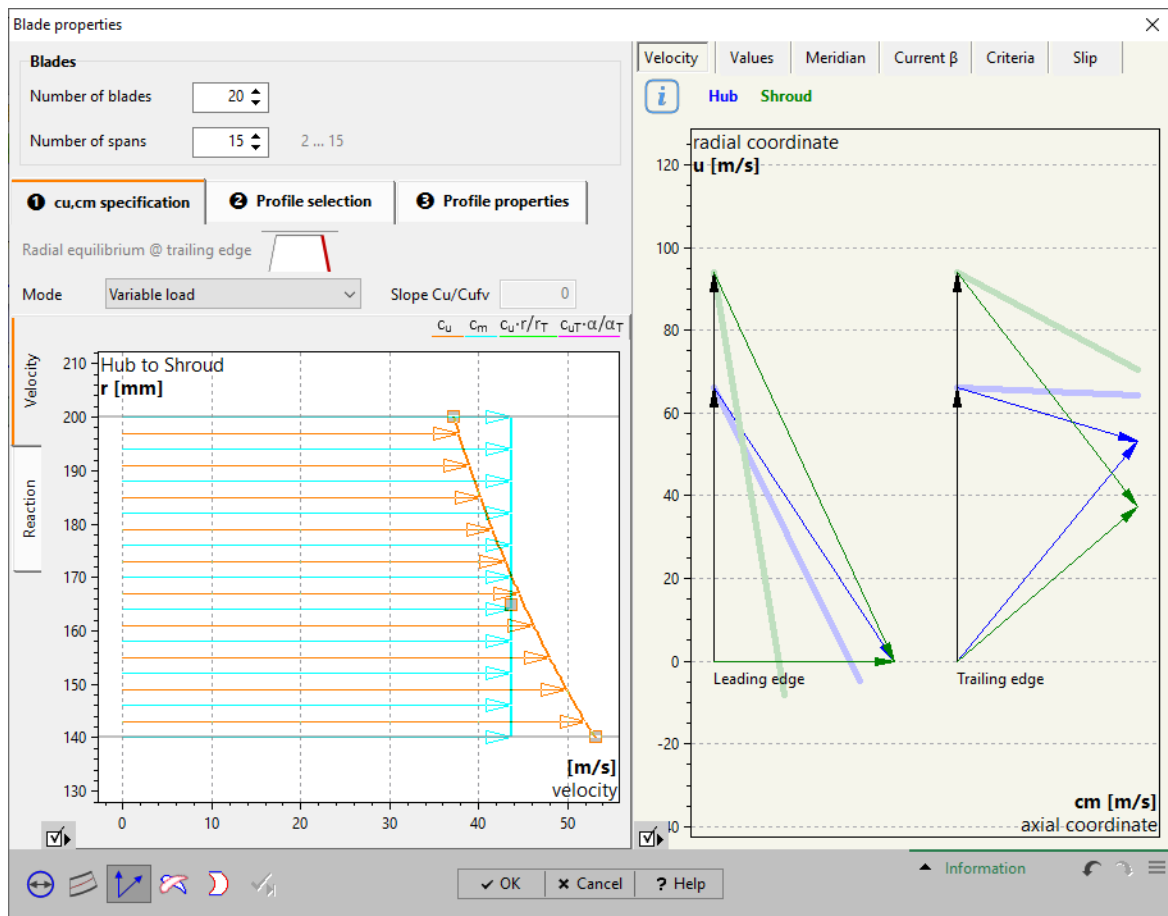
7.4.1.1 Cu-specification

? Impeller | Blade properties

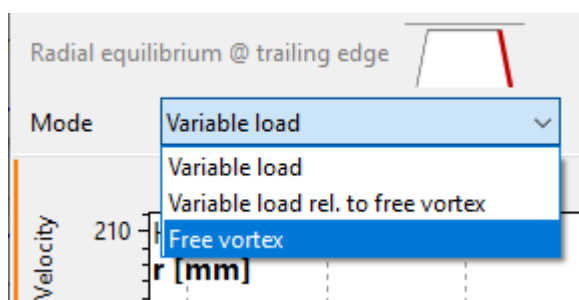


[Axial machines only]

On tabsheet **cu, cm definition** the velocity triangles at every span can be defined in accordance to the [radial equilibrium](#) ⁴⁶³.



It can be chosen from 3 different modes concerning the manipulation of $c_{u2}(r)$:

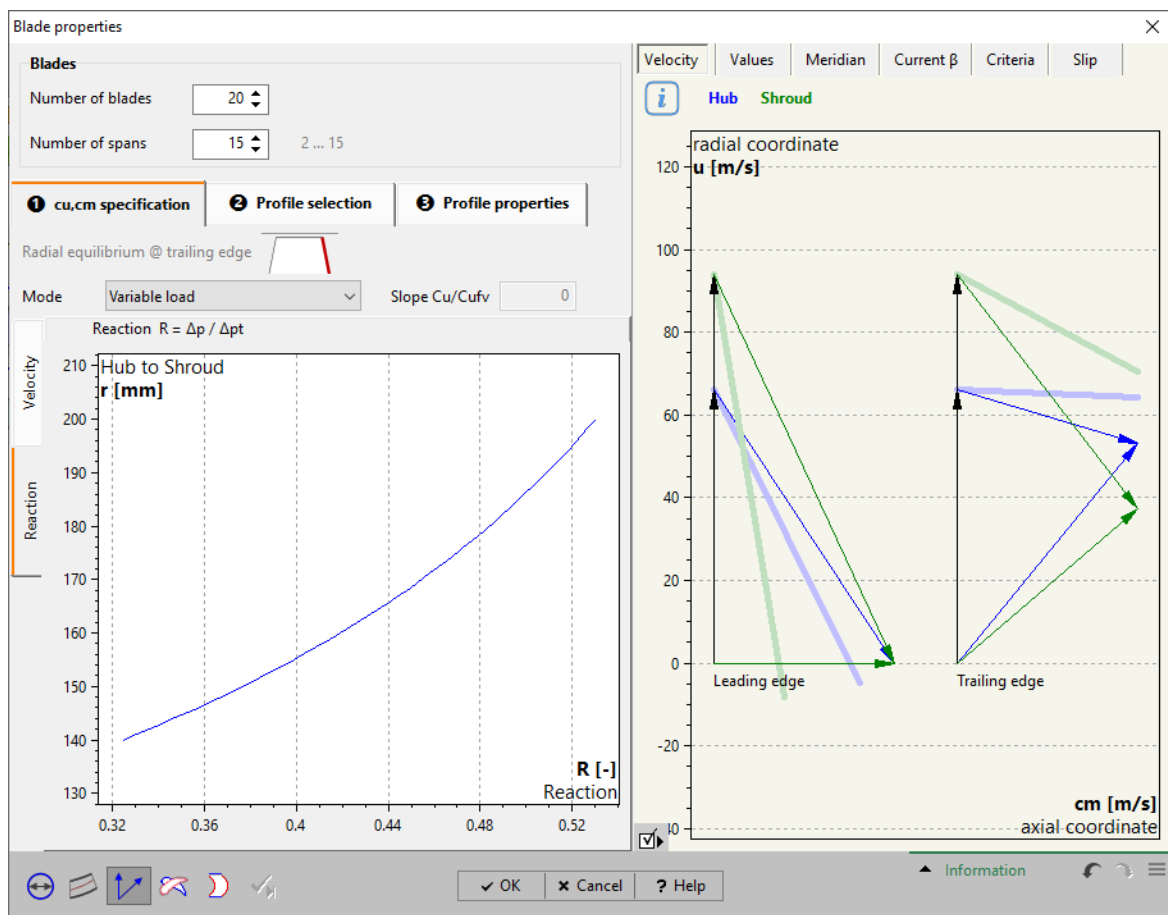


Variable load	Free vortex	Variable load rel. to free vortex
---------------	-------------	-----------------------------------

<p>The $c_{u2}(r)$-specification is controlled by a second order Bezier curve.</p>	<p>$c_{u2}(r)$ is defined to get the same swirl at every span:</p> $c_{u2}(r) \cdot r = c_{u2iso}(r) \cdot r = \text{const}$	<p>The slope is the derivative according to:</p> $\text{slope} = \frac{d\left(\frac{c_{u2}}{c_{u2iso}}\right)}{d\left(\frac{r}{r_{Tip}}\right)}$ <p>With a slope of zero a free vortex distribution is set.</p>
---	---	---

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: $R = h_{\text{stat}} / h_{\text{tot}}$



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, ventilator] and before a rotor [turbine] respectively:

$$0 = p_2 dA - (p_2 + dp_2) dA + r \omega^2 \rho \cdot dA \cdot dr$$

$$\Rightarrow \frac{dp_2}{dr} = \rho \frac{c_{u2}^2}{r}$$

The definition of total pressure in section 2 differentiated with respect to r plus above equation yield:

$$\frac{dp_{t2}}{dr} = \rho \frac{c_{u2}^2}{r} + \rho \left(c_{m2} \frac{dc_{m2}}{dr} + c_{u2} \frac{dc_{u2}}{dr} \right)$$

With the blade work according to Euler the equation becomes:

$$\eta_{\text{Imp}} \cdot 2\pi n \frac{d(rc_{u2})}{dr} = \frac{c_{u2}}{r} \frac{d(rc_{u2})}{dr} + c_{m2} \frac{dc_{m2}}{dr}$$

With the following boundary conditions and a given $c_{u2}(r)$ -specification the solution of the differential equation gives a $c_{m2}(r)$ -distribution and therefore the complete velocity triangles at every span.

$$\dot{m} = \int_{r_{\text{Hub}}}^{r_{\text{Shr}}} \rho \cdot c_{m2}(r) \cdot 2\pi r \cdot dr$$

$$P = \int_{r_{\text{Hub}}}^{r_{\text{Shr}}} u(r) \cdot c_{u2}(r) \cdot \rho \cdot c_{m2}(r) \cdot 2\pi r \cdot dr$$

From the velocity triangles the degree of reaction can be determined by the following equation:



$$R = \frac{\Delta h}{\Delta h_t} = 1 - \frac{\Delta(c^2)}{2\Delta(u_2 \cdot c_{u2})}$$

7.4.1.2 Blade profiles

? Impeller | Blade properties 

[Axial machines only]

On tabsheet **Profile selection** the axial blade profile properties are specified. To this end the profiles have to be selected from the [Profile manager](#)^[210].

Profile properties				
Span		Group	Profile	
Hub	1	<input checked="" type="checkbox"/> NACA 4 Digit	NACA 6508	
	2	<input type="checkbox"/>	(interpolated)	
	3	<input type="checkbox"/>	(interpolated)	
	4	<input type="checkbox"/>	(interpolated)	
Middle	5	<input type="checkbox"/>	(interpolated)	
	6	<input type="checkbox"/>	(interpolated)	
	7	<input type="checkbox"/>	(interpolated)	
	8	<input type="checkbox"/>	(interpolated)	
	9	<input type="checkbox"/>	(interpolated)	
Shroud	10	<input checked="" type="checkbox"/> NACA 4 Digit	NACA 6306	

Profile specification on 1 span position is necessary at least to use the same profile on all spans.

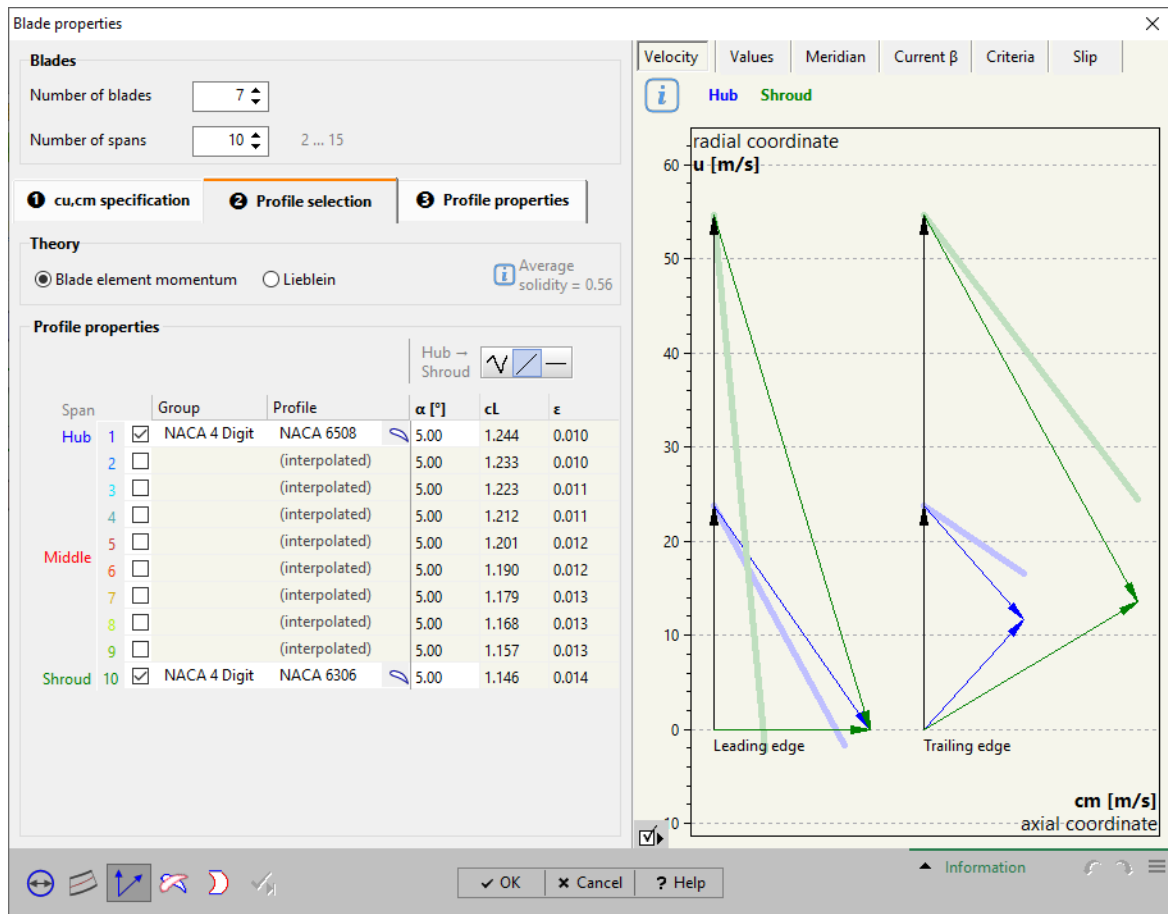
Alternatively on any other span position deviating profiles can be selected resulting in interpolation between different profiles.

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:

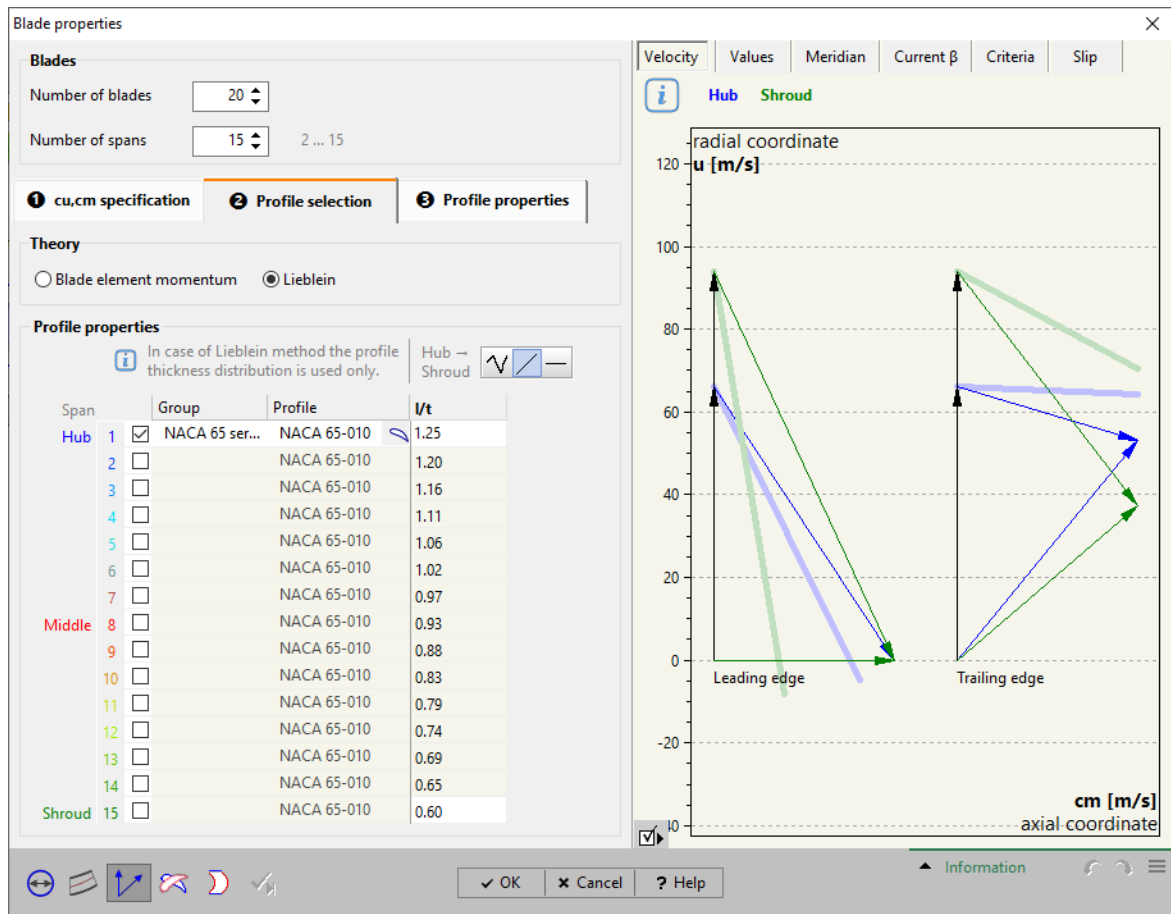
1. Blade element momentum method

Here either NACA 4 digit or point based profiles can be used. Also an angle of attack has to be specified, see [blade element momentum method](#)^[470].



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](#)^[47].



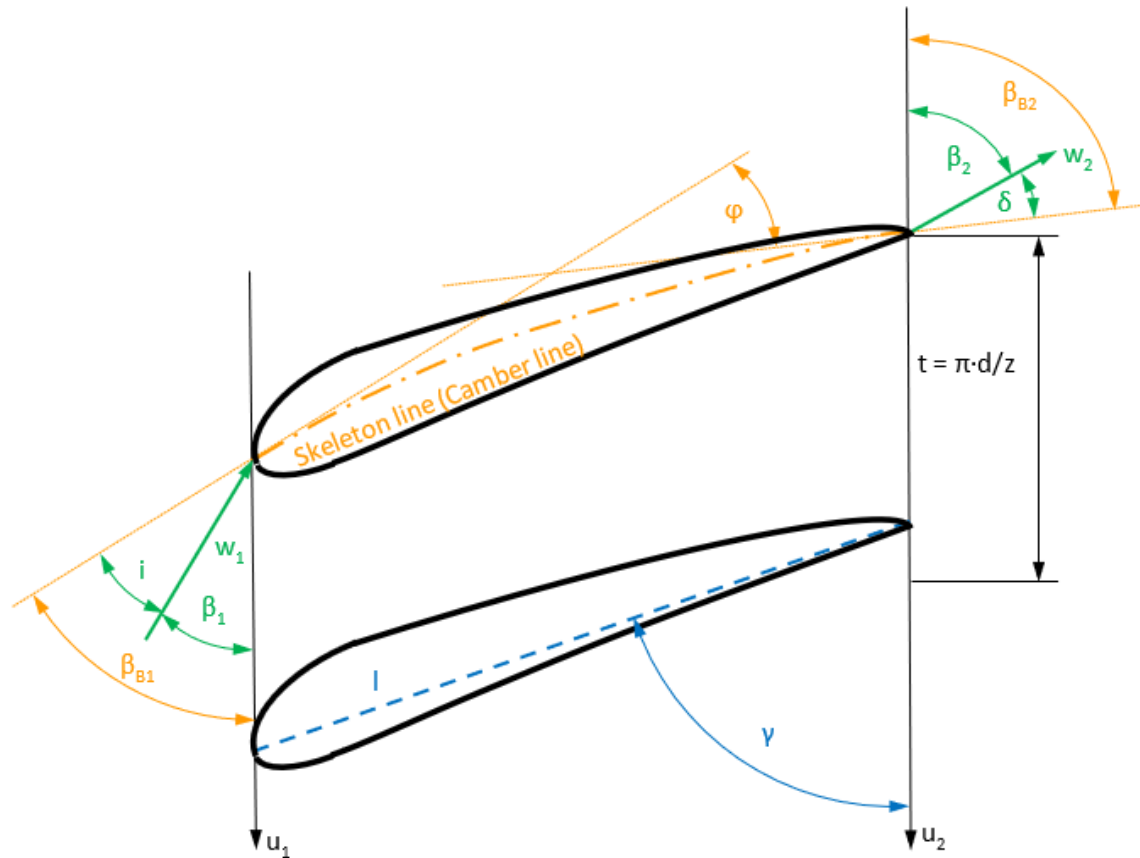
7.4.1.3 Kinematics

? Impeller | Blade properties

[Axial machines only]

The following parameters together with the chosen [profile](#)^[463] describe the blade at each span:

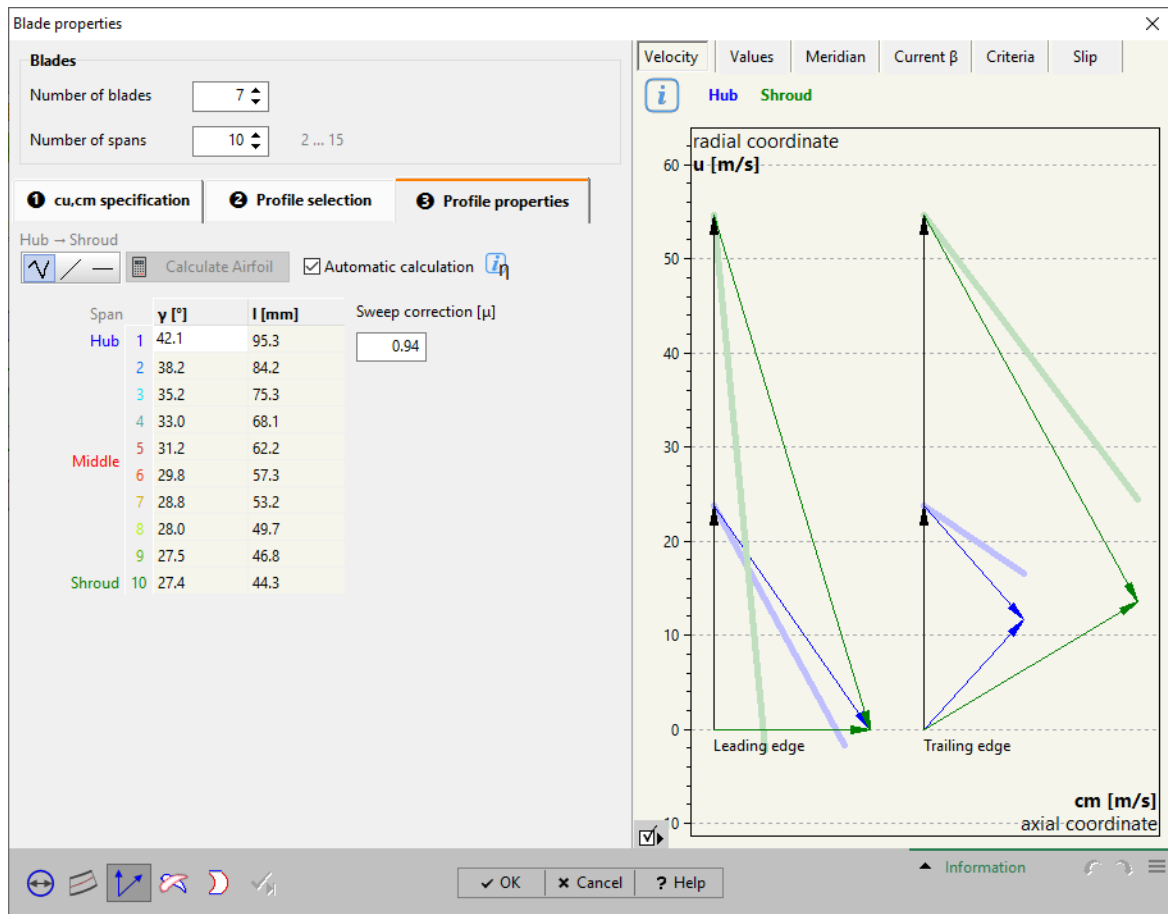
- l
- stagger angle
- chord length
- camber angle ([Lieblein method](#)^[471])



Two methods are available for the determination of the scaling (solidity) and staggering of the profiles:

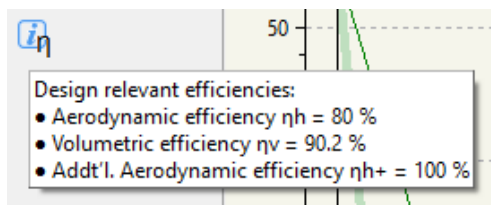
- **Blade element momentum method** ^[470] **[only ventilators]**
for low pressure applications (high specific speed nq)
- **Lieblein method** ^[471] **[pumps, ventilators]**
for high pressure applications (low specific speed nq)

On the tabsheet **Profile properties** the **stagger angles** and **solidity** are calculated.



Sweep correction μ

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](#)⁴⁷⁵.



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](#)⁵⁶⁸ criterion:

$$\left. \frac{w_2}{w_1} \right|_{\text{hub}} \geq 0.6..0.75$$

In a pipe flow having a swirl a dead water zone is built at small radii. Strscheletzky 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:

$$\left. \frac{c_{m2}}{c_{u2}} \right|_{\text{hub}} \geq 0.8$$

and for multi stage machines:

$$\left. \frac{c_{m2}}{c_{u2}} \right|_{\text{hub}} \geq 1$$

From boundary layer analysis the diffusion number applied for profiles with a maximum thickness of 10% was derived:

$$DF_{0.1} = \left(1 - \frac{w_2}{w_1} \right) + \frac{1}{2} \cdot \frac{t}{l} \cdot \frac{\Delta w}{w_1}$$

Special NACA-measurements yield a scope to be fulfilled of $DF_{0.1} \leq 0.6$.

Possible warnings

Problem	Possible solutions
Automated blade angles are active. Blades may adapt to changing input parameters.	
Blade angles or stagger angles + chord length are updated automatically if any input parameters are modified.	To fix the blade angles or stagger angles + chord length uncheck "Automatic" calculation. Then one must manually start the calculation if required.
Automated blade angles are NOT active. Dimensions are fixed, but may not reflect input parameters.	
Blade angles or stagger angles + chord length are not updated automatically if any input parameters are modified.	To be sure that all parameter modifications are considered one shall switch to an automatic calculation by checking the "Automatic" option.
The blade sweep yields a sweep correction factor of x and is different from the value currently set in blade properties (y).	

Problem	Possible solutions
See warning.	Set μ according to value given in blade sweep ^[475] .
Radial equilibrium calculation failed.	
There is not for all constellations a solution of the radial equilibrium ^[463] .	Try to change Bezier-points of cu-curve ^[460] in case of variable load or switch to free vortex.
Automatic mode not possible in case of failed radial equilibrium calculation.	
In case of failed radial equilibrium ^[460] calculation the velocity triangles are not correctly determined and cannot be used for setting the blade angles or stagger angles + chord length.	Uncheck "Automatic" and set blade angles or stagger angles + chord length manually.

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:

$$F_u = \sin(\beta_\infty + \delta) \cdot F$$

$$\approx \sin(\beta_\infty + \delta) \cdot c_L \cdot \rho \cdot \frac{w_\infty^2}{2} \cdot l \cdot b \quad \text{with} \quad F \approx F_L$$

whereas if it is derived from the force balance it reads as:

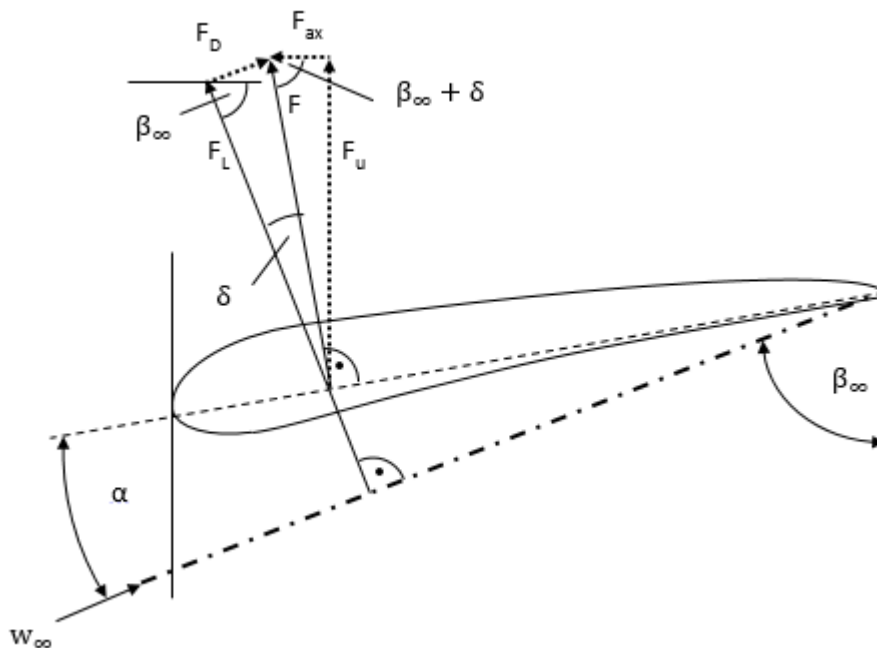
$$F_u = \dot{m} \cdot (c_{u2} - c_{u1})$$

$$= \rho \cdot c_m \cdot t \cdot b \cdot \frac{Y_{Imp}}{u}$$

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):

The meaning of the variables is given in the following table:

Y_{imp}	specific work of the impeller
l/t	solidity (chord length/pitch)
b	width of the profile
c_u	absolute circumferential velocity component
c_m	absolute meridional velocity component
β_∞	average rel. flow angle
w_∞	average rel. velocity
c_L	lift coefficient
	angle of attack
	angle between resulting force and lift force



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 a single blade but is dependent on the whole blade cascade.

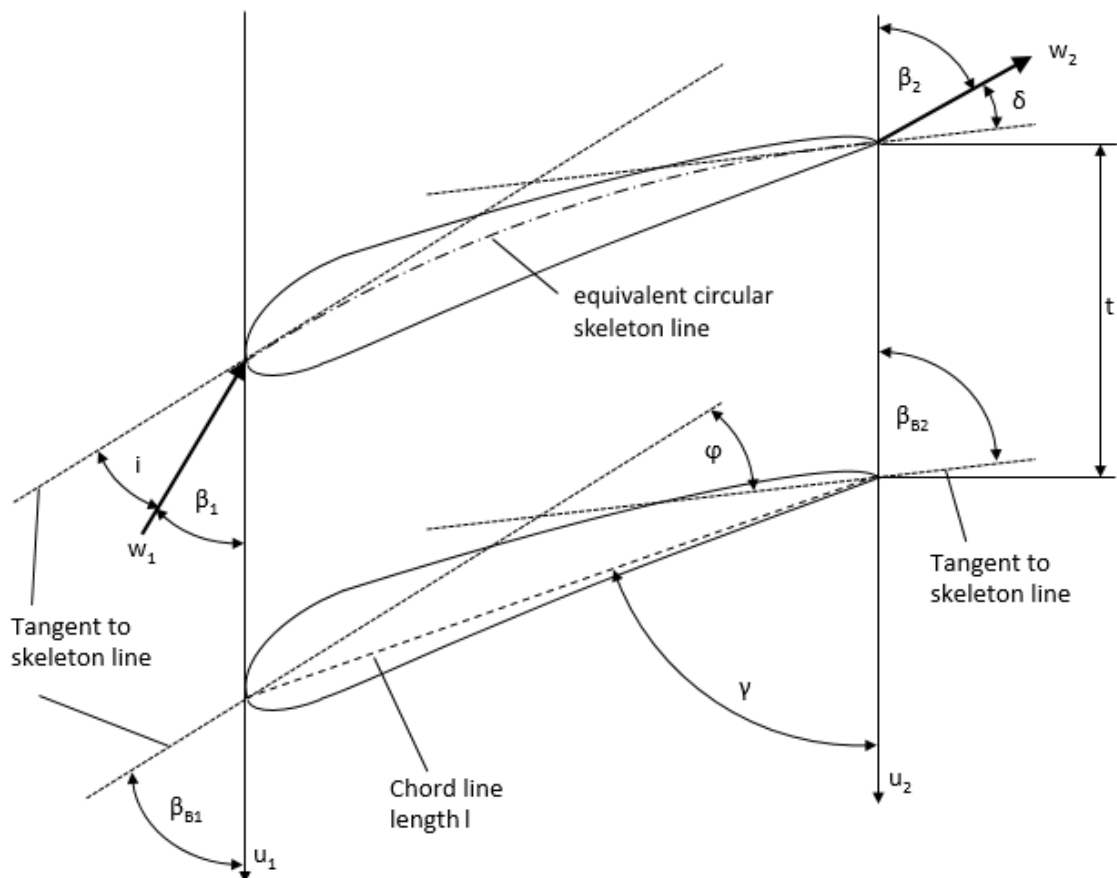
[Lieblein](#)^[568] carried out systematic wind tunnel investigations on the swirl change properties of the profiles of the NACA 65 series. The meaning of the used entities is given in the following table

	stagger angle
l/t	solidity (chord length/pitch)
	Angle of relative flow
B	Blade angle (of the equivalent circular skeleton line)

- camber angle
 u circumferential velocity
 w Relative velocity
 i Incidence angle: $i = \beta_1 - \alpha_1$
 Deviation angle: $\delta = \beta_2 - \alpha_2$

Three limitations apply for this approach:

- The maximum relative thickness must be $d/l < 0.1$.
- The Reynolds-Number must be $Re_l > 2 \cdot 10^5$.
- 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

The basic approach is as follows: with the specified solidity the skeleton length is calculated. With the relative flow angle β_1 (from [cu-specification](#)^[460]) 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 blade angles at leading and trailing edge are known. Note: The blade angles are applied to the equivalent circular skeleton line with the radius:

$$r_{eq} = \frac{l}{2 \cdot \sin \frac{\beta_{B2} - \beta_{B1}}{2}}$$

From the blade angles the stagger angle can be determined by:

$$\gamma = \frac{\beta_{B1} + \beta_{B2}}{2}$$

The dimensionless skeleton line used for the generation of the NACA 65 series profile is described by the following equation:

$$\frac{y_{sl}}{l} = -\frac{c_{fl}}{4\pi} \left[\left(1 - \frac{x}{l}\right) \cdot \ln \left(1 - \frac{x}{l}\right) + \frac{x}{l} \ln \left(\frac{x}{l}\right) \right],$$

with c_{fl} the theoretical lift coefficient:

$$c_{fl} = \frac{2\pi}{\ln(2)} \tan\left(\frac{\varphi}{4}\right)$$

Since this skeleton line is perpendicular at the very beginning (LE) and end (TE) to the chord line, blade angles at these locations derived by tangent angles are not reasonable. This is the reason for applying the equivalent circular skeleton line for the determination of β_{B1} and β_{B2} .

7.4.2 Blade profile

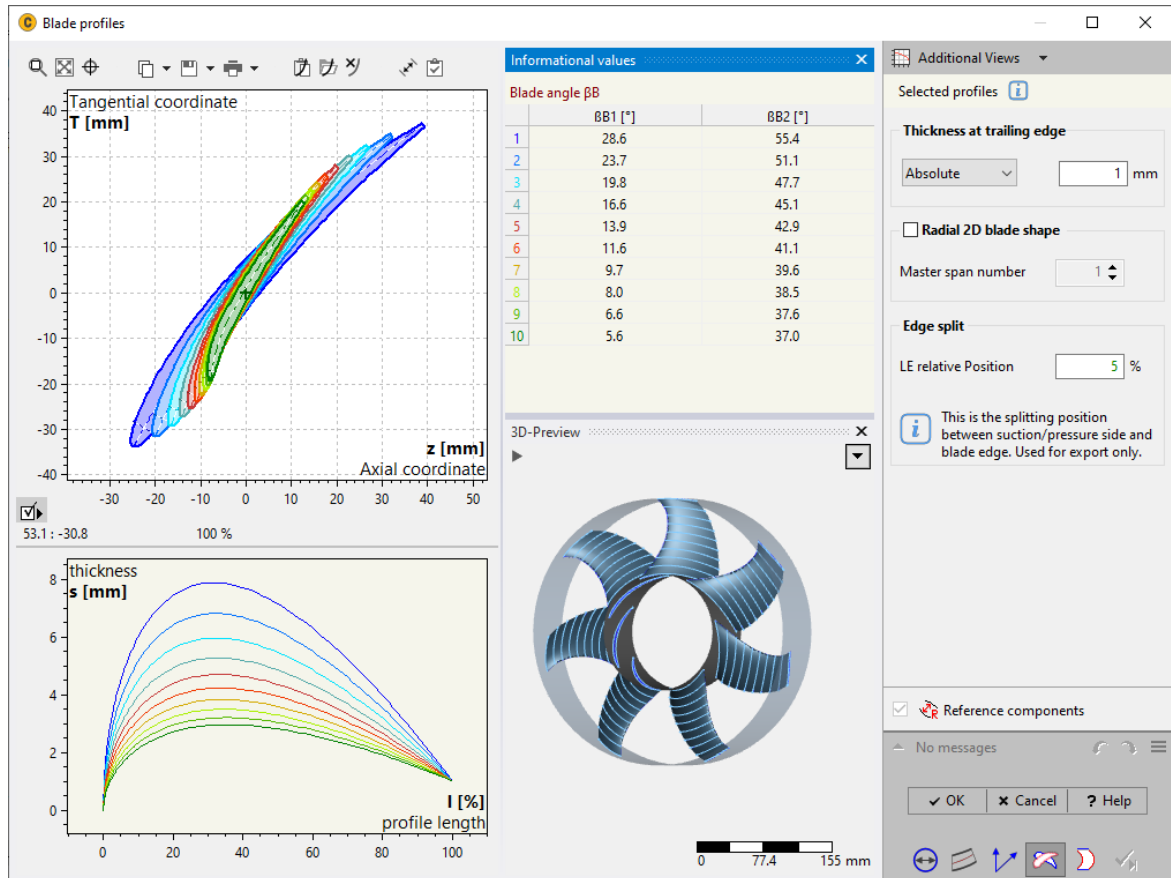
? IMPELLER | Blade profiles



To create 3D blade profiles the specified or calculated values from the [Blade properties](#)^[457] are used:

- Profile shape based on [profile selection](#)^[463]
- Chord length (scaling) and Stagger angle (rotation) of each profile at the respective span position based on [profile properties](#)^[466]

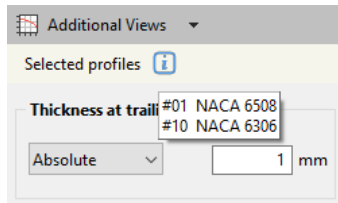
The resulting 2D profiles are displayed top left in the dialog whereas the thickness distribution at each span location can be found below.



The following information can be displayed using the "Additional views" button:

- **Informational values:** resulting blade angles at leading (β_{B1}) and trailing edge (β_{B2}). In case profiles of the NACA 65 series (see [Lieblein method](#)^[473]) have been used, blade angles according to the NACA 65 skeleton line specification are given. Because of this specification blade angle values especially at the leading edge may differ from those that would be derived by getting the skeleton line from upper and lower site averaging of the profile.
- **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](#) ¹³³.

Relative	Absolute
Thickness is chord length times relative thickness	Thickness is equal the absolute thickness

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](#) ⁴⁵⁷.

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

? IMPELLER | 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.

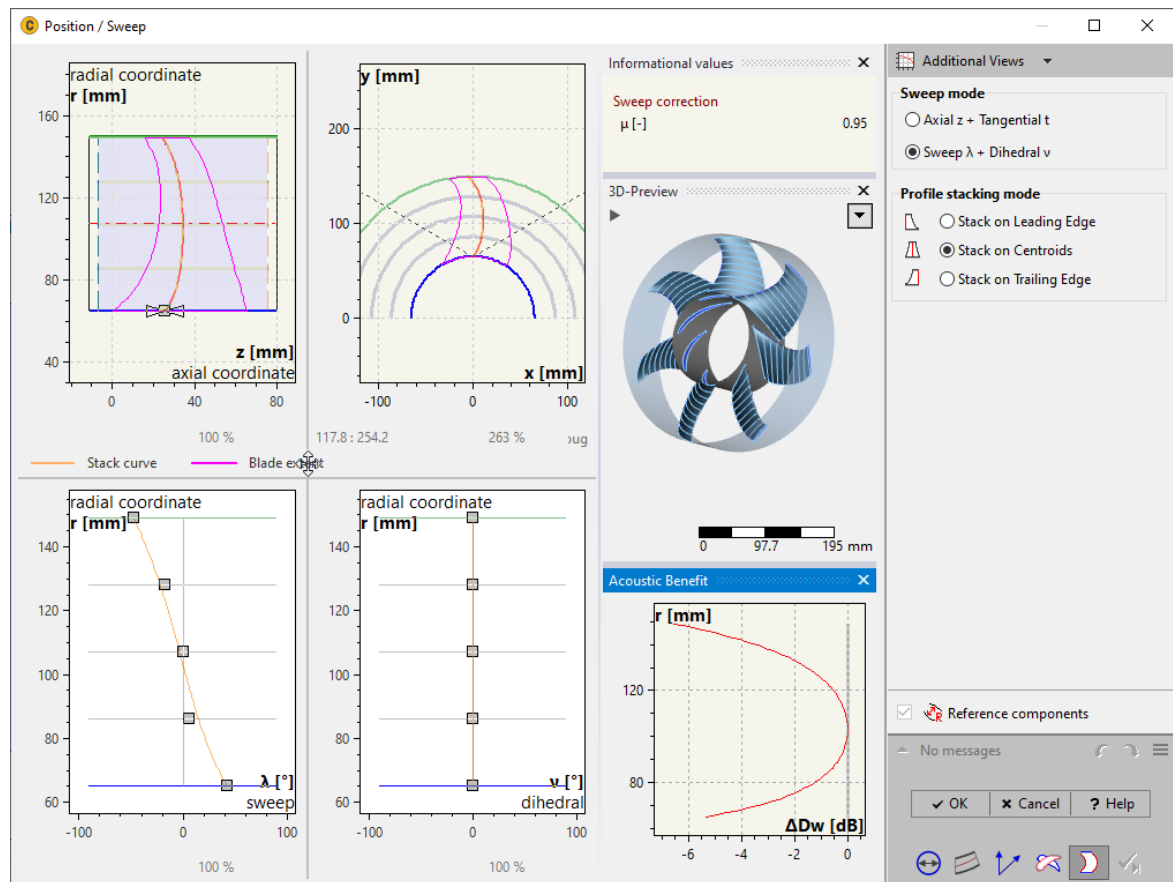
In default configuration this design step does not generate any sweep by aligning the centroid points of all profiles exactly in radial direction. You can return to an unswept configuration at any time by using the **Reset sweep curve** option.

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 μ 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](#)⁴⁶⁶ 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

$$L_W|_{\lambda \neq 0} - L_W|_{\lambda = 0} = 10 \cdot \lg(\cos(\lambda)^4) \text{ dB}$$

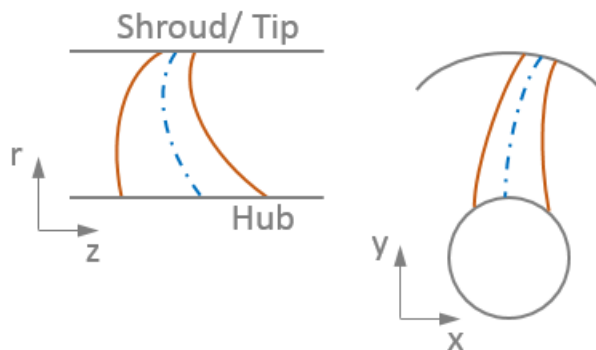
Sweep mode

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:

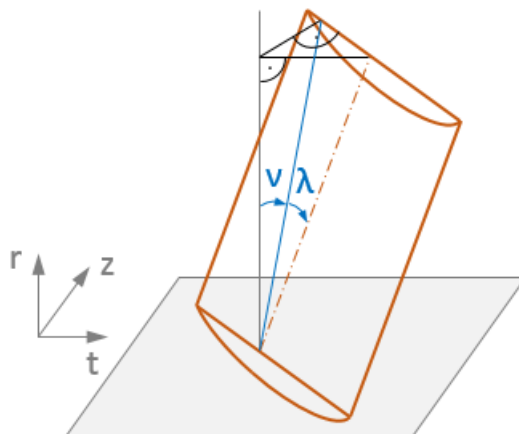
- **Axial z + Tangential t**

Sweep projected in meridional and axis-normal view.
This view also shows the blade outline.



- **Sweep + Dihedral (default)**




: Incidence not perpendicular to blade axis, blade area nevertheless in flow direction
: Blade plane not perpendicular on hub, defines V-positioning



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](#)^[473] 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

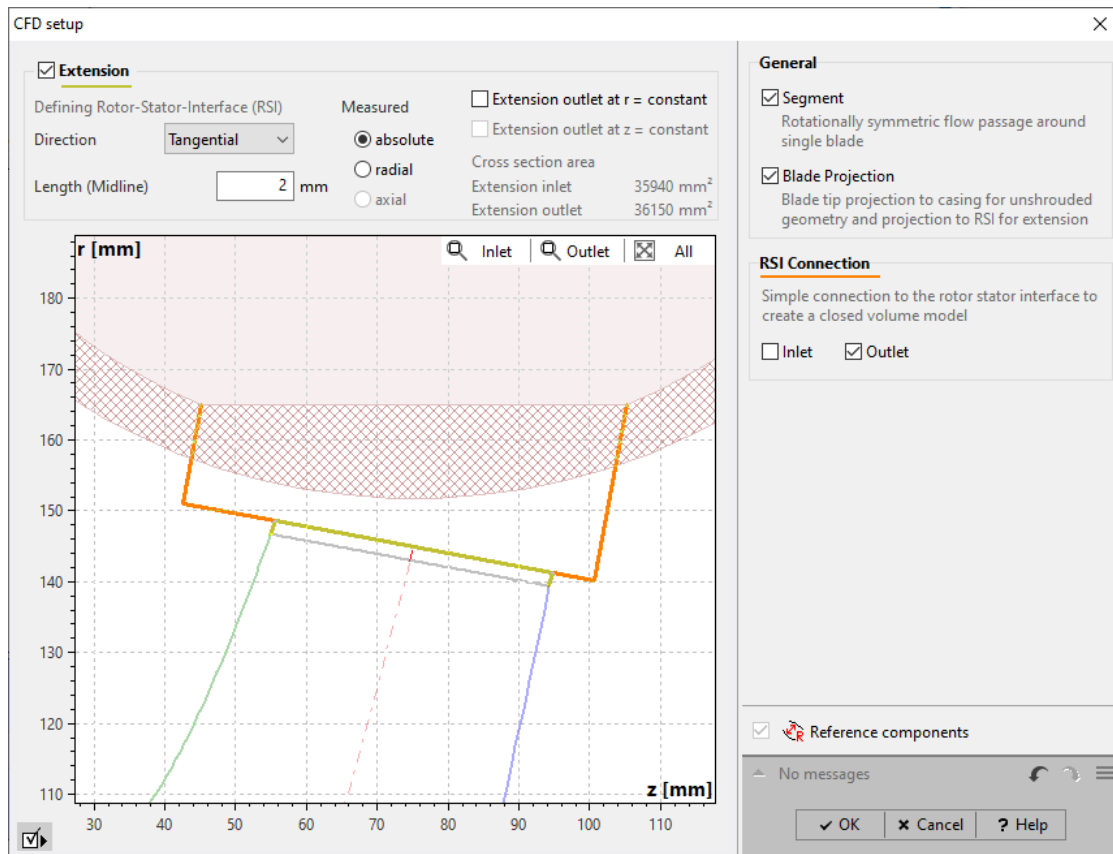
7.5 CFD setup

? IMPELLER | CFD setup



The designed geometry can be extended by **virtual** elements for flow simulation (CFD).

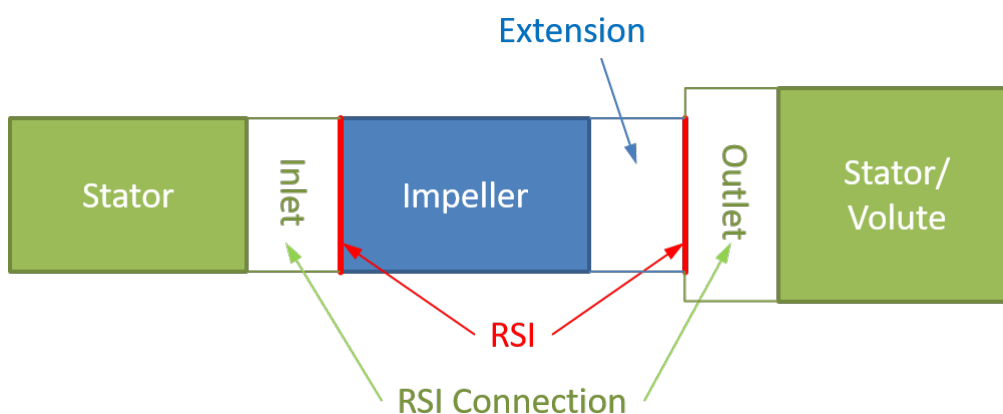
- [Extension](#)^[479]
- [Segment](#)^[482]
- [Blade projection](#)^[484]
- [RSI connection](#)



7.5.1 Extension

The designed geometry can be extended in meridional direction at the outlet.

The 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.



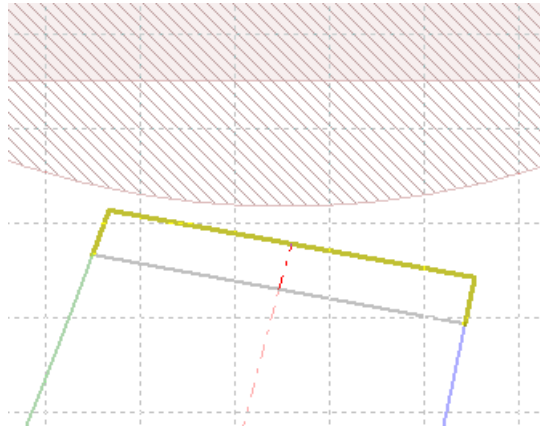
Using the extension is recommended, because otherwise the trailing edges of the blades would just lie on the rotor-stator interface, which can cause both meshing problems and numerical simulation errors. Meshing problems could occur, especially for small values of the blade angle β_2 .

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).

The designed extension will be displayed in the diagram automatically.



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](#)^[356] checkbox in the meridional contour dialog.

Possible warnings

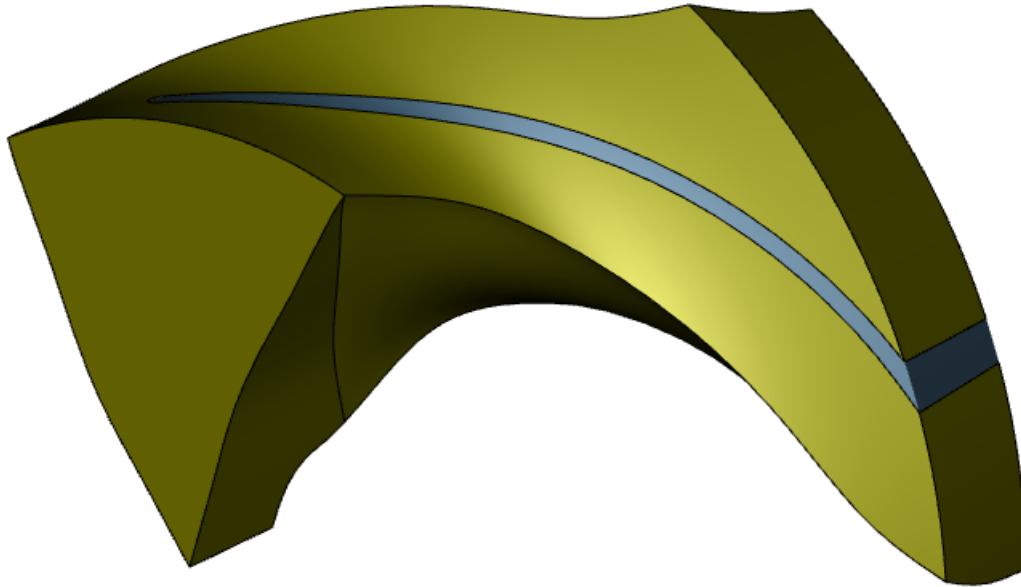
Problem	Possible solutions
Length of extension is smaller than the model tolerance. This is problematic for "Flow domain" creation during model finishing.	
The length of the extension is smaller or equal to the distance tolerance ^[486] . This might cause	If geometrical problems occur, change the distance tolerance or the length of the extension.

Problem	Possible solutions
geometrical defects when sewing faces during Model finishing ⁴⁸⁷ .	
Extension outlet has nearly constant radius. Selecting "Extension outlet at r = constant" is recommended.	
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.	Set the endpoints of the hub and shroud extension to the same radius by checking the "Extension outlet at r = constant" checkbox.
Extension outlet is nearly vertical. Selecting "Extension outlet at z = constant" is recommended.	
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.	Set the endpoints of the hub and shroud extension to the same z-coordinate by checking the "Extension outlet at z = constant" checkbox.
Extension is overlapping neighboring component.	
The interface defined by hub and shroud extension overlaps neighboring component.	Reduce extension length or increase gap between impeller and static neighboring component.
Extension is not well defined.	
Under certain circumstances it is possible that geometric interface and CFD-interface (extension) are misaligned.	<p>Check configuration of extension regarding:</p> <ul style="list-style-type: none"> • intersection • matching • inversion (negative extension) <p>See diagram for its visual representation.</p>

7.5.2 Segment

The segment is the flow passage around a single blade and represents the smallest rotation-symmetric part of the impeller.

The middle of blade pressure and suction side of two neighboring main blades forms the segment boundary. These periodic surfaces are extended into the inlet and outlet area.

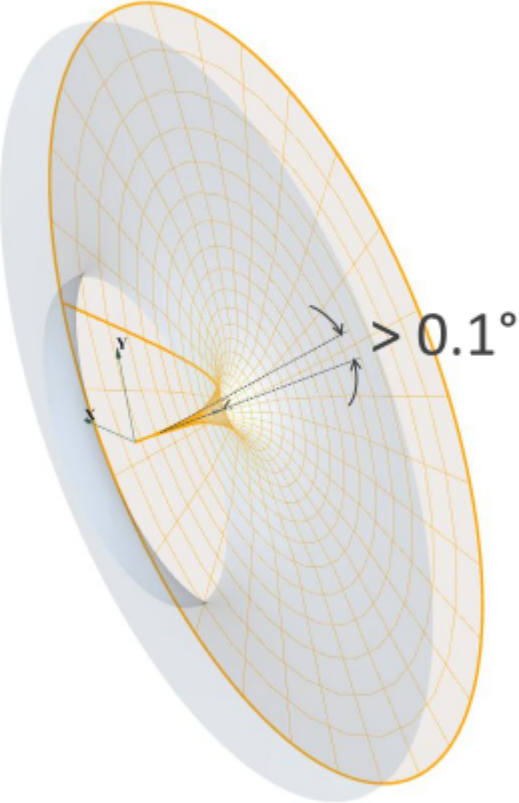


3D Model

In the 3D model the [extension](#)^[479] is part of the impeller segment solid. The rotor-stator-interface (RSI) is the border to the neighboring component. On the other side the [RSI connection](#)^[485] is part of the neighboring component solids.

Possible warnings

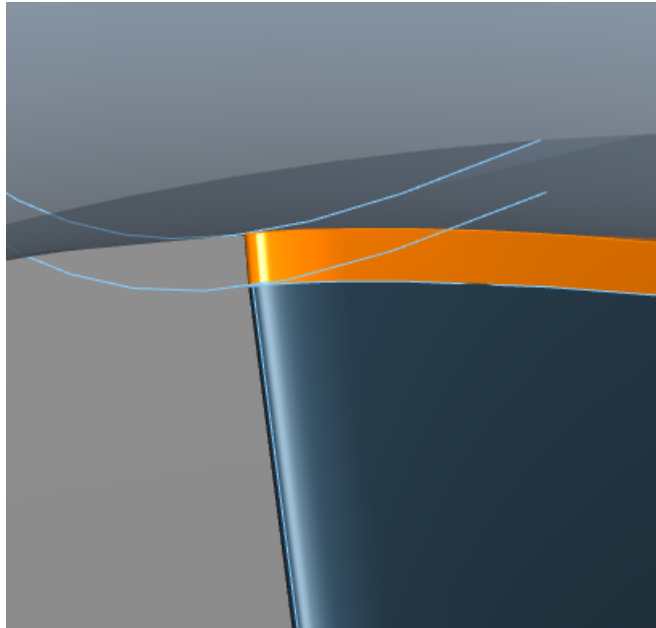
Problem	Possible solutions
Generation of segment may fail due to meridional contour.	
The angle between rotation axis and hub contour is too small (lower than 0.1 degree).	Set a value for hub diameter higher than zero or round the hub contour to avoid a geometry with needle form.

Problem	Possible solutions
	

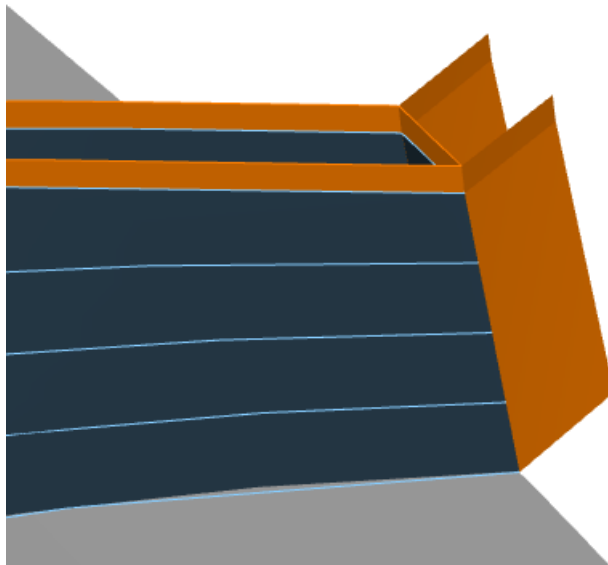
7.5.3 Blade projection

Blade projection can be used for meshing purposes.

In case of an unshrouded impeller the outer blade profile is projected onto the casing.

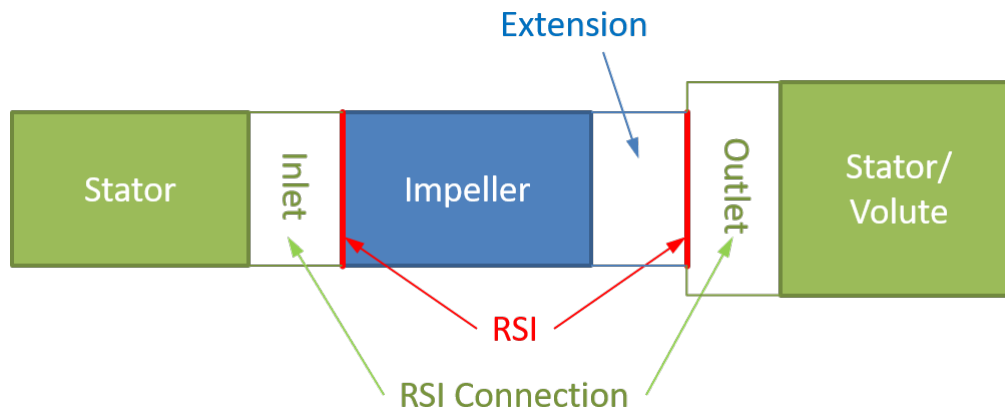


If an [Extension](#)⁴⁷⁹ exists, the blade trailing edge is projected onto the Rotor-Stator-Interface (RSI).

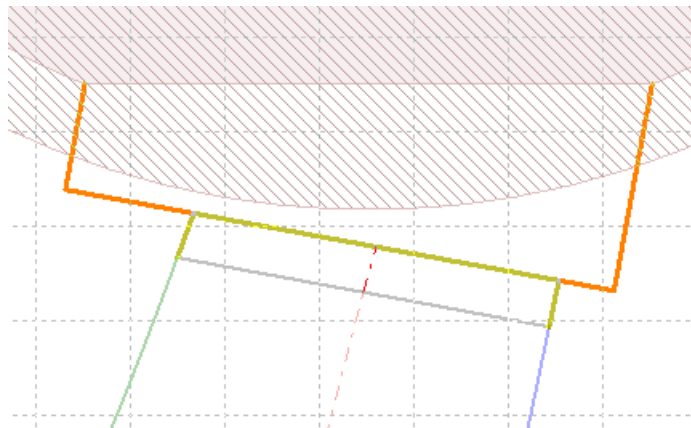


7.5.4 RSI connection

The RSI connection is intended to close the gap between impeller and static components. 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 will be displayed in the diagram automatically.



(see also [Extension](#) ⁴⁷⁸)

Possible warnings

Problem	Possible solutions
RSI connection at Inlet/ Outlet cannot be generated.	

Problem	Possible solutions
RSI connection can be generated only if a gap between components exists.	Disable RSI connection or perform corrections at neighbouring component interface if necessary.

7.6 Model settings

? IMPELLER | 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

Model settings [X]

3D Model | Point Export

Data points
used for 3D modeling

↓ Default

Meridian	Hub, Shroud	<input type="text" value="40"/>	20 .. 80
Blade	Mean line	<input type="text" value="50"/>	10 .. 50
	Profile	<input type="text" value="100"/>	50 .. 200

Distance tolerance
for joining surfaces and creating solids

Tolerance mm

▲ No messages

Model settings [X]

3D Model | **Point Export**

Data points
used for point-based export to external applications

Presetting Coarse | Middle | Fine

Meridian	Hub, Shroud	<input type="text" value="80"/>	20 .. 100
Blade	Mean line	<input type="text" value="80"/>	20 .. 100
	Pressure, Suction side	<input type="text" value="80"/>	20 .. 100
	Leading, Trailing edge	<input type="text" value="30"/>	10 .. 75

▲ No messages

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****Presetting**

Coarse

Middle

Fine

Select from 3 global presets.

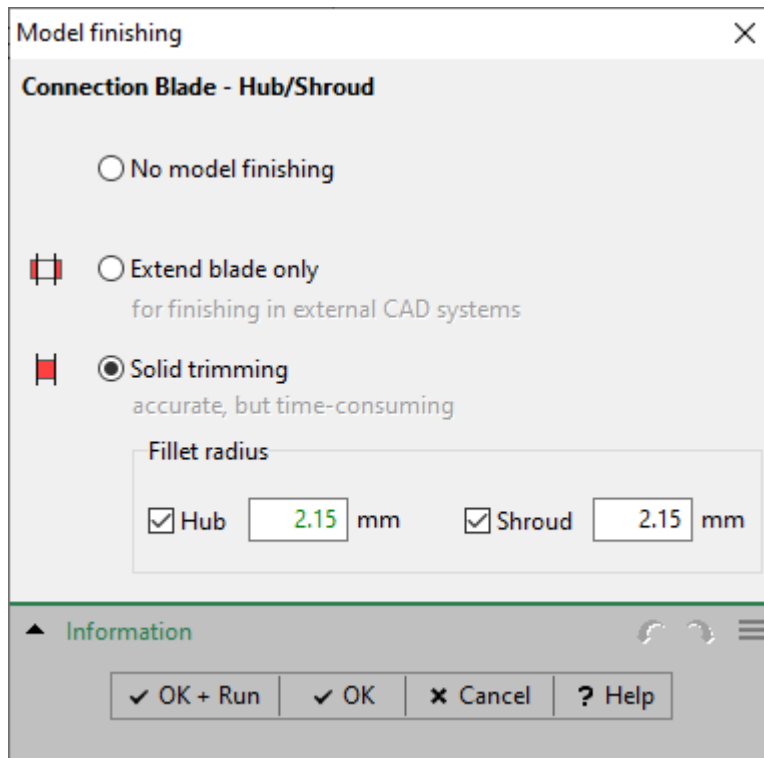
When a **new impeller** is created the model settings of the last opened impeller are carried over.

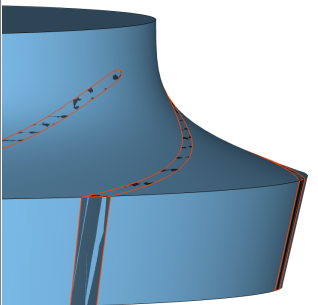

7.7 Model finishing

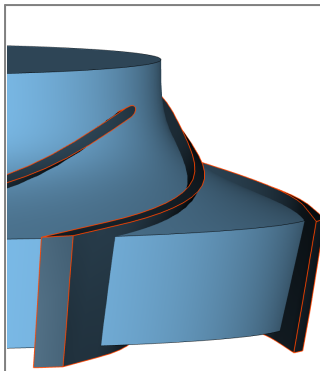
? IMPELLER | Model finishing



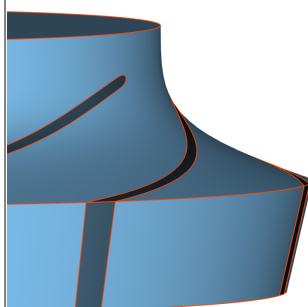
The dialog offers different possibilities to design the connection between blade, hub and shroud.



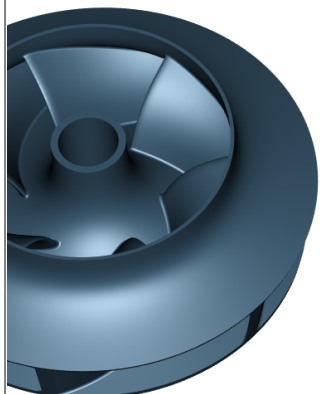
<p>No model finishing</p> 	
<p> Extend blade only</p>	<p>Extends blades through hub and shroud for later trimming in a CAD-system.</p> <p>Additionally, extends blades through leading/ trailing edge, if it is fixed³⁵⁶ as well as simple and trimmed⁴⁴⁷ on inlet/ outlet.</p>



Solid trimming



Meridian.Flow domain



Meridian.Material domain

Trims blades on hub, shroud and possibly leading and trailing edge.

The *solid trimming* affects only the **solids** (and solid faces) of *Meridian.Flow domain*, *Meridian.Material domain*, *CFD setup.Segment* and *Blade*.

Trimming is a **time-consuming** operation (up to 30 seconds).

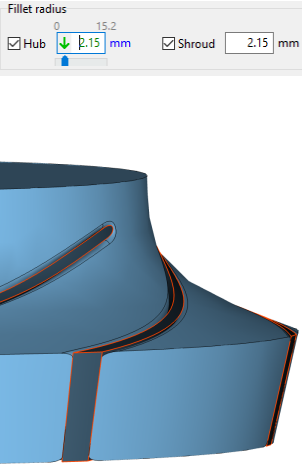
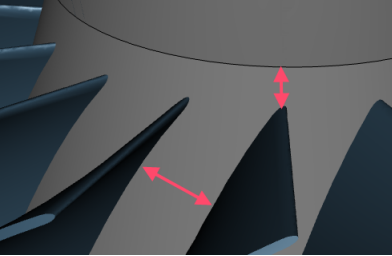

Because only solids are trimmed, **point-based exports cannot take advantage of this operation.**

Details:

Solid trimming is based on a [segment](#)^[482]. If no segment is defined, it is created temporarily, not visible to the user.

Internal workflow:

- The blades are extended (see [Extend blade only](#)^[488])
- A single blade is trimmed with *Meridian.Flow domain*
- From *Meridian.Flow domain*, a segment is cut. In this way the trimmed *Segment.Real geometry* is created.
- CFD setup option: If there is an [Extension](#)^[479] or [RSI connection](#)^[485], *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 *Meridian.Flow domain*.
- If [hub and shroud solids](#)^[360] have been defined, a *Meridian.Material domain* is created by fusing the blades and the solids of hub and shroud.

	<ul style="list-style-type: none"> • CFD setup option: If Blade projection¹⁸⁴ was chosen, the corresponding projection surfaces are exactly trimmed.
<p>Option: Blade root fillet</p> 	<p>The edges between blade and hub/ shroud are rounded.</p> <p>The fillet affects only the solids (and solid faces) of <i>Meridian.Flow domain</i>, <i>Meridian.Material domain</i> and <i>CFD setup.Segment</i>.</p> <p>The fillet radius should not be larger than the recommended value.</p> <p>Limitations</p> <div data-bbox="624 658 1018 913">  <p>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.</p> </div> <div data-bbox="624 987 1018 1153">  <p>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.</p> </div>

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](#)¹⁸⁴.

Possible warnings

Problem	Possible solutions
Extend/solid trimming may fail due to tangential difference between hub and shroud at leading/trailing edge, or low number of spans.	
Very low number of spans	Increase the number of spans ³⁸⁹ up to at least 4
Finishing type was reset to "No model finishing". Solid trimming impossible for nearly tangential contour at dH = 0.	
Solid trimming is not supported if the hub contour is nearly tangential to the Z-axis.	Change the hub contour ³³⁸ to form a bigger angle to the Z-axis.
Finishing type was reset to "No model finishing". Solid trimming impossible when blades exceed meridional boundaries.	
Solid trimming is not possible if the blade exceeds the meridional boundaries (caused by the blade thickness).	Change blade design so that it fits into meridional boundaries, e.g. change Blade edges ⁴⁴⁷
Fillets not supported.	
Fillets are not supported if solid trimming is not possible.	-
Fillets creation on shroud deactivated.	
Fillets on shroud are not supported for unshrouded designs.	-
Curvature of Leading edge/Trailing edge/Blade edges at Hub/Shroud could be too large for fillet creation.	
Sharp Blade edges could prohibit fillet creation between blade and hub/shroud and could cause long computation time respectively.	Less sharp design of blade edges or reduction/deactivation of fillet radius.
3D-Error: Finishing failed!	
Inlet (nearly) tangential to hub or shroud	Change Meridional contour ³³⁸ : Avoid tangentiality

Problem	Possible solutions
3D-Error: Finishing failed! (Could not extend blade)	
Extending the blade to a scaled Hub/ Shroud surface failed.	See solution of Error while extrapolating Blade to reach Hub/ Shroud surface ⁴⁵²
3D-Error: Finishing failed! (Could not trim solids of Blades and Flow domain)	
Trimming a single blade with <i>Meridian.Flow domain</i> failed.	Try using a different number of data points ⁴⁸⁶
3D-Error: Finishing failed! (Could not create fillet)	
Fillet creation at blade root failed (see limitations) ⁴⁹⁰	Decrease fillet radius or Deactivate fillet
3D-Error: Finishing failed! (Creating trimmed segment)	
Cutting a segment from <i>Meridian.Flow domain</i> failed.	Change the segment type ⁴⁸²
3D-Error: Finishing failed! (Fusing solids)	
Fusing of real geometry with CFD setup components (Extension or RSI connection) failed.	Increase the number of spans ³⁸⁹ or Remove Extension / RSI connection from CFD setup ⁴⁷⁸
3D-Error: Finishing failed! (Creating new Flow domain from segment)	
Creating a new <i>Meridian.Flow domain</i> from <i>Segment.Flow domain</i> failed.	Change the segment type ⁴⁸²
3D-Error: Finishing failed! (Creating Material domain from Flow domain)	
Creating a <i>Meridian.Material domain</i> failed.	Deactivate hub and shroud solid ³⁶⁰
3D-Error: Blade projection to RSI failed!	

Problem	Possible solutions
Projection of blade to RSI (Extension) failed.	Change CFD setup ⁴⁷⁶ : Modify Extension or remove Blade projection
3D-Error: Blade tip projection to casing failed!	
Projection of blade to casing (shroud) failed.	Change CFD setup ⁴⁷⁶ : Remove Blade projection or RSI connection

Part



8 Stator

? 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](#) 496
- [Meridional contour](#) 502
- [Blade properties](#) 503
- [Blade mean lines](#) 507
- [Blade profiles](#) 510
- [Blade edges](#) 510
- [Model finishing](#) 511
- [Model settings](#) 511
- [CFD setup](#) 510

Possible warnings

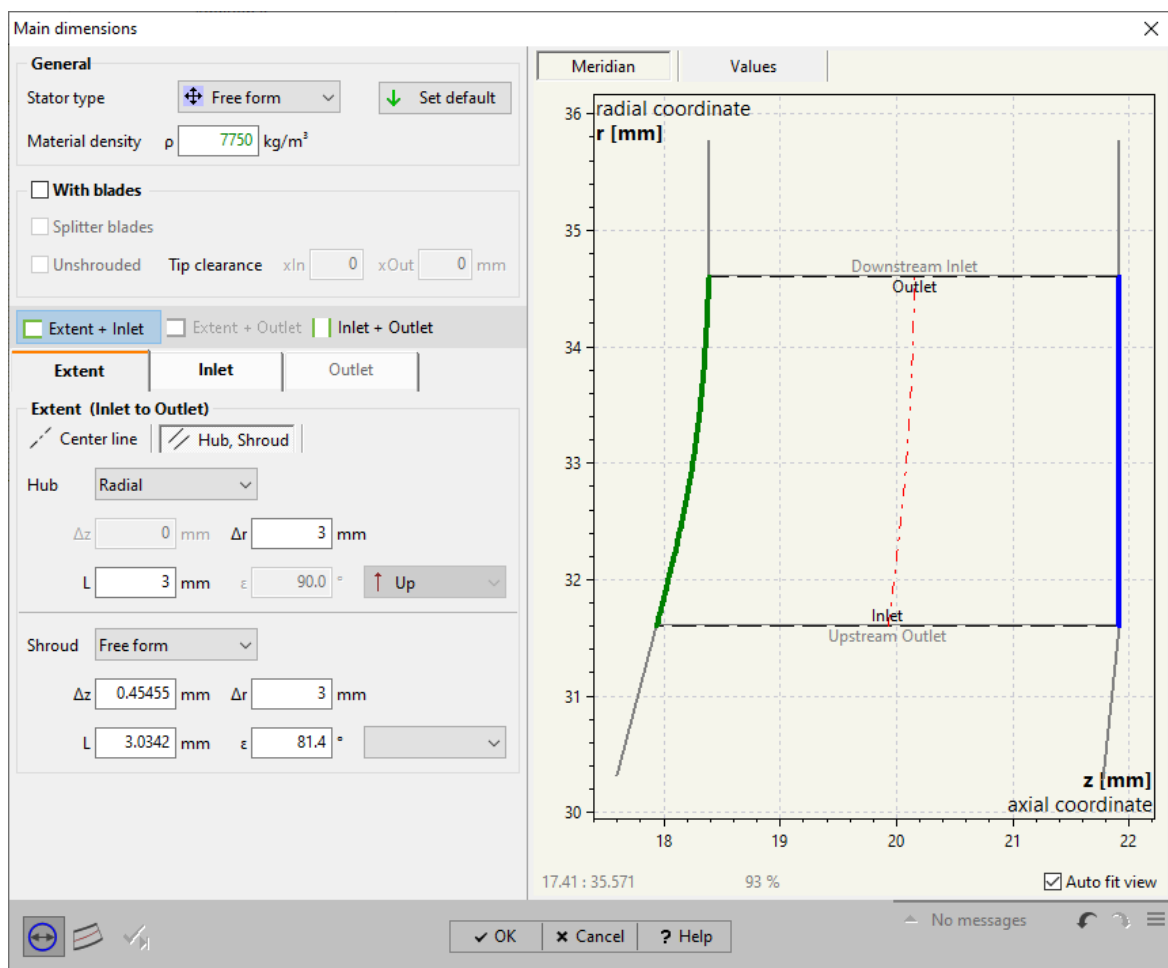
Problem	Possible solutions
Neighboring blades are intersecting each other.	
(see message)	<p>Numerous details of the design influence the blade shape. Some examples for possible solutions:</p> <ul style="list-style-type: none"> • Modify main dimensions

Problem	Possible solutions
	<ul style="list-style-type: none"> • Reduce number of blades • Reduce blade wrap angle • Reduce blade thickness

8.1 Main dimensions







? STATOR | Main dimensions

The Main Dimensions menu item is used to define main dimensions of the stator.





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 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](#)^[499], [Inlet](#)^[501] or [Outlet](#)^[502].
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

Problem	Possible solution
Hub/ Shroud/ Midline length is zero (invalid geometry).	
The extent ^[499] of the stator is 0 at hub, shroud or midline.	Specify a reasonable length value or remove the stator completely.
Thermodynamic state could not be calculated for given main dimensions. [for compressors and turbines only]	
The dimensions might be too tight for the specified mass flow and inlet conditions.	Increase the dimensions (width etc.) or change the Global setup ^[86] (e.g. decrease mass flow). If the stator is vaned, its blade angles ^[503] can be changed too.

8.1.1 Extent

Stator extent has to be considered in relation to its [inlet](#)^[501] and [outlet](#)^[502]. 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

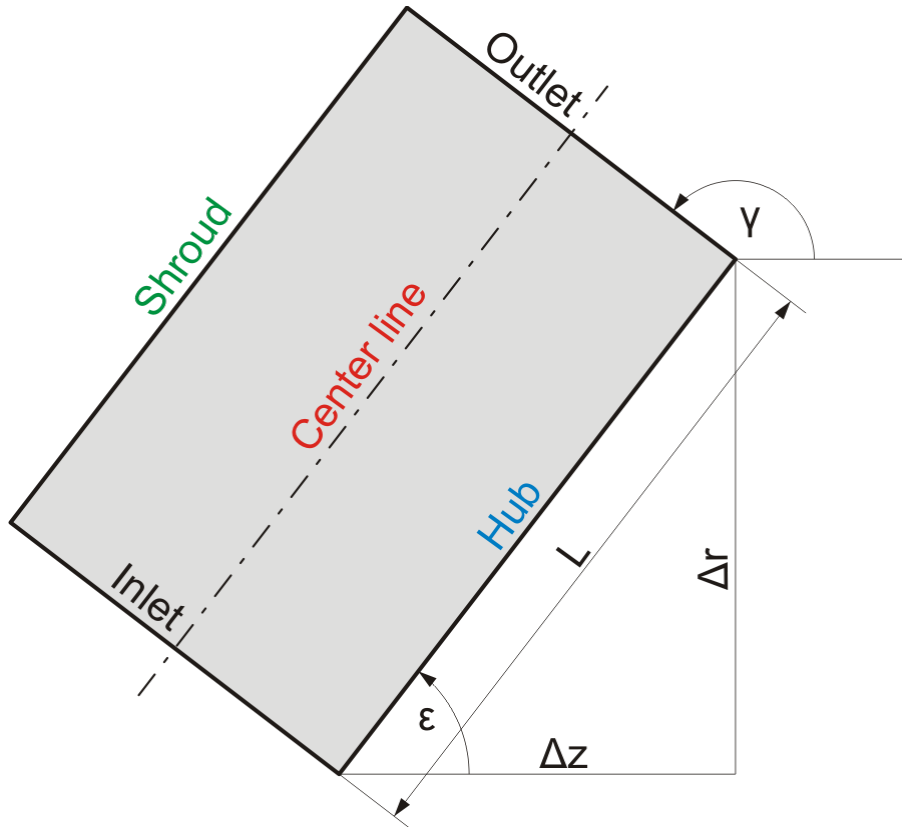
- preselection of extension direction: Radial, Axial, Tangential (to outlet of previous component), Free form
- Definition of axial extension z and radial extension r
or
length L and angle of center line to horizontal direction
- Definition of end cross section (Inlet or Outlet) by width b and angle to horizontal direction

2. Hub, Shroud

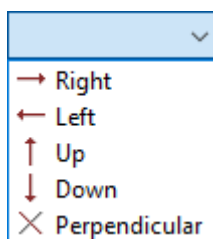
separately for hub and shroud:

- 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 Δz and radial extension Δr
or
length L and angle of hub/shroud to horizontal direction



The angles ϵ and γ are defined by 0° horizontal right and rising in counter clockwise direction (mathematical positive). A menu with some default angles is supporting angle input:



0°

180°

90°

270°

Perpendicular: perpendicular to inlet or outlet cross section

Parallel: parallel to inlet or outlet cross section

Depending on the type of [geometric coupling](#)^[42] 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 at outlet then the extent of the stator cannot be specified because it's clearly defined by its neighbor.

Information

Design point	Design point information, see Global setup ^[86]
Ratio outlet to inlet	
Diameter ratio	$d_{\text{Out}}/d_{\text{In}}$
Width ratio	$b_{\text{Out}}/b_{\text{In}}$
Area ratio	$A_{\text{Out}}/A_{\text{In}}$
Inlet area	A_{In}
Outlet area	A_{Out}

8.1.2 Inlet

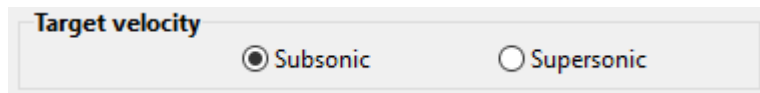
Here the inlet of the stator can be defined.

If the outlet can be modified, it will be updated by addition of extent to inlet. Otherwise the extent will be adapted.

→ Details: see [geometric coupling](#)^[42]

[Compressors and Turbine only]

Radio buttons are available that allow to define whether the flow in the stator inlet is supposed to be subsonic or supersonic. This will influence the values displayed in the information panel as well as the [isentropic Mach number](#)^[412] distribution.



A rectangular control panel with a light gray background. It has a title 'Target velocity' in bold black text at the top left. Below the title, there are two radio buttons. The first is labeled 'Subsonic' and is selected, indicated by a filled black circle. The second is labeled 'Supersonic' and is not selected, indicated by an empty circle.

8.1.3 Outlet

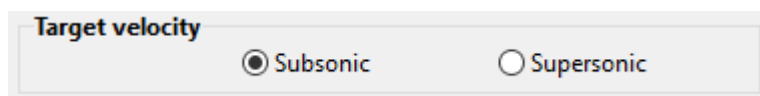
Here the outlet of the stator can be defined.

If the inlet can be modified, then it will be updated by subtraction of extent from outlet. Otherwise the extent will be adapted.

→ Details: see [geometric coupling](#)^[42]

[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](#)^[412] distribution.



A rectangular control panel with a light gray background. It has a title 'Target velocity' in bold black text at the top left. Below the title, there are two radio buttons. The first is labeled 'Subsonic' and is selected, indicated by a filled black circle. The second is labeled 'Supersonic' and is not selected, indicated by an empty circle.

8.2 Meridional contour

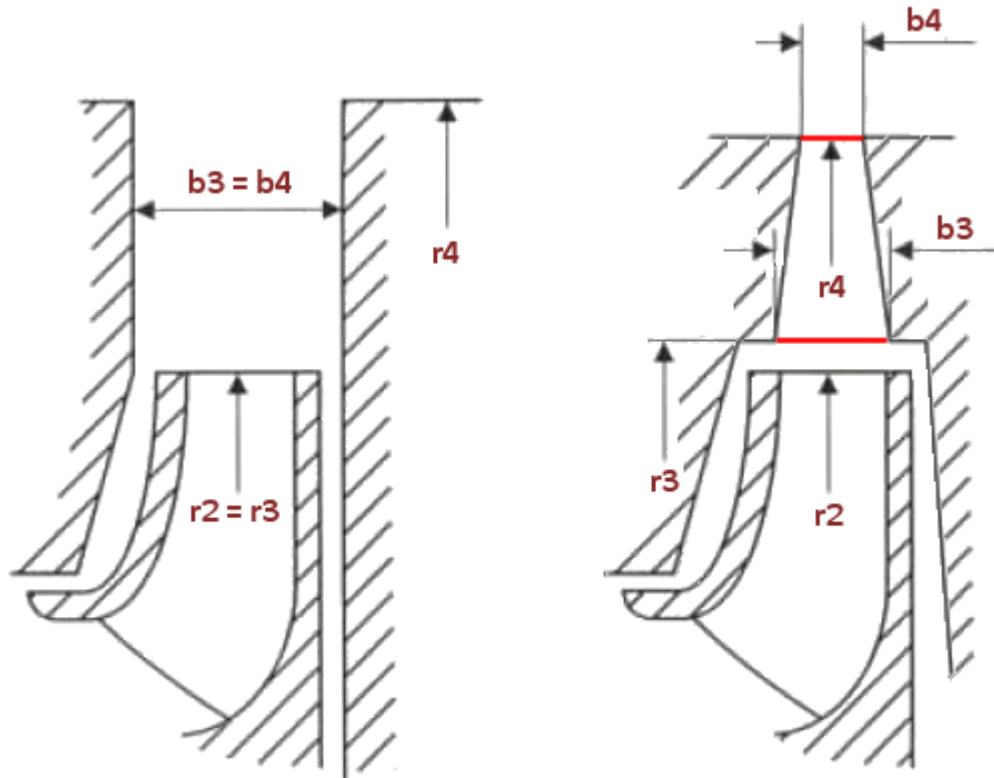
? STATOR | Meridional contour



In principle, the same features are available as for the [meridional design](#)^[338] of impellers.

The endpoints of hub and shroud curves are fixed by [main dimensions](#)^[496] and cannot be modified here.

For "Radial diffuser" type of stators (see [main dimensions](#)^[496]) the following geometrical dimensions are defined:



8.3 Blade properties

? STATOR | Blade properties

In principle, the same features are available as for the [blade properties](#)^[371] of impellers.

To support the selection of a suitable blade count a separate [dialog](#)^[505] can be used, which can be started by pressing the button right beside the edit field.

The outlet angles TE 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.

Two additional special **blade shapes** are available for "Radial diffuser" type stators (see [Main dimensions](#)^[496]):

1. Log. Spiral + Straight 2D

The inlet section of the vanes without overlapping is noneffective and configured as a logarithmic spiral (similar to spiral casing).

The diffuser part in the overlapping area is straight. The transition point between these areas can be moved along the logarithmic spiral curve (see [mean line](#)^[507]).

2. Circular + Free-form 2D

The inlet section of the vanes without overlapping is configured as a circular arc with the boundary conditions inlet radius r_3 , inlet angle β_3 and ideal throat width a_3 .

The diffuser part in the overlapping area 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 [mean line](#)^[507]).

Calculation of throat width a_3 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, whereat the deceleration is increased by using the factor f_{a3} (1.1...1.3).

$$a_3 = f_{a3} \cdot \frac{d_3}{2} \left\{ \exp \left(\frac{Q}{b_3 c_{u2} r_2 z} \right) - 1 \right\}$$

b) Deceleration

Alternatively one can use the deceleration ratio c_{3q}/c_2 (0.7...0.85) for throat width calculation.

$$a_3 = \frac{Q}{z b_3 c_2} \cdot \left(\frac{c_2}{c_{3q}} \right)$$

Trailing edge angle β_{TE} is a result of mean line design for these special blade shapes and therefore cannot be specified explicitly ("var.").

3. Straight 2D (axial)

A diffuser opening angle ϑ is calculated and displayed for information in accordance to [Bohl](#)^[566] with:

$$\tan(\vartheta) = \frac{a_4 - a_5}{l},$$

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

Problem	Possible solution
A reasonable thermodynamic state could not be calculated. [for compressors and turbines only]	
The dimensions might be too tight for the specified mass flow and inlet conditions. The chosen blade angles might also yield this state.	Increase the dimensions (width etc.), change blade angle or change the Global setup ^[86] (e.g. decrease mass flow).
Diffuser opening angle bigger than 10°. Flow separation possible. [for Straight 2D (axial) only]	
The diffuser opening angle is too big and might give room for flow separation.	Change s_{BLE} , s_1 , s_2 , LE or TE radius (in meridian)

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.

Impeller-Stator-Interference

Number of blades

Impeller blades z_I

Stator blades z_{II}

Recommended 8,10,14,15,16

Periodicity

$m = |v_I z_I - v_{II} z_{II}|$

v_I	v_{II}	m
1	1	8
1	2	22
1	3	36
2	1	2
2	2	16
2	3	30
3	1	4
3	2	10
3	3	24

Minimum m-value

- $m=0$: not allowed
- $m=1$: not allowed for $v_I=1..2$
- $m=2$: unfavorable but acceptable

✓ OK ✕ Cancel ? Help

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 the impeller, 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 :

- impeller periodicity $p_I = v_I \cdot z_I$
- stator periodicity $p_{II} = v_{II} \cdot z_{II}$
(v = 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:

$$m = |p_I - p_{II}| = |v_I \cdot z_I - v_{II} \cdot z_{II}|$$

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 ($v_I=1$; $v_I=2$) due to unacceptable shaft vibration, if possible also in third order ($v_I=3$).
- $m = 2$ 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 ($v_I = 1..3$) and ($v_{II} = 1..3$) are calculated and displayed in the table. Values $m=0$ are marked in red color, $m=1$ in orange and $m=2$ in yellow.

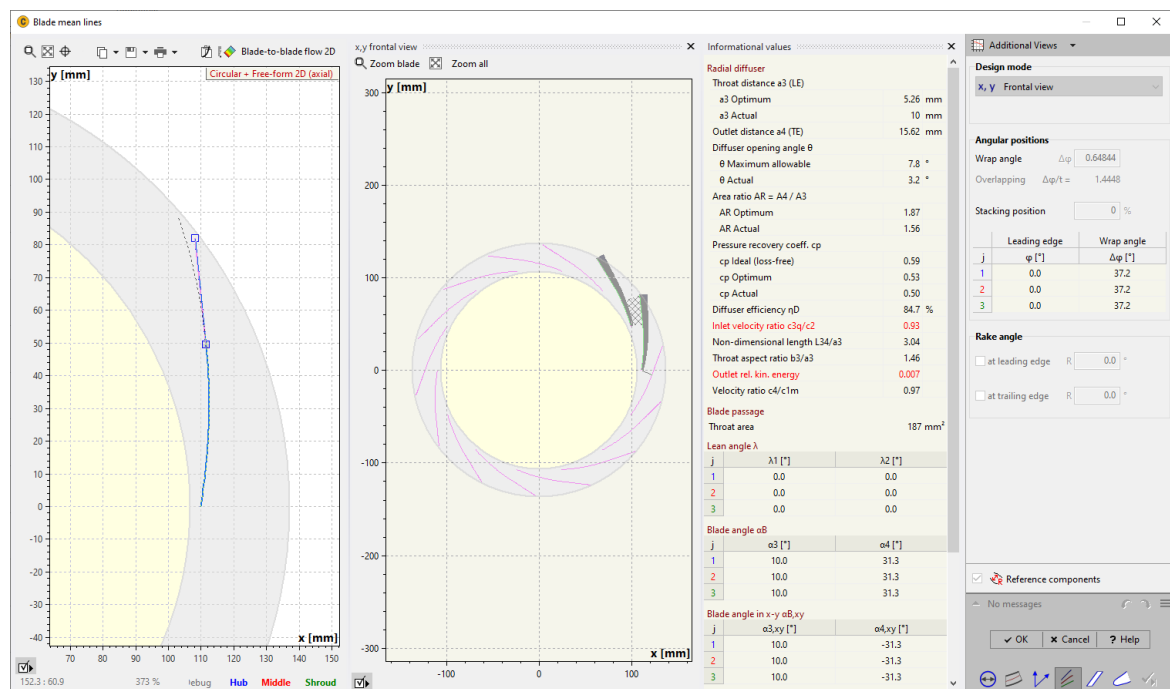
The recommended stator blade count according to the current number of impeller blades are represented below the input field.

8.4 Blade mean lines

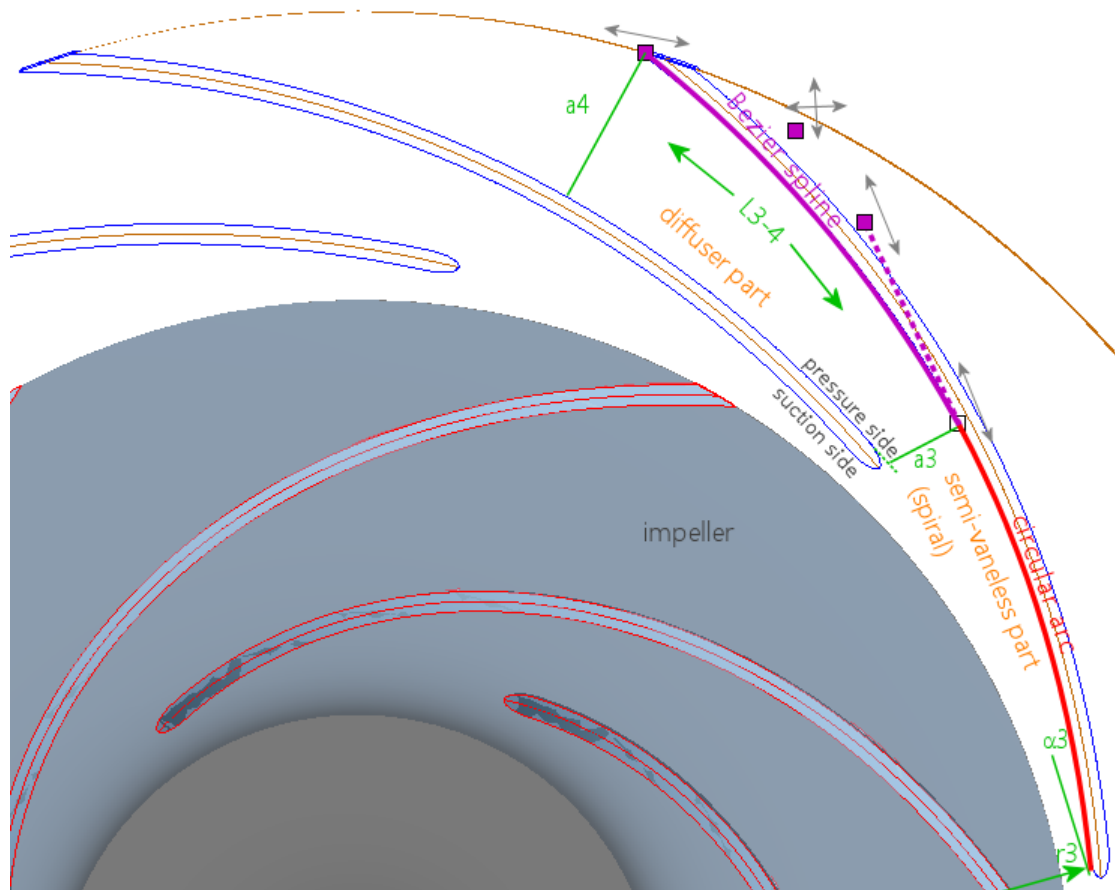
? STATOR | Blade mean lines

In principle, the same features are available as for the [mean lines](#)^[405] 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 of 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](#)^[503]), if the thickness distribution was not defined yet. Otherwise the thickness distribution defined in the [blade profile](#)^[510] design is used. In the later blade profile design the thickness is added to one side of the mean line only.



Diffuser 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.

Name	Description	Definition/ recommended range
Throat distance a_3 (LE)	Throat width at inlet (leading edge)	
a_3 Optimum *	Optimal value: average of calculation by const. swirl and deceleration ratio	see blade properties ⁵⁰³
a_3 Actual	Actual value: shortest distance from vane leading edge to neighboring vane	
Outlet distance a_4	Shortest distance from vane trailing edge to neighboring vane	
Diffuser opening angle	Allowable diffusion angle	

Maximum allowable	Max. allowable value to avoid flow separation depending on equivalent inlet radius and length	$\vartheta_{\max} = 16.5^\circ \cdot \sqrt{\frac{R_{3,eq}}{L}}$ $R_{3,eq} = \sqrt{\frac{a_3 b_3}{\pi}}$
Actual	Actual value calculated by equivalent inlet radius, length, inlet and outlet area	$\vartheta_{eq} = \frac{R_{3,eq}}{L} \left(\sqrt{\frac{A_4}{A_3}} - 1 \right)$
Area ratio $AR=A_4/A_3$	Area or deceleration ratio	$A_R = \frac{A_4}{A_3}$
AR Optimum *	Optimal value	$A_{R,opt} = 1.05 + 0.184 \frac{L_{3-4}}{R_{3,eq}}$
AR Actual	Actual value	$A_R < 3$
Pressure recovery coeff. c_p	Pressure recovery of the diffuser identified by a dimensionless coefficient	$c_p = \frac{p_4 - p_3}{\frac{\rho}{2} c_3^2}$
c_p Ideal (loss-free) *	Pressure recovery in an ideal (loss-free) diffuser	$c_{p,id} = 1 - \frac{1}{A_R^2}$
c_p Optimum *	Pressure recovery for optimal area ratio A_R	$c_{p,opt} = 0.36 \left(\frac{L_{3-4}}{R_{3,eq}} \right)^{0.26}$
c_p Actual *	Pressure recovery in real diffuser (with energy losses)	based on test results; plotted in diagrams; target: $c_{p,act} = c_{p,opt}$
Diffuser efficiency η_D *	Diffuser efficiency	$\eta_D = \frac{c_p}{c_{p,id}} = \frac{c_p}{1 - \frac{1}{A_R^2}}$
Inlet velocity ratio c_{3q}/c_2	Inlet deceleration ratio	$c_{3q}/c_2 = 0.7 \dots 0.85$ for low specific speed
Non-dimensional length L_{34}/a_3	Ratio of length to throat width	$L_{3-4}/a_3 = 2.5 \dots 6$

Throat aspect ratio b_3/a_3	Ratio of inlet width to throat width	$b_3/a_3 = 0.8...2$
Outlet rel. kin. energy *	Kinetic energy of diffuser outlet; to minimize losses in the overflow channels of multistage machines	$\frac{c_4^2}{2gH_{opt}} = 0.02...0.04$
Velocity ratio c_4/c_{1m} *	Ratio of outlet velocity to inlet velocity of downstream impeller of multistage machines	$c_4/c_{m1} = 0.85...1.25$

* for radial diffusers of pumps only

8.5 Blade profiles

? STATOR | Blade profile 

In principle, the same features are available as for the [blade profiles](#) ^[438] 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](#) ^[507]).

For radial diffusers the same informational values as in the [mean line design](#) ^[507] are displayed in the Info area. The reason is the influence of the blade thickness to these numbers.

8.6 Blade edges

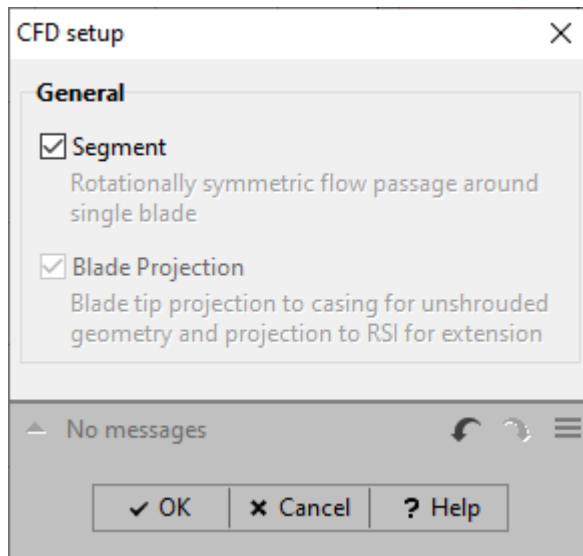
? STATOR | Blade edges 

In principle, the same features are available as for the [blade edges](#) ^[447] of impellers.

8.7 CFD setup

? STATOR | CFD setup 

The designed geometry can be extended by **virtual** elements for flow simulation (CFD).



The features are described in the [CFD setup](#)^[476] of impellers.

8.8 Model settings

? [STATOR | Model settings](#)



In principle, the same features are available as for the [model settings](#)^[486] of impellers.

8.9 Model finishing

? [STATOR | Model finishing](#)



In principle, the same features are available as for the [model finishing](#)^[487] of impellers.

Part



IX

9 Volute

? Volute



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](#) ⁵¹³
- [Cross section](#) ⁵²⁰
- [Spiral development areas](#) ⁵³¹
- [Diffuser](#) ⁵⁴³
- [Cut-water](#) ⁵⁵⁰
- [Model settings](#) ⁵⁶³
- [CFD setup](#) ⁵⁶²

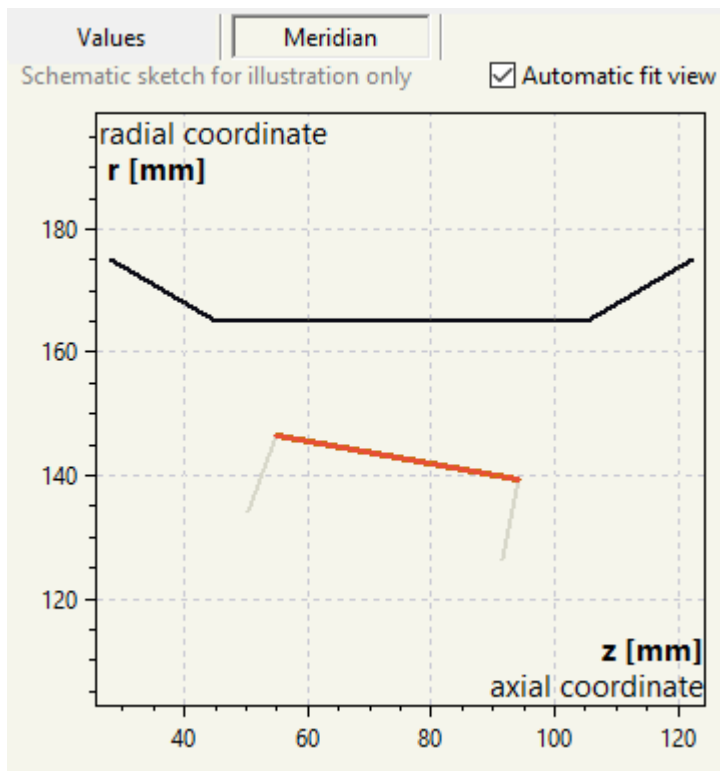
9.1 Setup + Inlet

? VOLUTE | Setup + Inlet



The first design step of the volute is to define the inlet side. It consists of 2 steps:

- (1) [Setup](#) ⁵¹⁴
- (2) [Inlet details](#) ⁵¹⁹



On page **Meridian** for the right panel you can find a meridional preview (z, r) of the designed volute inlet.

The outlet of the upstream component is represented schematically in gray, the interface position in brown.

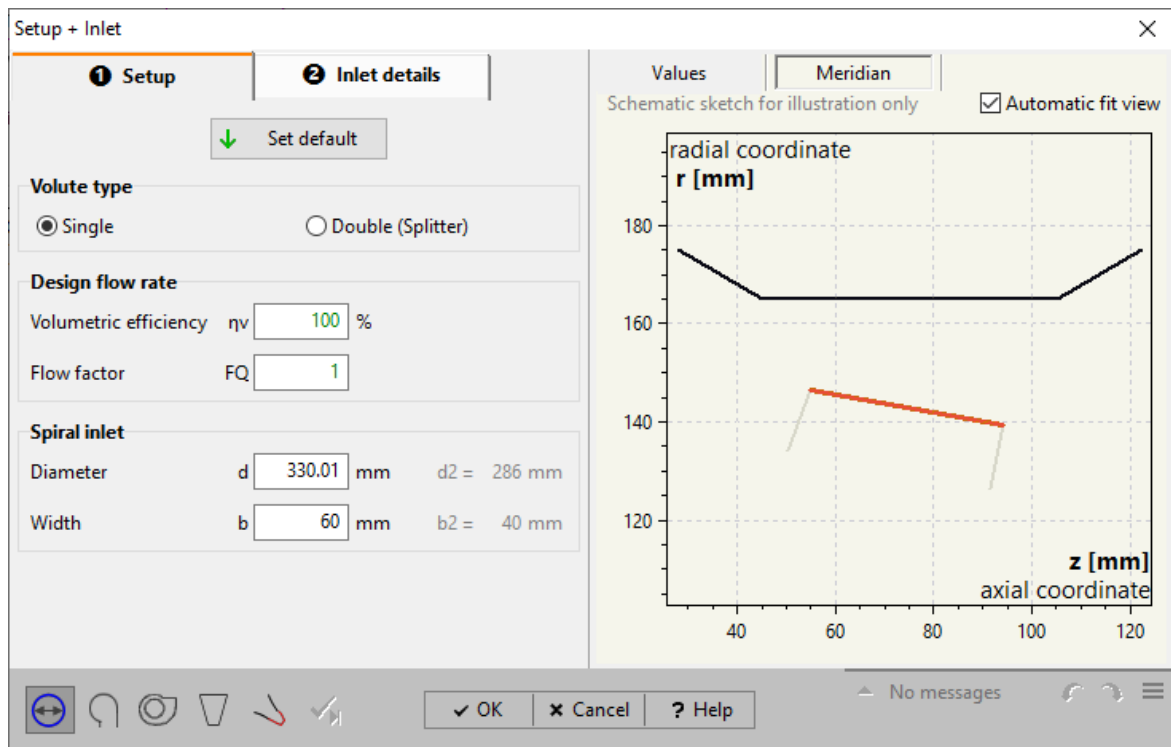
Auto fit view results in automatic scaling of the diagram if geometrical values are changing.

9.1.1 Setup

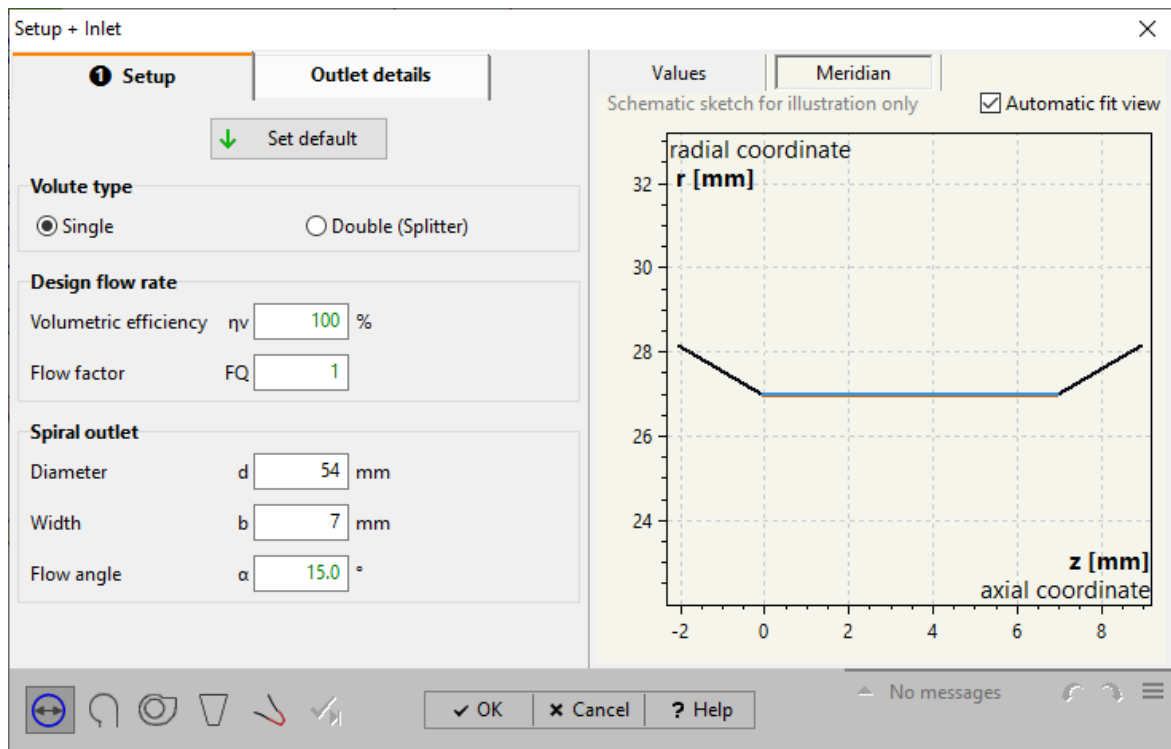
On page **Setup** you can define some general properties used for the spiral design.

Depending on the project type different input parameters are required (see below).

for pumps, ventilators, compressors



for turbines



Volute type

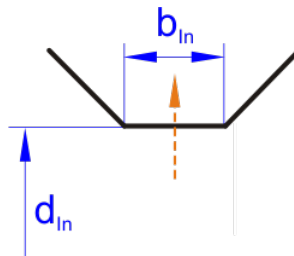
- Single volute (default)
This simple type is commonly used and has a single cut-water.
- Double volute
A second cut-water (splitter) is designed in order to reduce the radial forces.

Design flow rate

- Volumetric efficiency η_v (default: 1.0)
to consider any internal volumetric losses (recirculation)
- Flow factor F_Q (default: 1.0)
for over dimensioning, particularly for better efficiency at overload operation

Spiral inlet (outlet for turbines)

- Inlet diameter d_{in} (d_4)
- Inlet width b_{in} (b_4)
- Abs. flow angle α_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](#)⁵¹⁹, instead of specifying d_{in} and b_{in} values.

[for pumps, ventilators, compressors]

d_{in} and b_{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 nq (see [Approximation function](#)¹⁹⁸).

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_{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 radial 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_{in} and b_{in} are coupled to the corresponding [interface values](#)⁵¹⁹.

[for turbines]

d_{out} and b_{out} has to be set by the user.

Information

Various calculated values are shown, for information purposes, on the right side (**Values**):

Calculated internal flow rate Q_i	
Inlet/Outlet diameter ratio	d_{in}/d_2

Inlet/Outlet width ratio	b_{in}/b_2
Inlet/Outlet meridional velocity	c_m
Inlet/Outlet circumferential velocity	c_u
Inlet/Outlet velocity	c
Inlet/Outlet flow angle	

[for compressors, turbines]

Static pressure	p
Temperature	T
Density	
Mach-Number	Ma

Possible warnings

Problem	Possible solutions
Swirl-free inflow is implausible for volute designs.	
The volute is the first component of the project and the inflow swirl is defined by the Global setup ^[86] . With swirl-free inflow a volute calculation will be impossible.	Adapt pre-swirl in the Global setup ^[86]
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.	e.g. reduce the mass flow or increase the cross section

Problem	Possible solutions
Then an automatic velocity based contour design ⁵³¹ will not be possible since the necessary values of c_{u4} and Q_i are not available.	


9.1.2 Inlet details

On page **Inlet details** the details of the inlet interface can be specified.

Details: see [Interface Definition](#)

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](#)⁵¹⁴.

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](#)⁸⁶⁾) and flow properties on the outlet of upstream component.

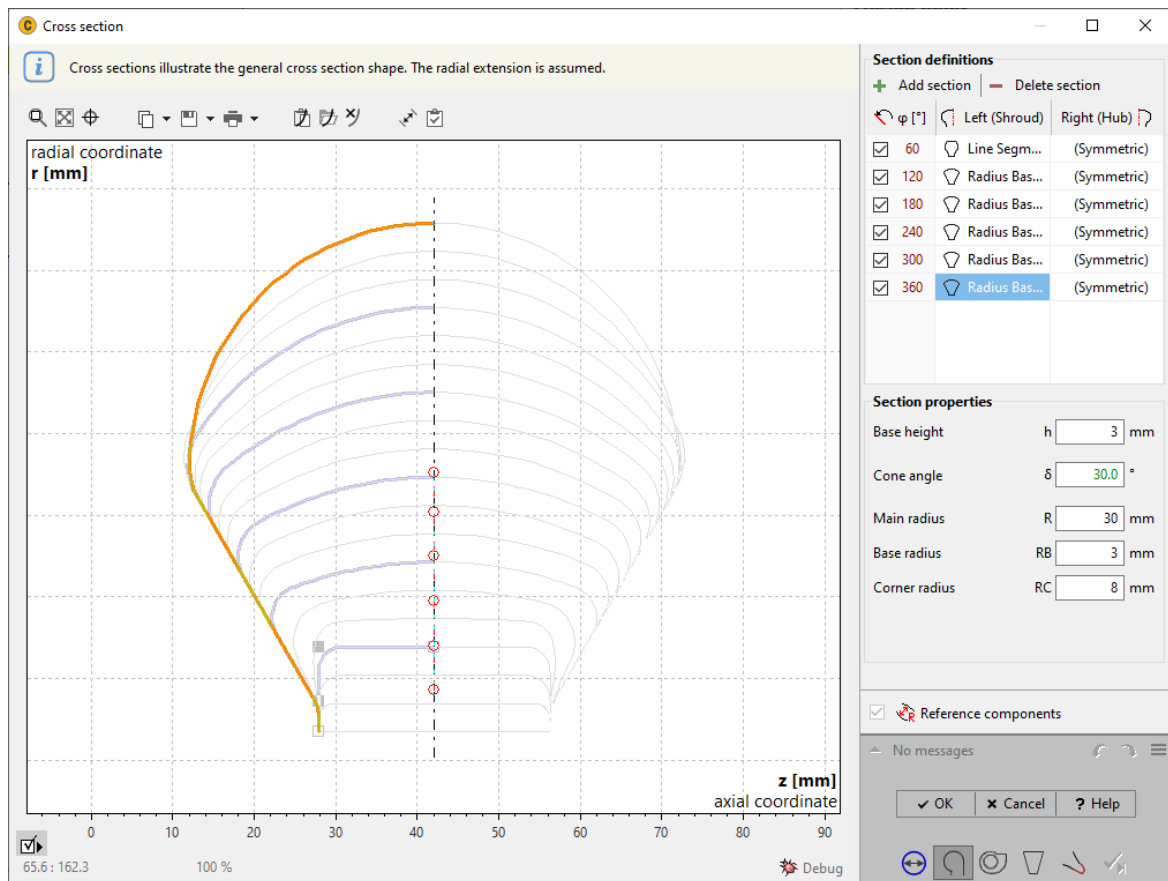
9.2 Cross Section

? VOLUTE | 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.



Sections

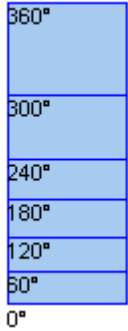
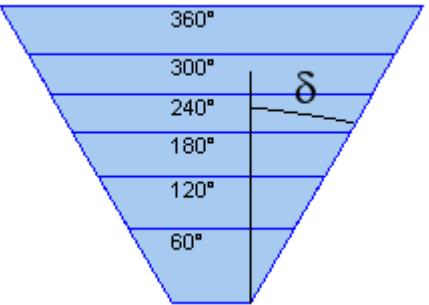
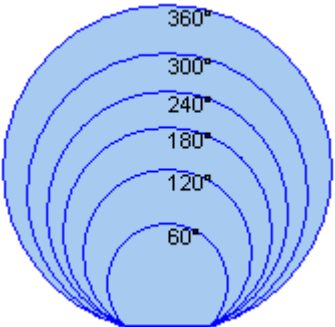
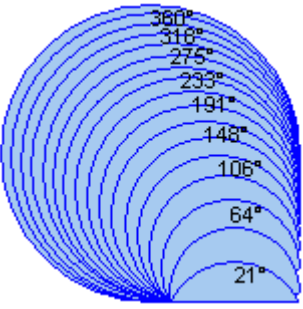
The table contains the cross section definitions (at least 1 cross section). Each cross section is defined by:

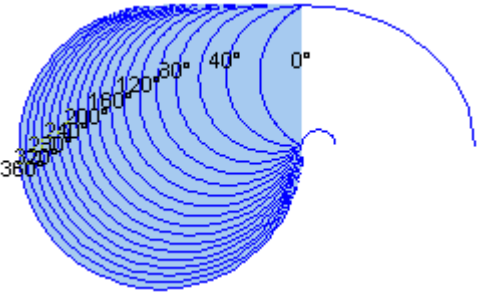
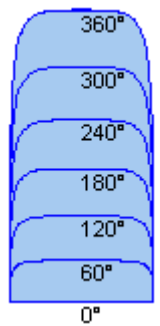
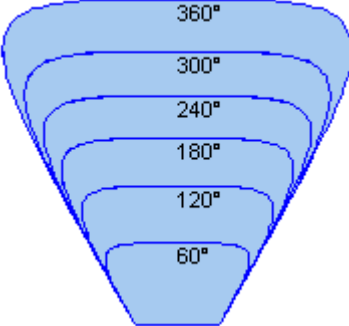
- the circumferential position: angle
- (de)activation by selecting the checkbox on the left side (at least 1 cross section has to be active)
- cross section type on the left side
- optional cross section type on the right side or symmetric

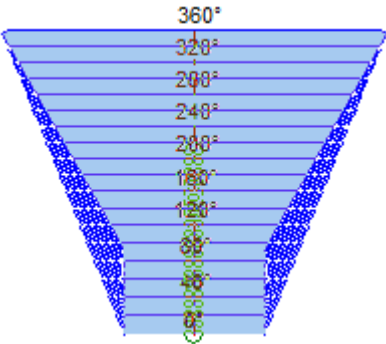
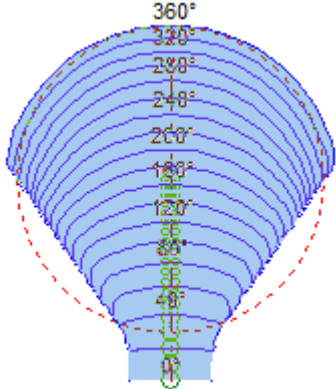
The section definition is running in the range $0^\circ < \leq 360^\circ$. The section at $=0^\circ$ is flat always - therefore a section definition at this position makes no sense.

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:

	<p>Rectangular</p> <p>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</p>
	<p>Trapezoid</p> <p>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</p>
	<p>Round - symmetric</p> <p>simple elliptic geometry with a beneficial stress distribution; does not develop on rotation surfaces</p> <ul style="list-style-type: none"> • Ratio of ellipse radii: axis ratio is preserved for developing sections. (ratio of one gives a circular section)
	<p>Round - asymmetric, external</p> <p>more favorable secondary flow structure than with a symmetrical cross-section; often with mixed-flow impellers</p> <ul style="list-style-type: none"> • Ratio of ellipse radii: axis ratio is preserved for developing sections. (ratio of one gives a circular section)

	<ul style="list-style-type: none"> • 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
	<p>Round - asymmetric, internal</p> <p>limitation of radial extension; ratio of ellipse radii possible;</p> <p>additional bend necessary</p> <p>see Internal cross sections ⁵³⁰</p>
	<p>Bezier - Rectangle type</p> <p>analogous with Rectangle; with chamfers (cast radii)</p> <p>see Bezier cross section ⁵²⁵</p>
	<p>Bezier - Trapezoid type</p> <p>analogous with Trapezoid; with chamfers (cast radii)</p> <p>see Bezier cross section ⁵²⁵</p>

	<p>Line segments</p> <p>see Line Segments cross section ⁵²⁶</p>
	<p>Radius based</p> <p>see Radius based cross section ⁵²⁹</p>

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

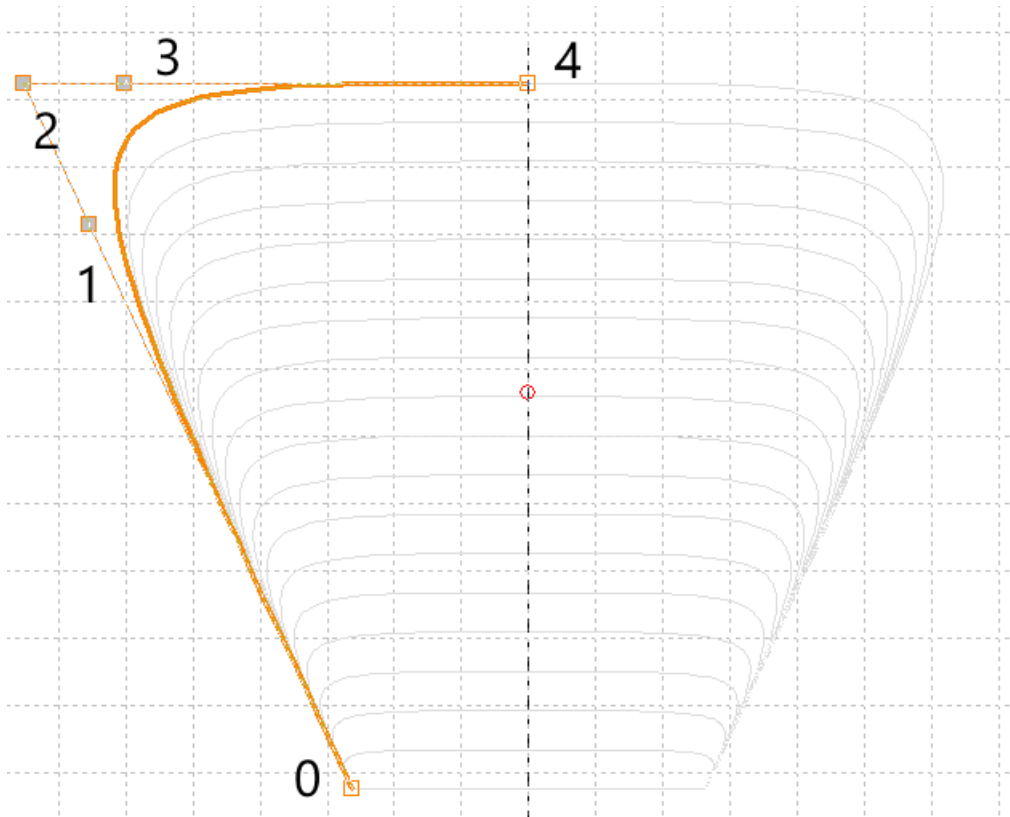
For double volutes the cone angle (opening) of all cross sections has to be constant. Therefore, round types and Line segments are not available.

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](#) ⁵¹³). 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.



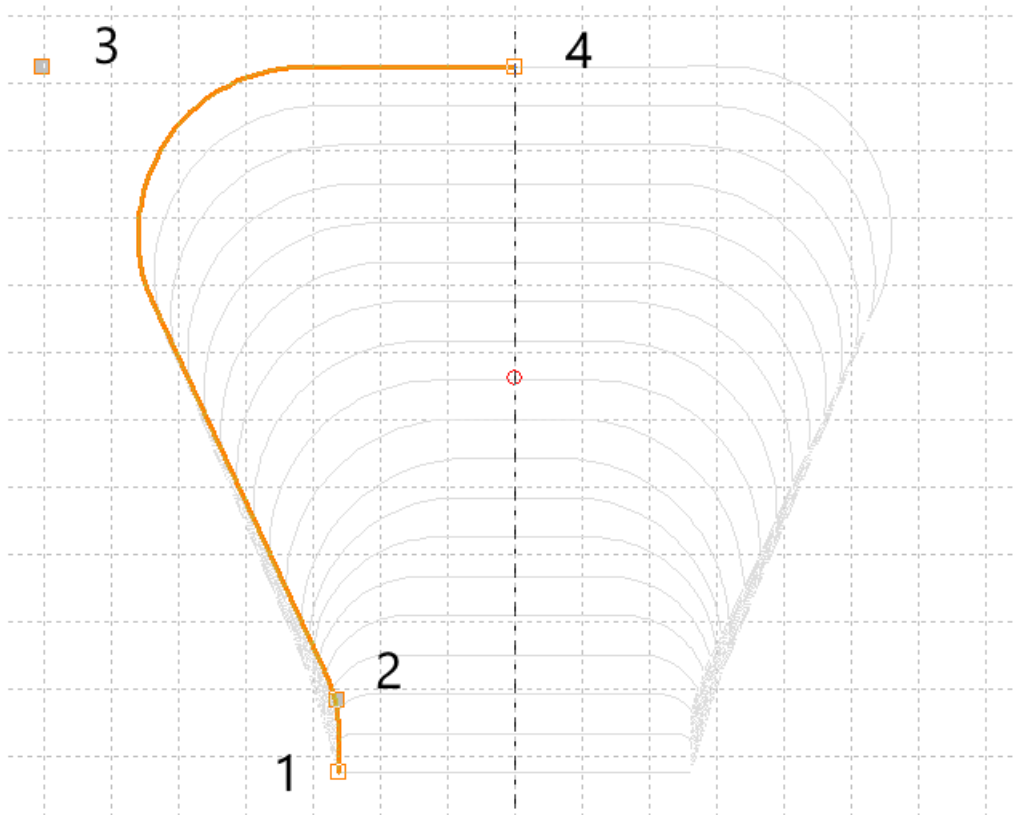
One half of the shape of the cross-section is described using a 4th degree Bezier polynomial. Points 0 and 4 are the end points and cannot be changed. Point 1 can be moved along a straight line which corresponds to the cone angle of the cross-section (0° for a rectangle type, δ for a trapezoid type). 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.

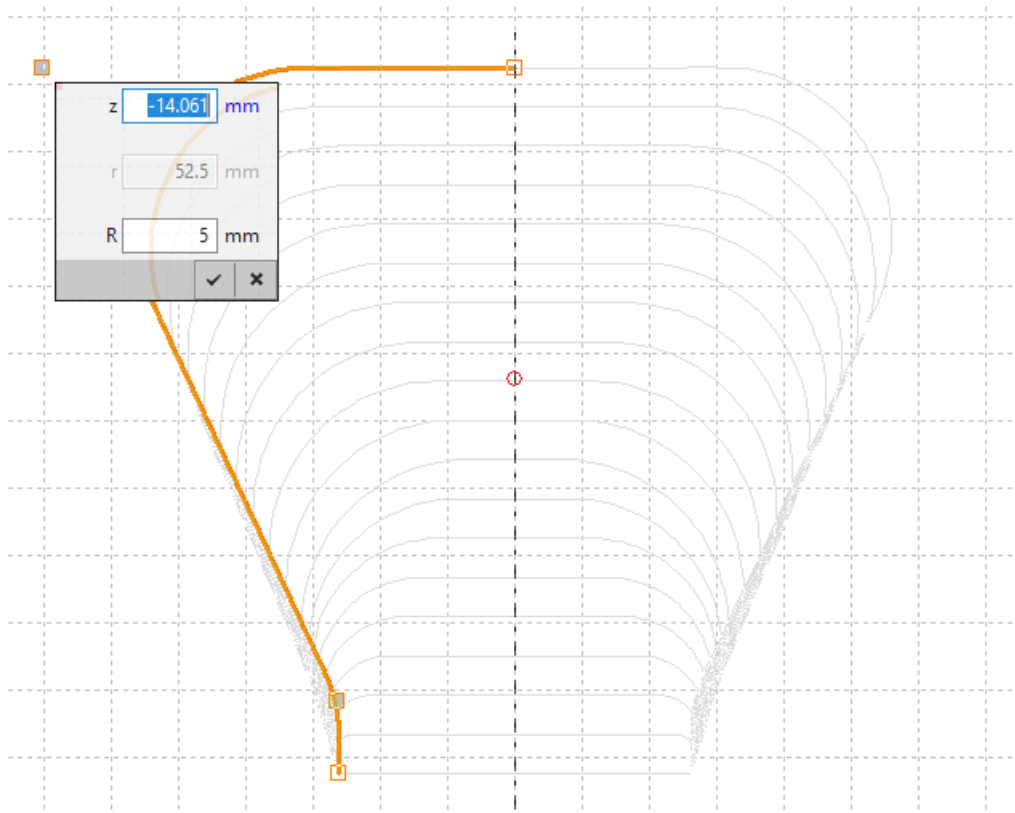
9.2.2 Line Segments cross section

The shape of a **Line segments** cross-section is described by a series of line segments.



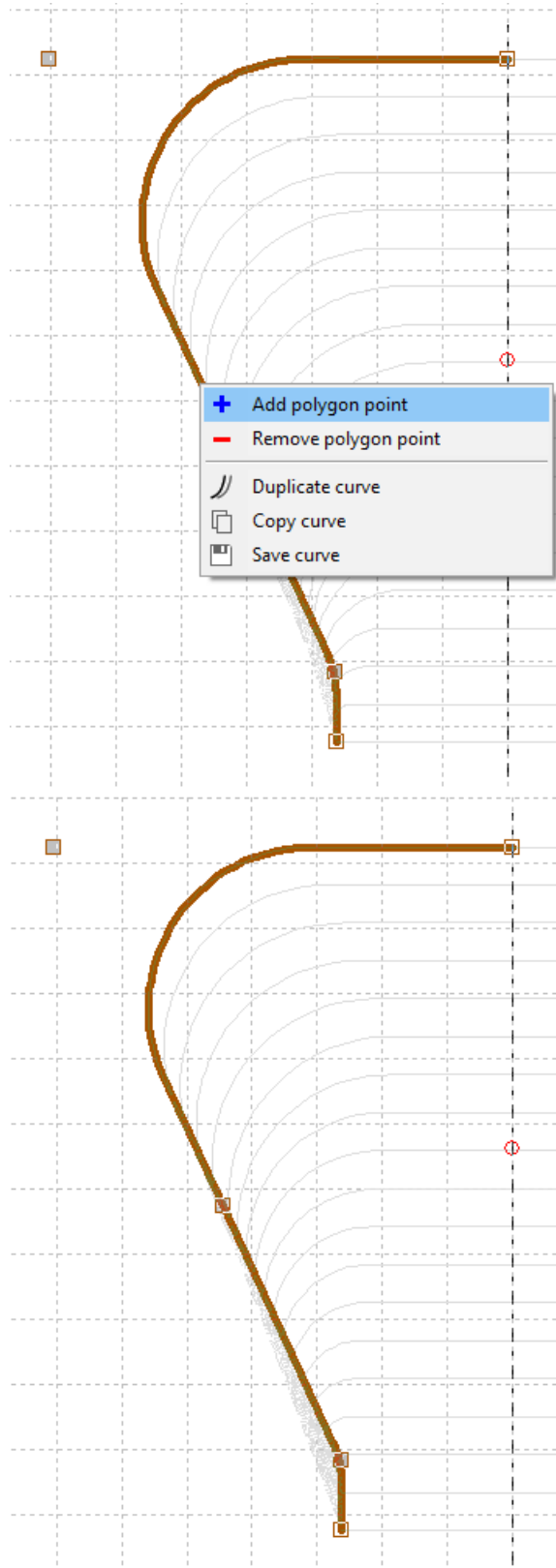
One half of the shape of the cross-section is initially based on line segments arranged in a trapezoid shape. Points 1 and 4 are the fix start- and endpoint.

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:

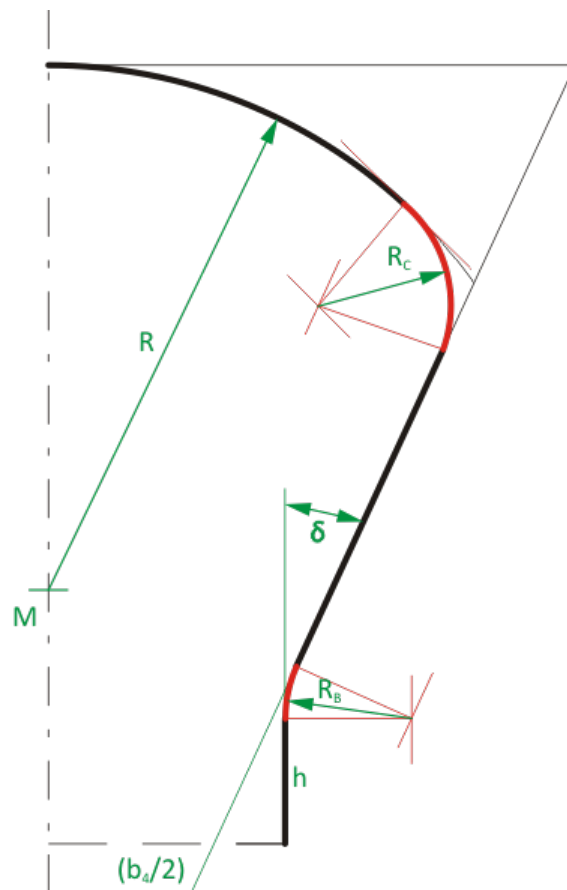


When moving points the following constraints can be enforced by pressing a key on keyboard:

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.

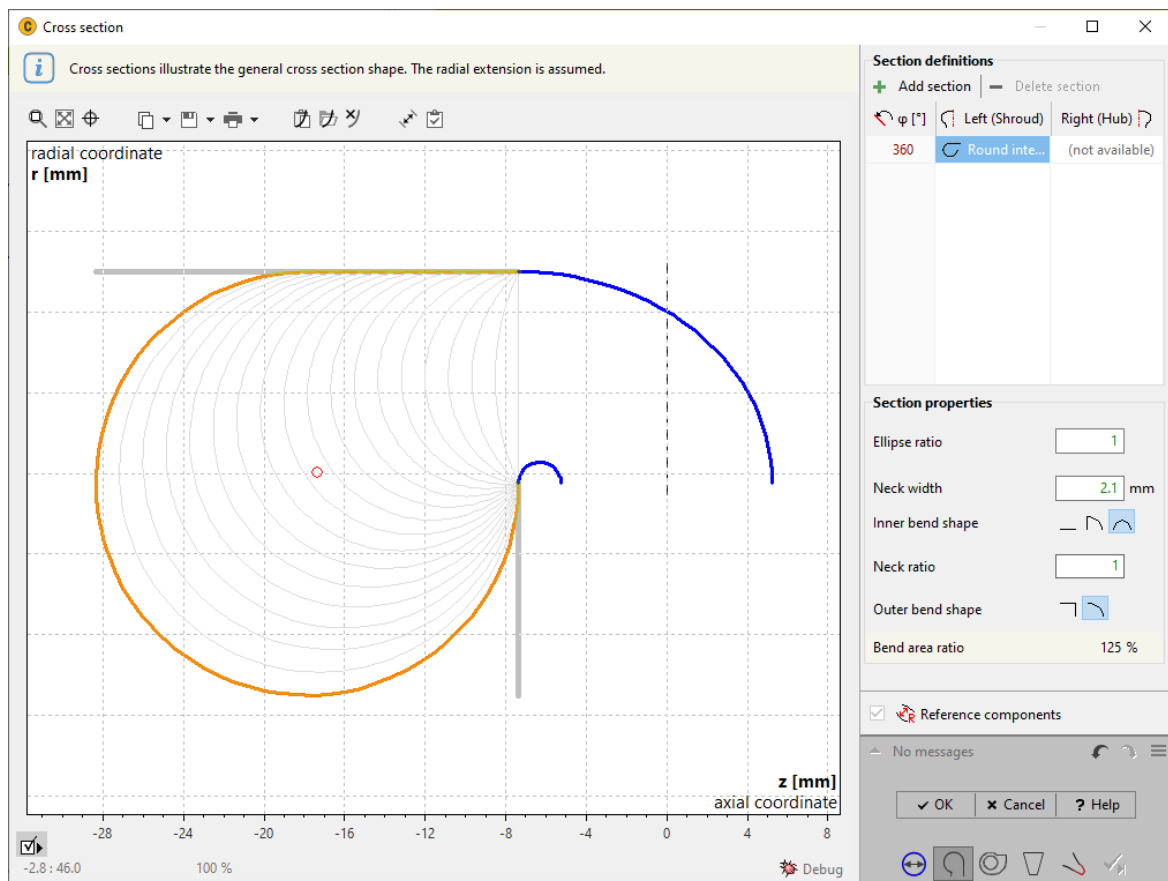


The geometry is described by the following parameters:

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		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:

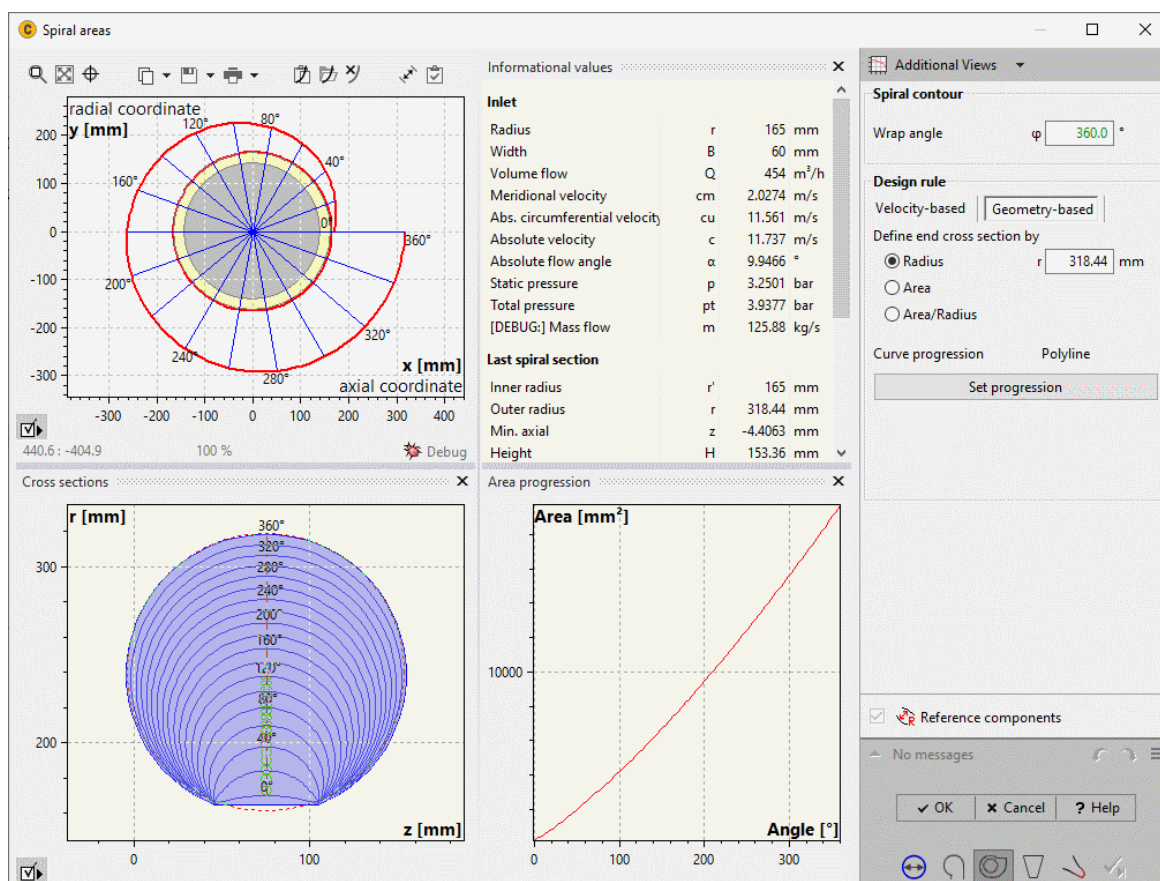
Neck width	side distance from volute inlet to actual volute cross sections
Inner bend shape	shape of the inner bend wall
Ratio	semiaxis ratio for quarter bend
Outer bend shape	shape of the outer bend wall
Bend area ratio	ratio of outlet to inlet section of the bend

9.3 Spiral development areas

? VOLUTE | Spiral development areas



The spiral development areas can be designed and calculated in this dialog box.



Spiral contour

The **wrap angle** can be defined - default value is 360°.

Design rule

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 outer radius, cross section area or ratio of area to center of mass.

→ [Details Design Rule](#) ⁵³⁴

Cut-water compensation

In panel **Cut-water compensation** you can specify parameters for the cut-water design.

→ [Details Cut-water compensation](#) ⁵³⁷

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 maximal deviation from the initial contour, which is defined by the design rule. 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.

Possible warnings

Problem	Possible solutions
Spiral contour could not be calculated exactly. Check geometry and "Volute/ Inlet definition".	
Spiral sections cannot be calculated due to unusual inflow direction or volute cross section definition.	Too narrow cross section shape can result in unreasonable high height-width-ratio. Try to select another cross section shape.

Problem	Possible solutions
Volute end cross section not sensible. Check geometry and "Volute/ Inlet definition".	
The properties of the end cross section are not reasonable, e.g. the ratio H5/B5 is too low or too high.	Check the properties of the end cross section. See also the hints to the error "It's not possible to calculate spiral contour exactly.".
Flow angle at volute inlet is too large. Check flow properties of design rule.	
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 $c_m = Q_i / (d \cdot b \cdot \pi),$ circumferential velocity is set directly. Increase c_u or reduce Q_i to decrease flow angle. $\tan(\) = 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.	The flow angle on volute inlet should be small (<~45°, 90° is completely invalid). It can be checked in "Volute/ Inlet definition", page "Volute" right at "Values": Flow angle . The inlet flow angle is defined by the previous component. If no previous component exists, the inflow angle is defined by "Global setup/ Inflow".
Angle of last cross section definition is higher than spiral wrap angle.	
One or more cross sections are defined at positions > spiral wrap angle	Adapt circumferential position of the cross section definition ("Volute/ Cross section") or spiral wrap angle ("Volute/ Spiral areas").
Automated contour update active. Cross section sizes are not fixed and adapt to external flow properties.	

Problem	Possible solutions
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.
Automated contour update NOT active. Cross section sizes are fixed but may be inappropriate for external flow properties.	
Spiral cross section extents 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.

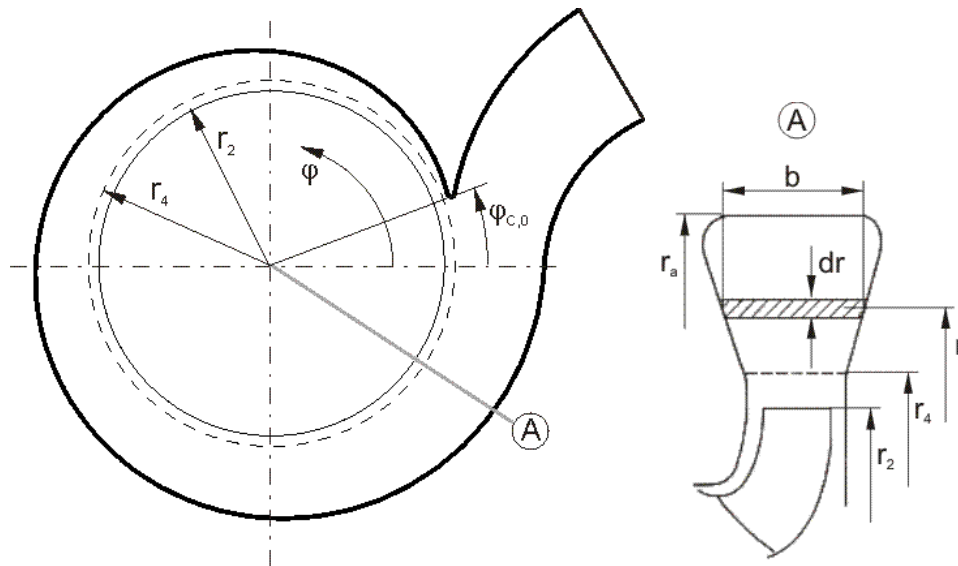
9.3.1 Design rule

The flow rate through a cross-section, A, of the circumferential angle, φ , is generally calculated as:

$$Q_{\varphi} = \int c_u dA = \int_{r_4}^{r_a(\varphi)} c_u b(r) dr$$

Using $Q_{\varphi} = \varphi / (2\pi) Q_i + Q_0$ results in an equation to calculate the circumferential angle, φ , dependent on the outer radius r_a :

$$\varphi = \frac{2\pi}{Q_i} \int_{r_4}^{r_a(\varphi)} c_u b(r) dr$$



$b(r)$ 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

Design rule

Velocity-based | Geometry-based

☒ Use flow properties from inlet

Ref. velocity c_u 11.561 m/s

Ref. volume flow Q_i 454 m³/h

Blind volume flow Q_0 0 m³/h

Design theory Pfleiderer

$c_u \cdot r^x = \text{const.}$

Swirl exponent x 1.7

c_u $c_u \cdot r^2$

For all velocity-based rules the area for each cross section is calculated using a linearly increasing volume flow Q starting at Q_0 for $\varphi=0$ (blind volume flow) and an assumed velocity distribution c over r . While Q depends on a total reference volume flow Q_i , the velocity distribution is defined by a

reference c_u at r_4 and one of the following velocity rules. Note that both reference values can be chosen manually by deactivating **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.

$$\varphi = \frac{2\pi c_{u,4} r_4^x}{Q_i} \int_{r_4}^{r_a(\varphi)} \frac{b(r)}{r^x} dr \Rightarrow Q_\varphi = \int_{r_4}^{r_a(\varphi)} c_{u,4} \left(\frac{r_4}{r} \right)^x b(r) dr \Rightarrow c_u(r) = c_{u,4} \left(\frac{r_4}{r} \right)^x$$

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](#)^[198]).

$$\varphi = \frac{2\pi k_s \sqrt{2gH}}{Q_i} \int_{r_4}^{r_a(\varphi)} b(r) dr \Rightarrow Q_\varphi = k_s \sqrt{2gH} \int_{r_4}^{r_a(\varphi)} b(r) dr \Rightarrow c_u = k_s \sqrt{2gH} = \text{const.}$$

Note, for manually defined reference velocities $c_{u,4}$, k_s has no influence because c is constant over r .

Geometry-based rules

Design rule

Velocity-based | **Geometry-based**

Define end cross section by

☒ Radius r mm

☐ Area

☐ Area/Radius

Curve progression Linear

[Set progression](#)

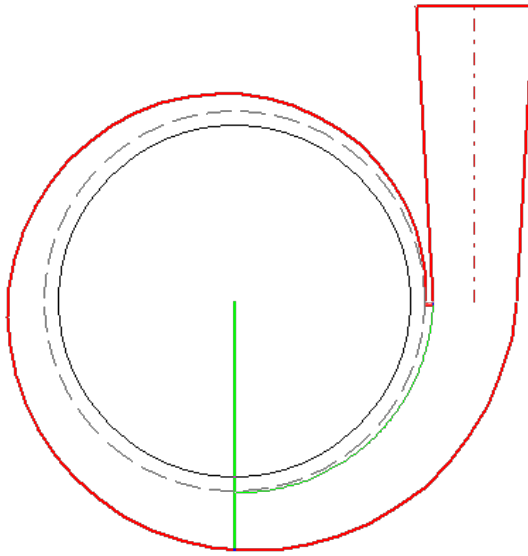
Contrary to velocity-based rules the geometry progression is defined directly. The end cross section is defined by radius or cross section area, the distribution by Radius-, Area- or Area/Radius-progression ([Set Progression](#)^[70]).

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:

Inner radius r	Informative, see Inlet ^[513] r is the inlet radius of the volute and/or outlet radius of radial diffusers
Thickness e	Thickness of the cut-water at the start of the volute (for compensation)
Compensation φ_C	Angle, above which cut-water correction begins (standard: 270°)



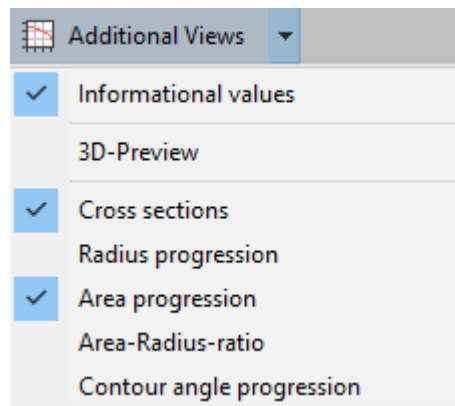
The cut-water does disturb the flow, since the cross-section of the flow is narrowed suddenly by the thickness of the cut-water.

To weaken this negative influence, the cut-water can be corrected. This is achieved by assuming that from the angle φ_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 Additional views

The following information can be displayed in the spiral dialog using the "Additional views" button:

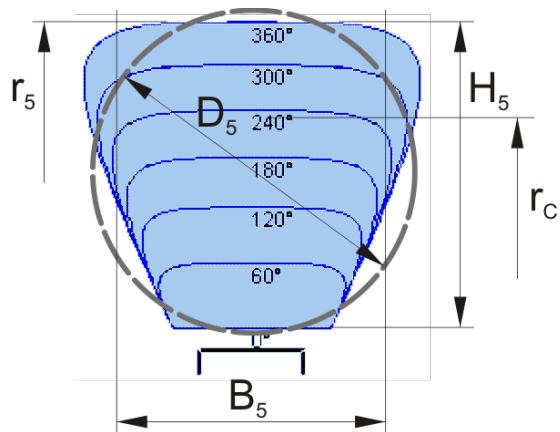


3D-Preview

[3D model](#)  of the currently designed spiral development areas.

Informational values

Some informative values relating to the **end cross-section** are displayed:



Inner radius	r'
Outer radius	r
Min. axial	z
Height	H
Width	B
Side ratio	H/B
Equivalent diameter	D
Area	A
Area-radius-ratio	A/r_c
Volume flow	Q
Average velocity	c
Static pressure	p
Total pressure	p_t
Density	
Temperature	T
Mach-number	Ma
Total temperature	T_t

Cross sections

Volute cross sections (z - r)

Radius progression

Radius distribution (r - r)

Area progression

Area distribution (A - A)

Area-radius-ratio

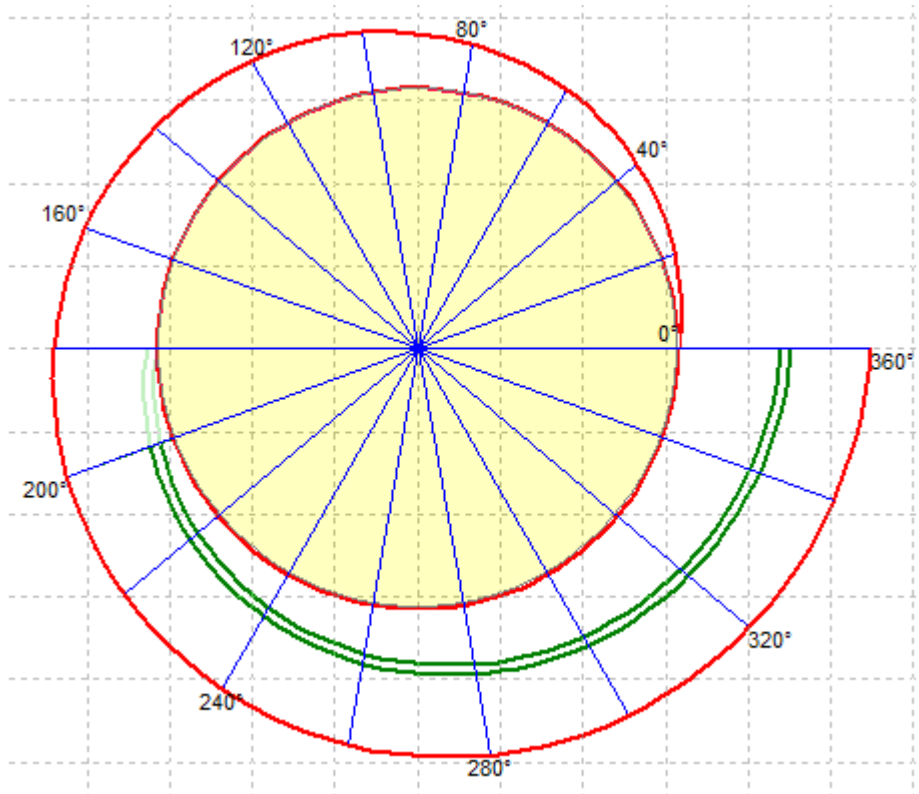
Area/Gravity center radius (r_c) distribution (A/r_c - A/r_c)

Contour angle progression

Angle between the outer spiral contour and the circumferential direction (α - α). Note, that due to the differential characteristic of the contour angle, the continuity of this distribution is decreased by one.

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](#)⁵¹⁴.



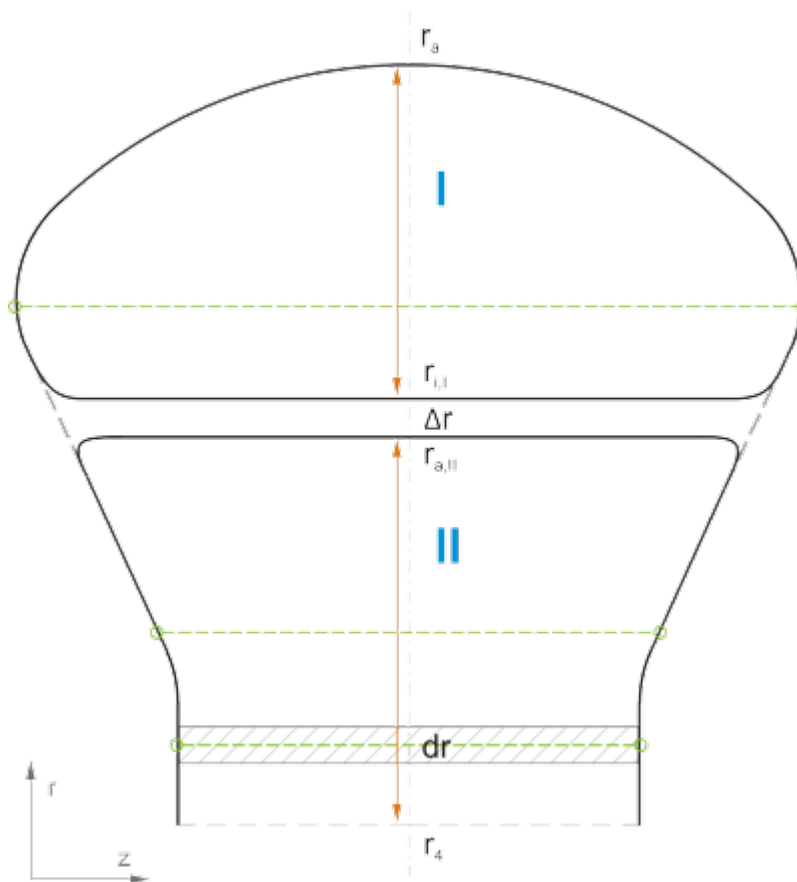
General procedure for Double Volute design

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](#)⁵³⁷. 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 φ is given by the outer radius r_a at $\varphi - \varphi_{Spl}$.

The Outer area (I) is calculated based on the [Design rule](#)⁵³⁴ for

- * a constant flow rate defined by the splitter start angle (normally 50% of overall flow rate)
- * starting from the splitter outside radius $r_{i,I} = r_{a,II} + r$.



Splitter of Double Volute

For double volutes you can define additional properties of the spiral and splitter.

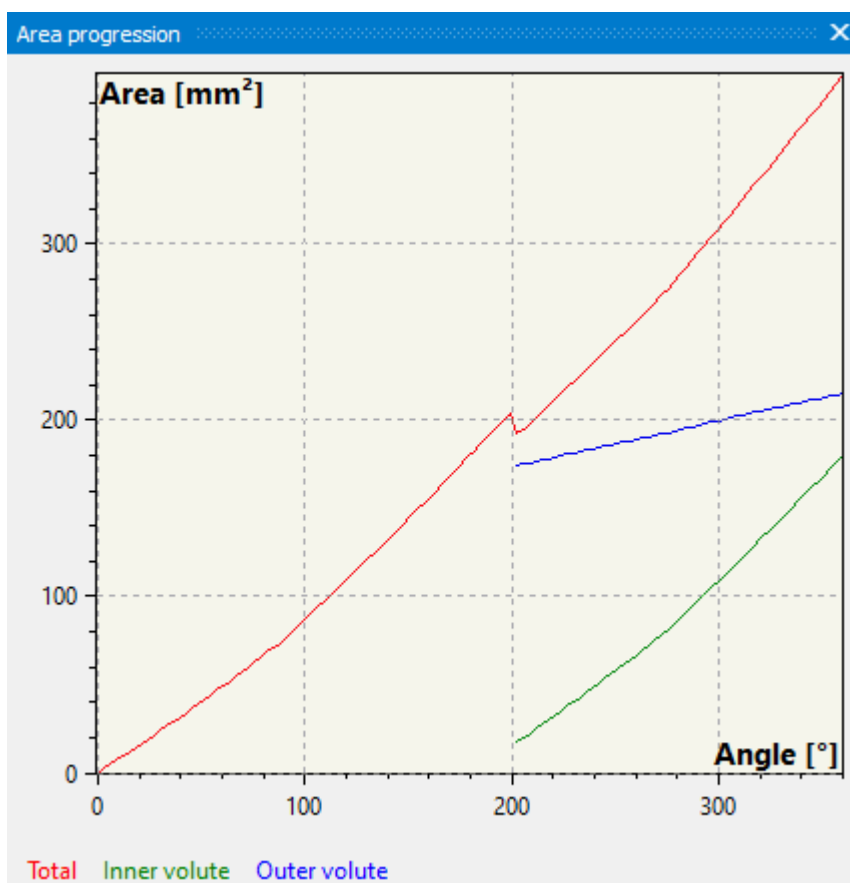
- The **start angle** φ_{Spl} is the angular position where the splitter starts. It also determines the splitter contour.
- The **angular offset** φ_{Spl} can be used to achieve a radial offset without changing the contour.
- The **thickness** e_{Spl} defines the distance between the inner and outer splitter contour.
- The **compensation** $\varphi_{Spl,C}$ is used analogous to the cut-water compensation.
- The **fillet radius** defines the radial corner radius between spiral and splitter surface.

Splitter of Double Volute

Start angle	φ_{Spl}	<input type="text" value="180.0"/>	°
Angular offset	$\Delta\varphi_{Spl}$	<input type="text" value="20.0"/>	°
Thickness	e_{Spl}	<input type="text" value="1.2"/>	mm
Compensation	$\varphi_{Spl,C}$	<input type="text" value="90.0"/>	°
Fillet radius	r_{Spl}	<input type="text" value="1"/>	mm

Additional views

The progression diagrams contain curves for each part of the volute, like the area progression below.



Beside the default [informational values](#) ⁵³⁸ separate values for inner and outer part of the volute are reported.

Furthermore 2 additional ratios are displayed:

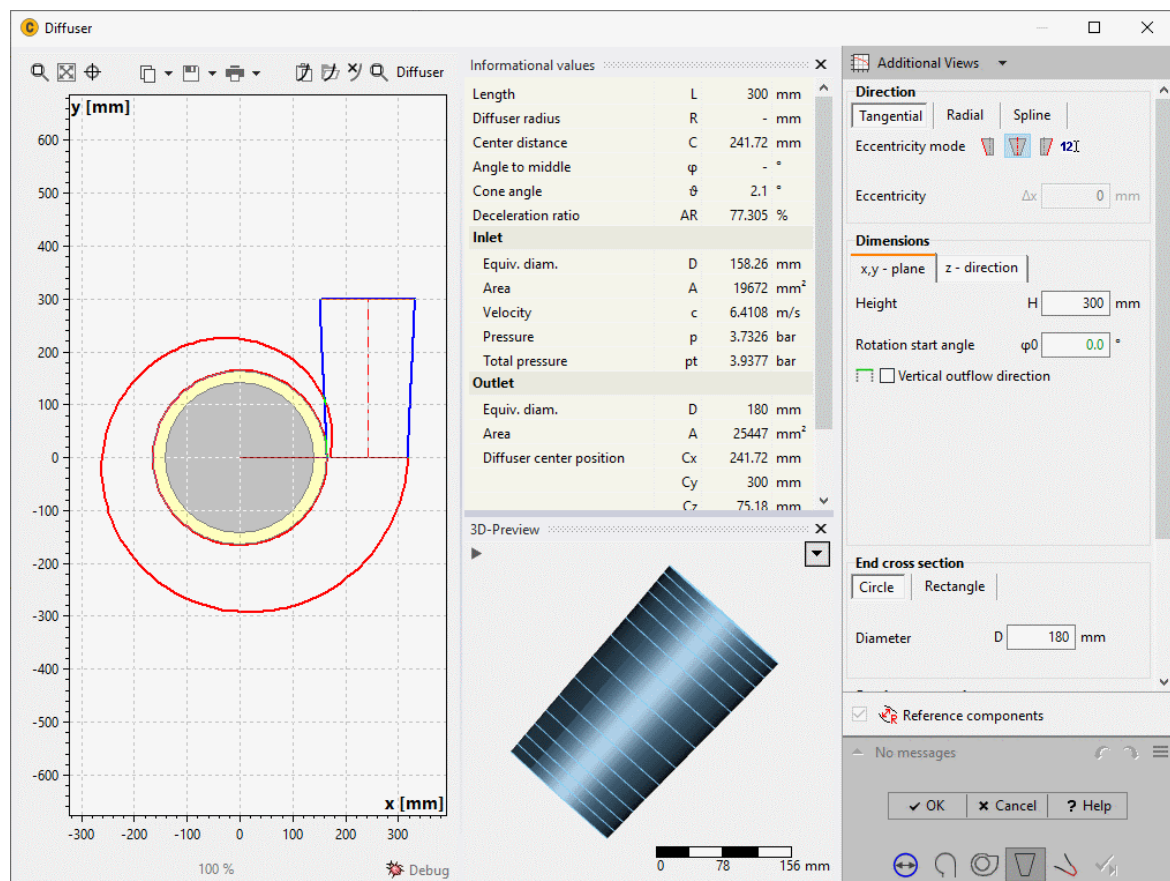
- Expansion of outer volute (using end point of blue curve / start point of blue curve)
- Ratio of outer to inner throat (using end point of blue curve / end point of green curve)

9.4 Diffuser

? VOLUTE | Diffuser

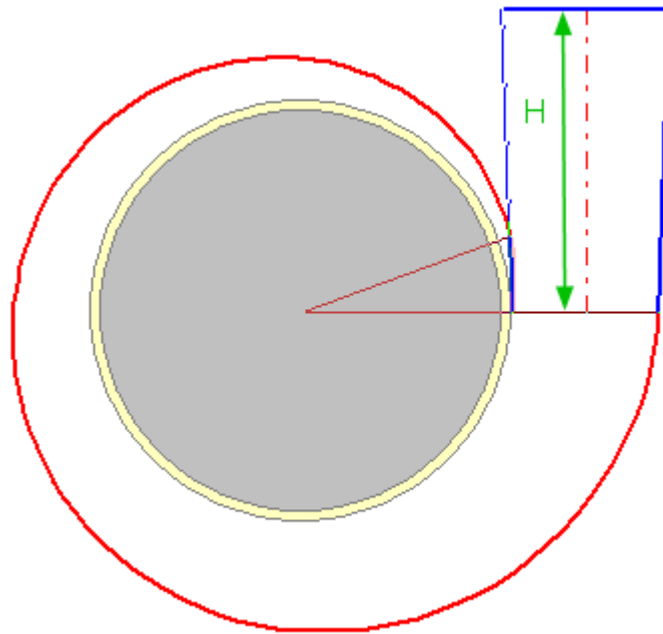


The geometry of the outlet diffuser (inlet diffuser for turbines) can be designed and calculated in this dialog box.

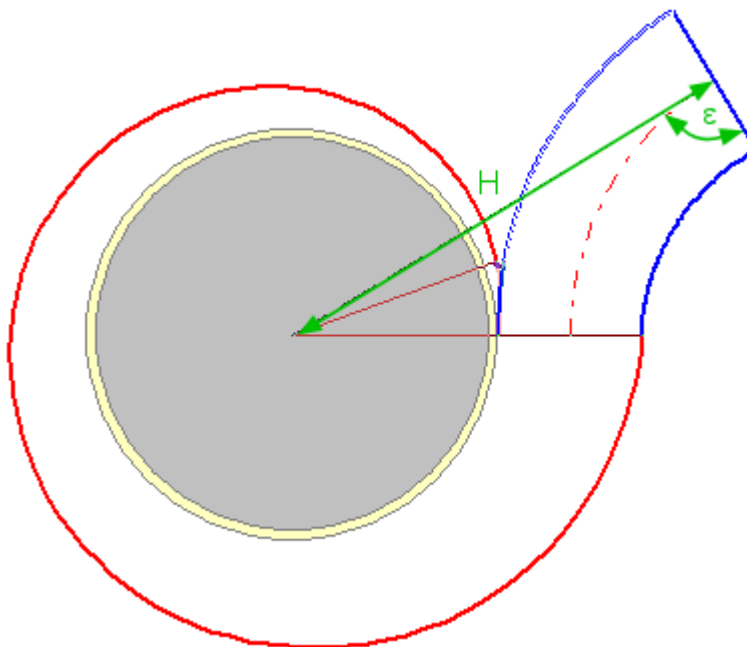


Direction

In general, 3 basic shapes are available:

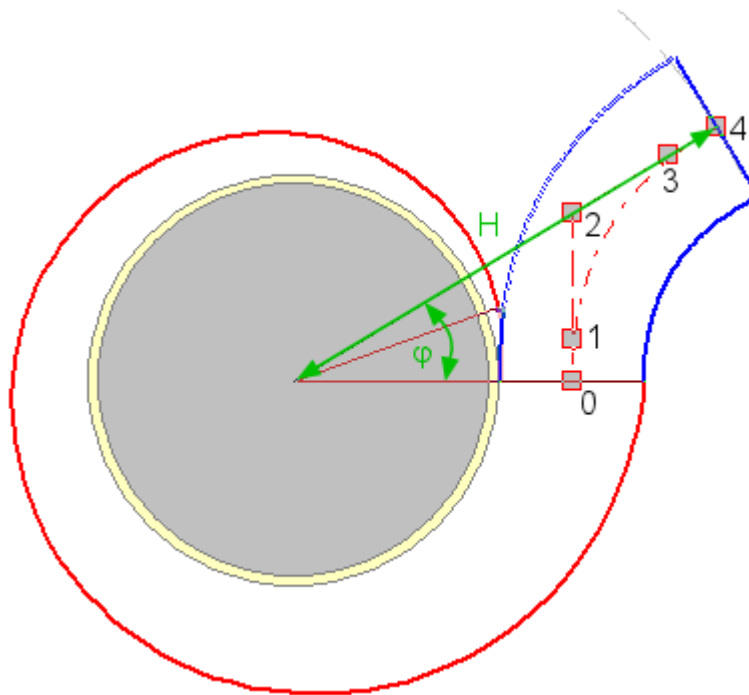


Tangential diffuser



Radial diffuser





Spline-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.
-  The eccentricity can be specified manually.

Radial diffuser

In the case of a radial diffuser, the angle ε between the outlet branch and the line connecting impeller-center and outlet branch center can be selected.

Spline diffuser

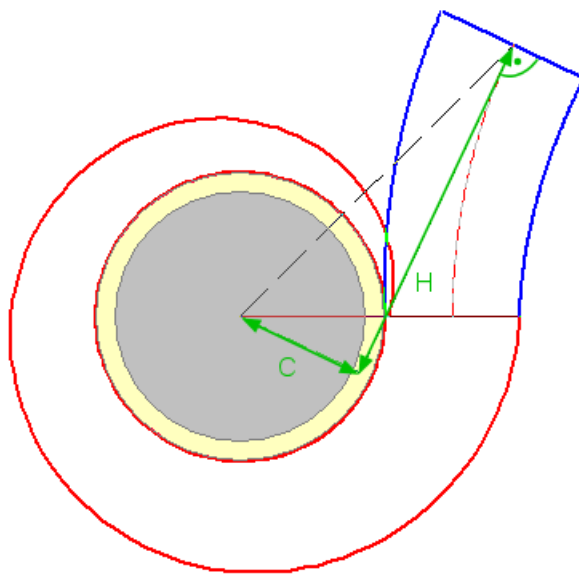
For the Spline-diffuser the angle φ between connecting line impeller-center \leftrightarrow outlet branch center and diffuser start section has to be defined. Points 0 and 4 are start and endpoint of the middle line

on the inlet and outlet cross section, point 2 is fixed by the intersection of appropriate perpendiculars 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.

Dimensions

The extension of the diffuser can be defined in panel **Dimensions**. Parameters in the **x,y-plane** can be specified, as well as a rake of the diffuser in **z-direction**.



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.

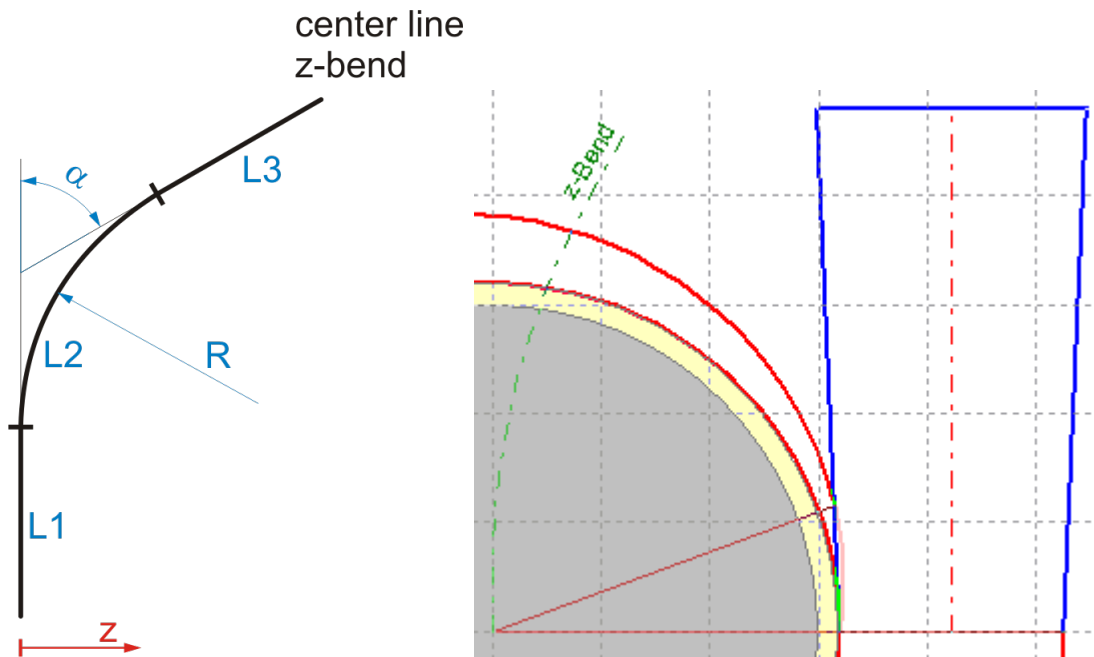
Additionally the starting position of the diffuser is defined by the angle φ_0 , whereas 0° is horizontal right. The whole volute can be rotated by this value. By using the button **Vertical outflow direction** the volute can be rotated for vertical direction of the pressure joint.

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 **L₁**, **L₂** and **L₃** are specified as percentage.

The curvature is defined by the radius **R**, the direction by the angle .

The z-bend is illustrated in the diagram by a green center line.



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 **H** and width **B** 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.

Linear blending	The morph between two different cross sections is linear which results in an quadratic area progression. (unscaled)
Linear area	The size of the morphed cross sections is scaled to achieve a linear area progression.
Quadratic area	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

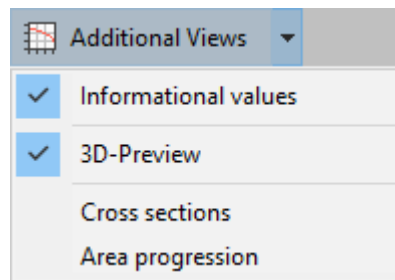
	diffuser outlet is linear again.
Custom area	The size of the morphed cross sections is scaled with respect to a Beziér curve.

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:



3D-Preview

[3D model](#)  of the currently designed diffuser geometry as well as spiral surfaces.

Informational values

Deceleration ratio AR	$A_R = (D_{In} / D_{Out})^2$
Length L	Length of the diffuser
Angle to middle φ	Angle between connecting line impeller-center ↔ outlet branch center and diffuser start section
Center distance C	Distance from the h-line to the center point
Cone angle ϑ	Cone angle from D_{In} to D_{Out} over the length L

Max. theo. cone angle ϑ_{\max}	$\vartheta_{\max} = 16.5^\circ (D_{\text{In}}/2L)^{1/2}$, see Gülich ⁵⁶⁶ , not for turbines
Diffuser radius R	Radius of middle line (for radial diffuser only)
Diffuser outlet center C	Spatial position of diffuser outlet

Inlet & Outlet information

Equivalent diameter D	Diameter of the equivalent circle at the diffuser inlet / outlet
Area A	Area at diffuser inlet / outlet
Velocity c	Velocity at the diffuser inlet / outlet
Pressure p	Pressure at the diffuser inlet / outlet
Total pressure p_t	Total pressure at the diffuser inlet / outlet

For turbines and compressor additional information is visible:

Temperature T	Temperature at the diffuser inlet / outlet
Total temperature T_t	Total temperature at the diffuser inlet / outlet
Density ρ	Density at the diffuser inlet / outlet
Mach-Number Ma	Mach-Number at the diffuser inlet / outlet

Cross sections

Diffuser cross sections (z-r)

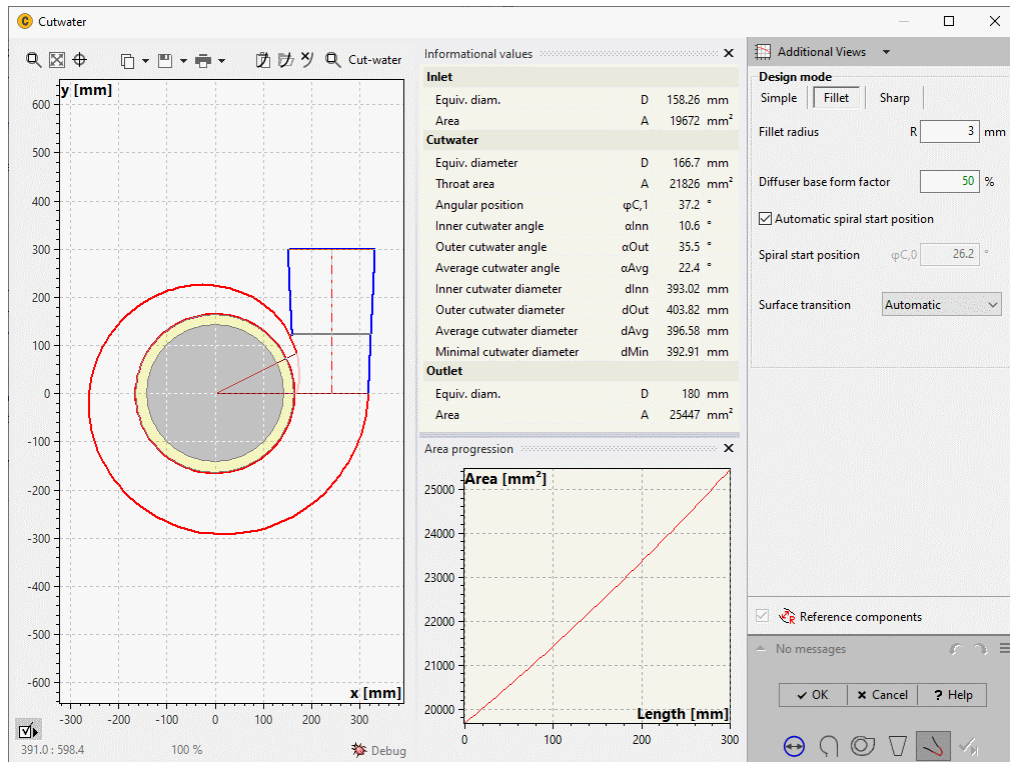
Area progression

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](#)⁵⁵³, [Fillet](#)⁵⁵⁶ or [Sharp](#)⁵⁵⁹.

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

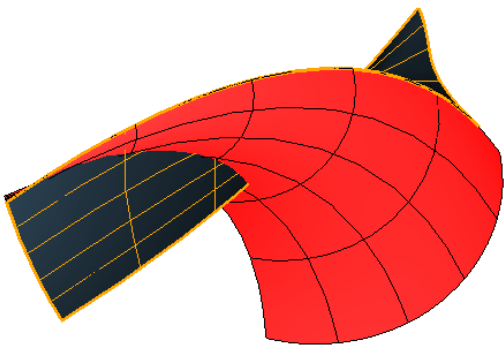
General

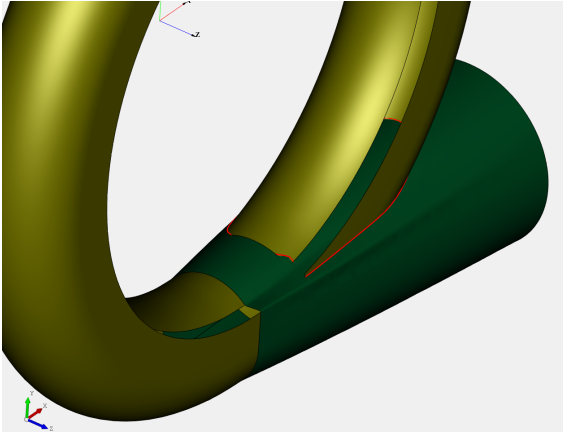
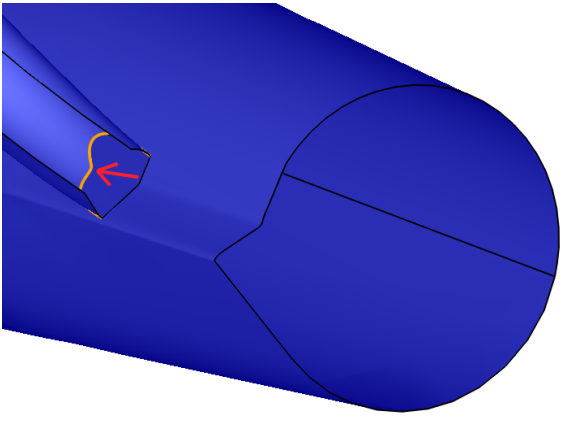
The [wrap angle](#)⁵³¹ must be at least 330°.

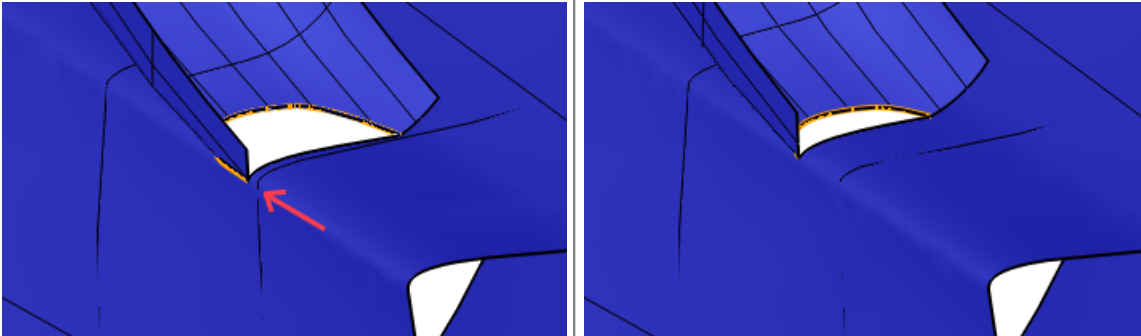
Simple	For cornered spiral cross sections the side position is fixed to the corner position and cannot be modified individually.
	Rounding of cut-water edges (Round edges) is possible only if side position is higher than the position of maximum curvature and if no radial offset is defined.
	Radial offset is available for strictly external volutes with 360° wrap angle only.
Fillet	Fillet cut-water is not available for cornered cross sections, either spiral or diffuser.
	Intersection of spiral and diffuser geometry is necessary to create a fillet cut-water.
	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.
Sharp	Sharp cut-water is not available for cornered cross sections, either spiral or diffuser.
	Intersection of spiral and diffuser geometry is necessary to create a sharp cut-water.

Cut-water design is not available for **internal volutes**.

Possible warnings

Problem	Possible solutions
Cutwater is self-intersecting.	
<p>Cut-water faces intersect each other.</p> 	<p>The problem might have various reasons. Therefore, modify spiral, diffuser or cutwater design.</p> <p>E.g. define a flat radius progression at the start of spiral development areas⁵³¹, or change angular position / radial offset of the cutwater.</p>

Problem	Possible solutions
Multiple intersection curves between spiral and diffuser.	
<p>Spiral and diffuser may have multiple intersection curves. (red)</p> 	<p>This problem can be solved by manipulating the spiral start position. A single intersection curve is required for generation of sharp and fillet cut-waters.</p>
3D-Error: Could not create bounded surface for Cut-water.Patch!	
<p>Parameter side position is disadvantageous.</p>	<p>The side position should not be too low when edges are rounded.</p>
3D-Error: Could not create fillet for Cut-water! Possibly, the fillet radius is too large.	
<p>[for asymmetric volutes]</p> <p>Fillet cannot be created because intersection curve of spiral and diffuser is wavy.</p> 	<p>Modify the Position of end shape⁵⁴⁷ in the Diffuser dialog to avoid wavy intersection curve.</p>

Problem	Possible solutions
<p>[for asymmetric volutes]</p> <p>Fillet cannot be created because intersection curve of spiral and diffuser is tangential to the sharp diffuser edge.</p> 	<p>Modify Spiral start position</p>

9.5.1 Simple

The simple cut-water is a rounding-off between spiral and diffuser.

Design mode

Simple | Fillet | Sharp

Spiral start position $\varphi_{C,0}$ °

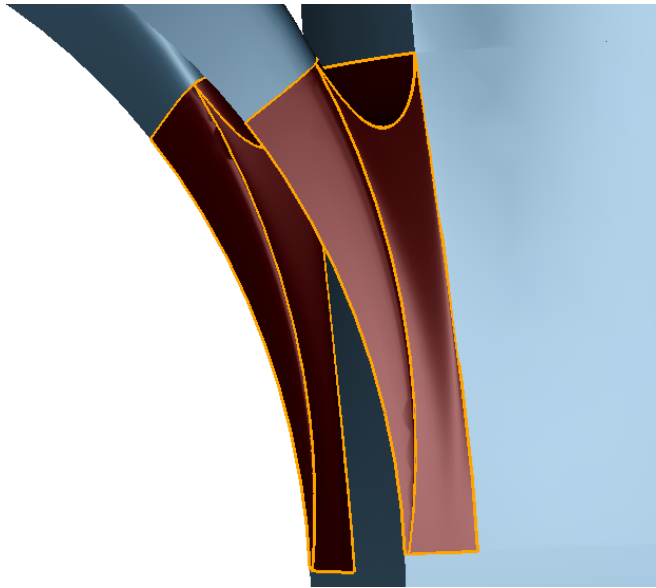
Radial offset Δr_C mm

Height position left % Height position right %

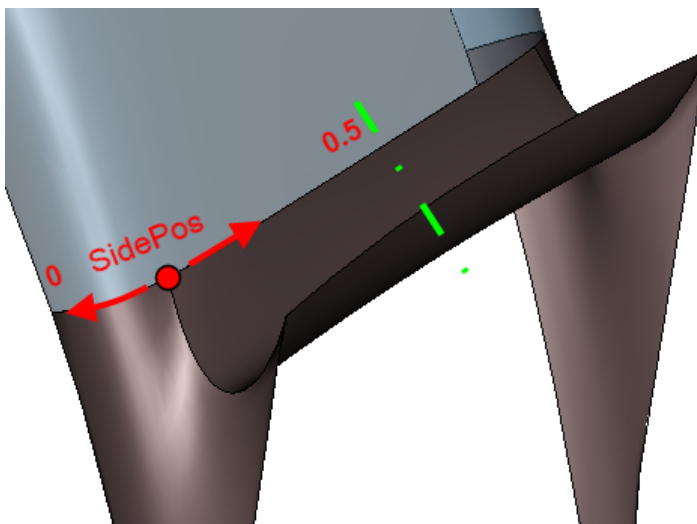
Side position left % Side position right %

☐ Rounded edges

The rounding is defined by the **angular position** $\varphi_{C,0}$ (0° =start of volute). Underneath, the minimum necessary angular position is displayed to prevent overlap of the actual volute and the diffuser.

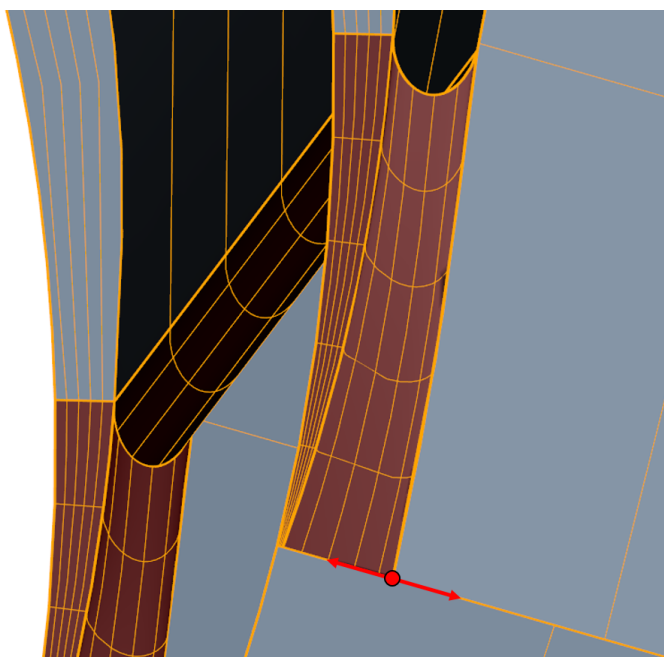


Additionally, the diffuser can be shifted in radial direction by the **radial offset** r_c to reduce the intersection of spiral and diffuser. This radial offset corresponds to the cut-water thickness.

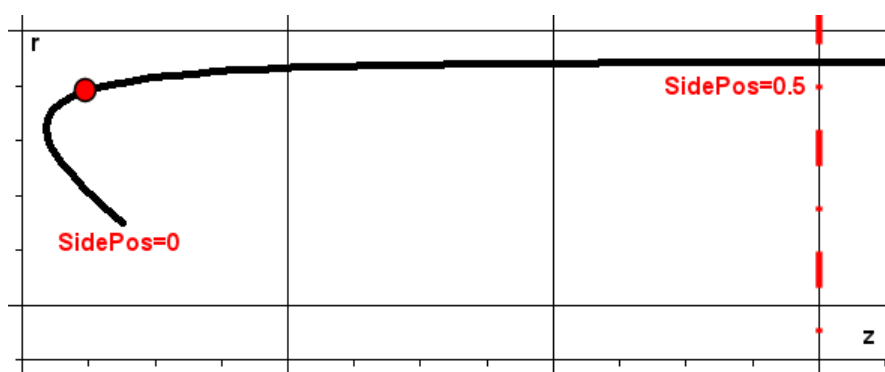


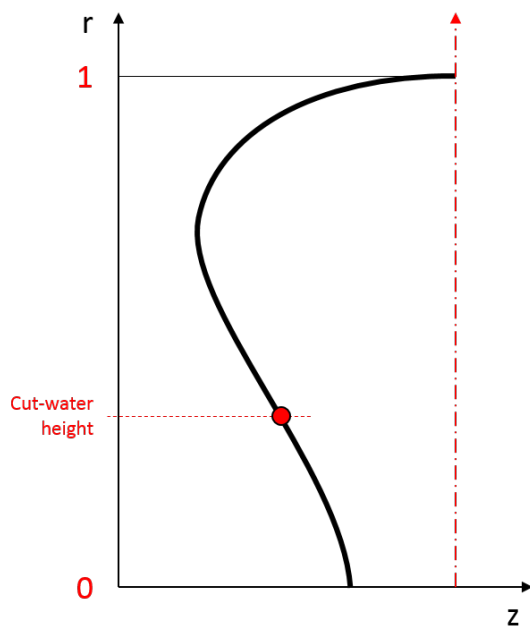
Side position defines the transition position from the central rounding surface to the side surfaces. For asymmetric spiral cross sections two independent values can be specified for left and right side.

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.

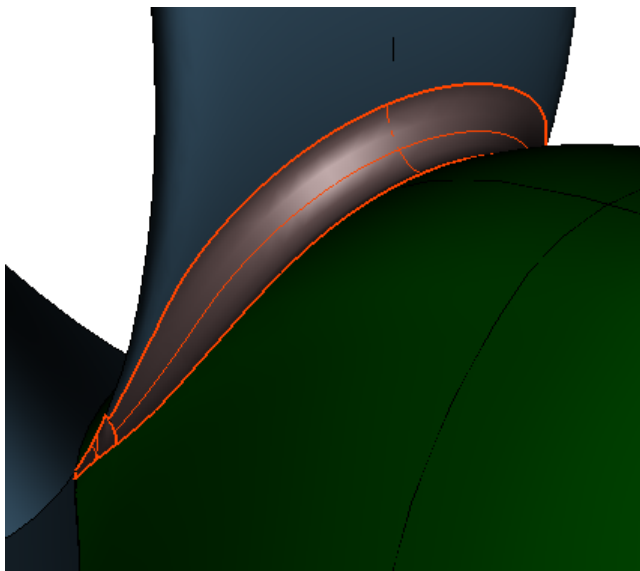




The cut-water itself is designed by a 4th order Bezier curve. The shape can be modified interactively after zooming in (**Zoom Cut-water**).

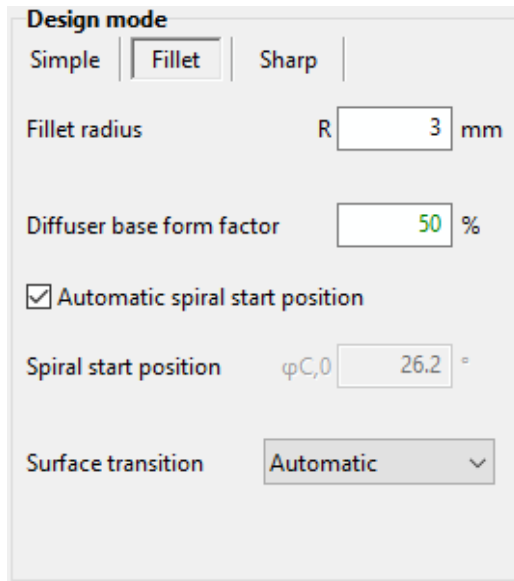
9.5.2 Fillet

For fillet cut-water design the spiral and the diffuser are trimmed and rounded at their intersection curve.



Prerequisites:

- The [wrap angle](#) ⁵³¹ must be high enough so that spiral and diffuser intersect.



Design mode

Simple | **Fillet** | Sharp

Fillet radius R mm

Diffuser base form factor %

☒ Automatic spiral start position

Spiral start position $\varphi_{C,0}$ °

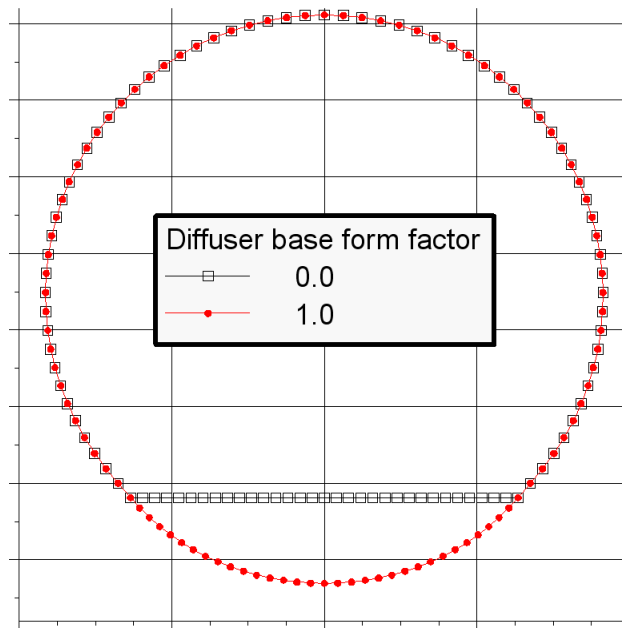
Surface transition **Automatic** ▾

The corresponding **fillet radius** can be specified.

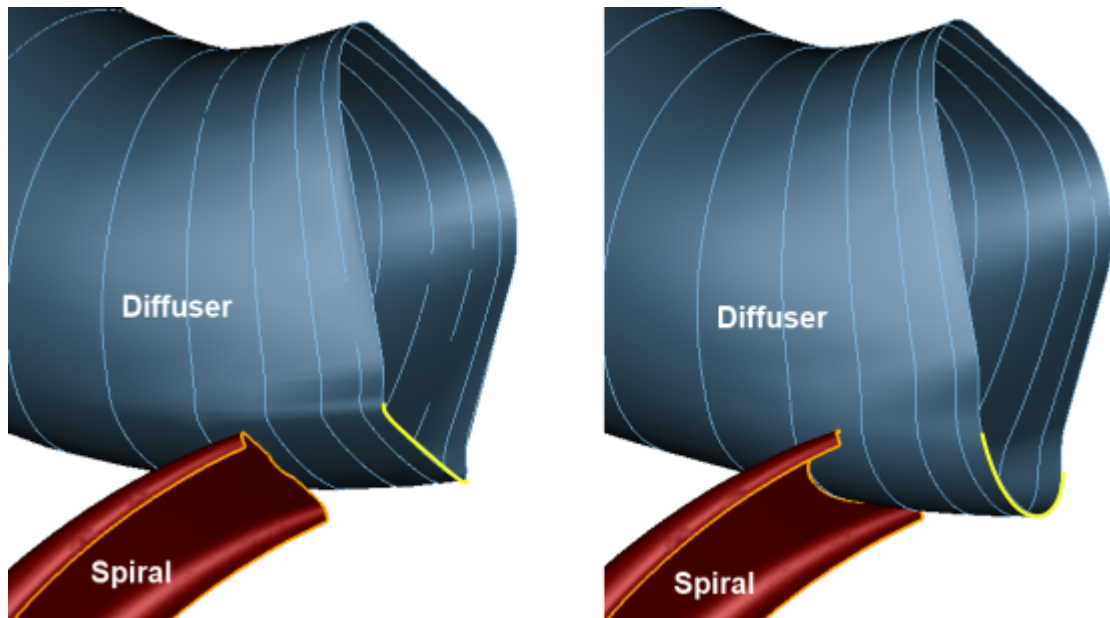
Additionally, the **Diffuser base form factor** defines the roundness of the first diffuser cross section on its base side and is between 0.2 and 1:

- 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.



Diffuser base form factor for a round spiral cross section



Compares diffuser base form factor of 0.2 and 1.0 for a spiral cross section of type line segments

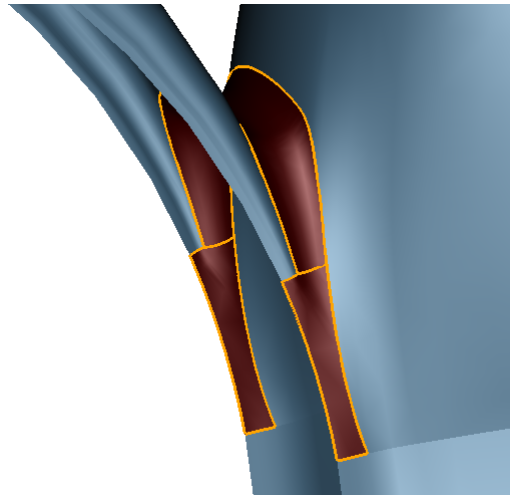
The **Spiral start position** indicates the angular position at which the spiral begins and influences the intersection of spiral and diffuser. It has to be at least 1° and must be lower than the intersection position of spiral and diffuser. If **Automatic** is activated the optimal angular position 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)

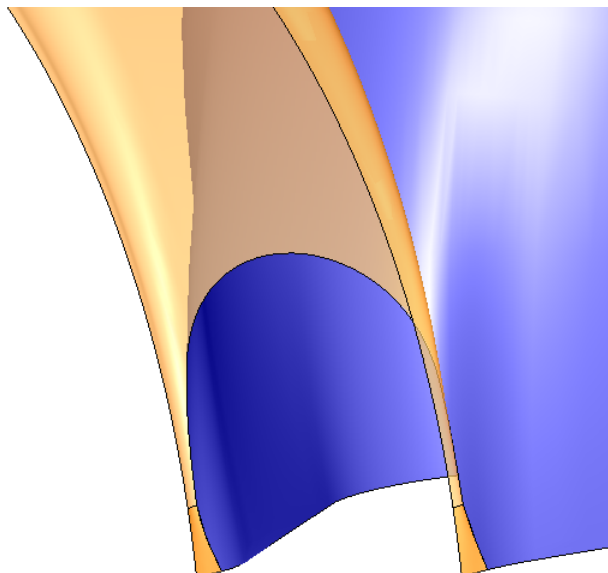
- Non-tangential: No tangential transition between both surfaces
- Automatic: Tries tangential transition. If it fails, a non-tangential transition is used. (Time-consuming)

If the fillet cut-water mode has been chosen, the **3D-model** is set to the [model state](#)^[237] "Solids only" after every update of the design because only then the spiral and diffuser surfaces that are trimmed according to the fillet are visible.



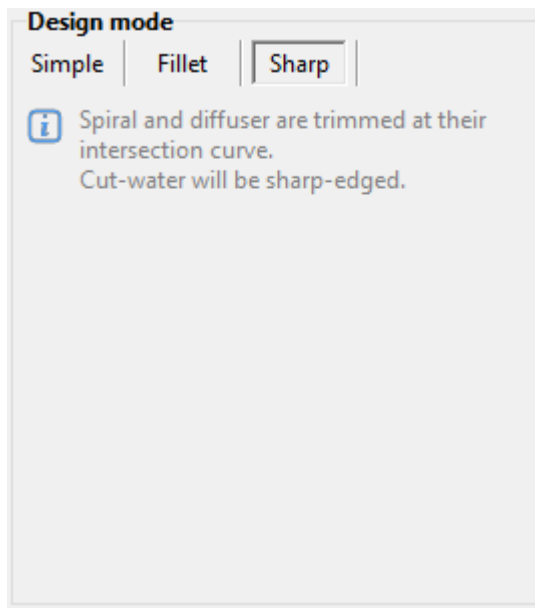
9.5.3 Sharp

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.



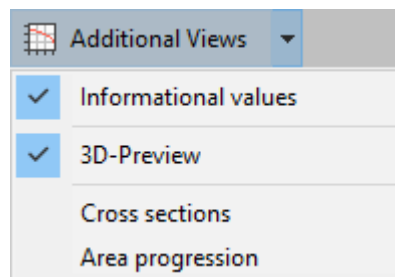
Prerequisites:

- The [wrap angle](#) ^{53.1} 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:



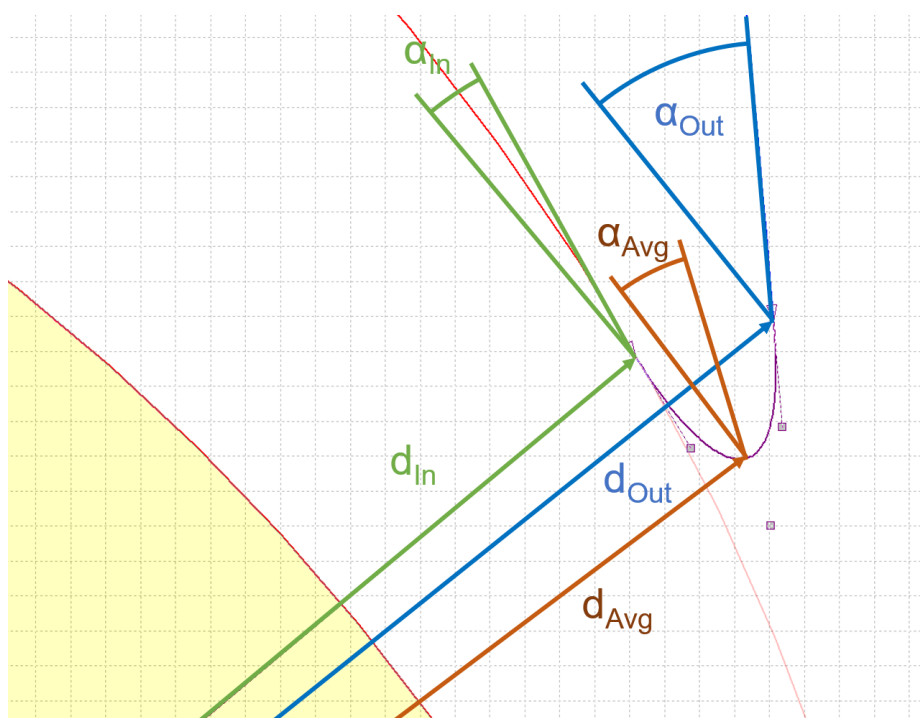
Informational values

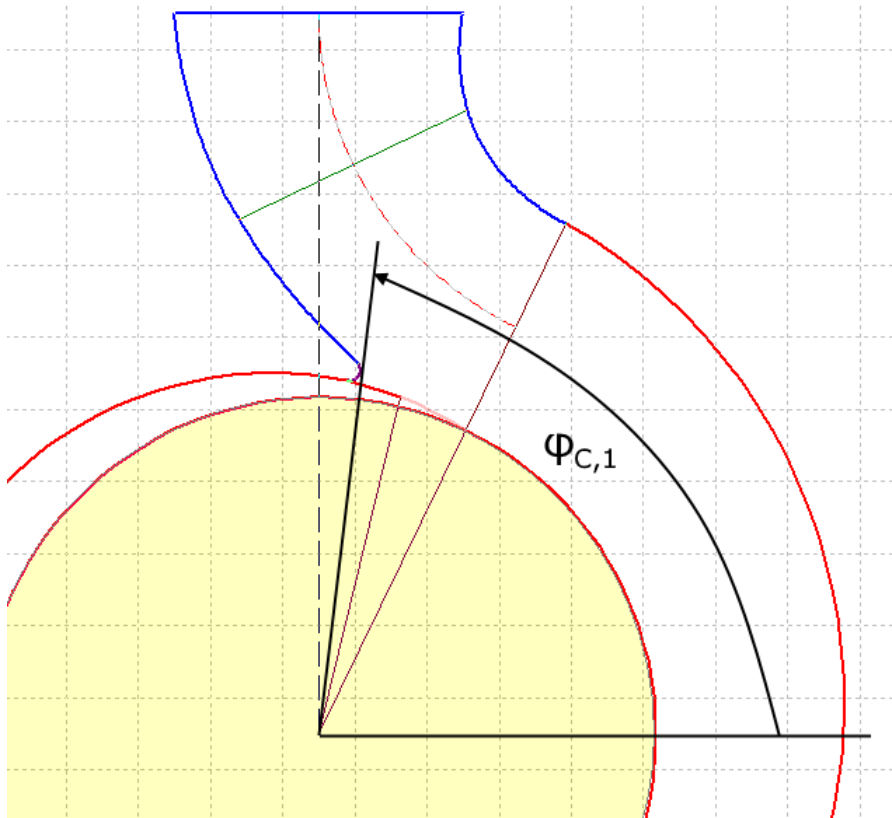
Some informative values relating to both **end cross-sections** and the **cut-water cross-section** are displayed:

Equivalent diameter	D
Cross-section area	A

Especially for cut-water, some diameters and angle are displayed:

Inner / Outer / Average cut-water angle	Inn / Out / Avg
Inner / Outer / Average cut-water diameter	dInn / dOut / dAvg
Minimal cut-water diameter	dMin
Cutwater angular position	c,1



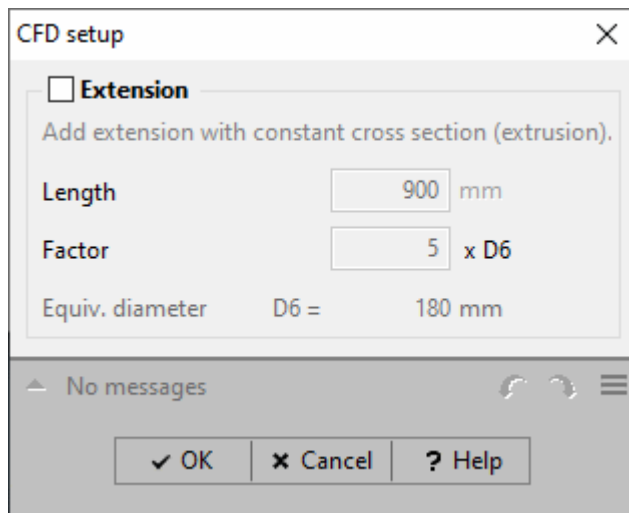


9.6 CFD setup

? VOLUTE | 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.



9.7 Model settings

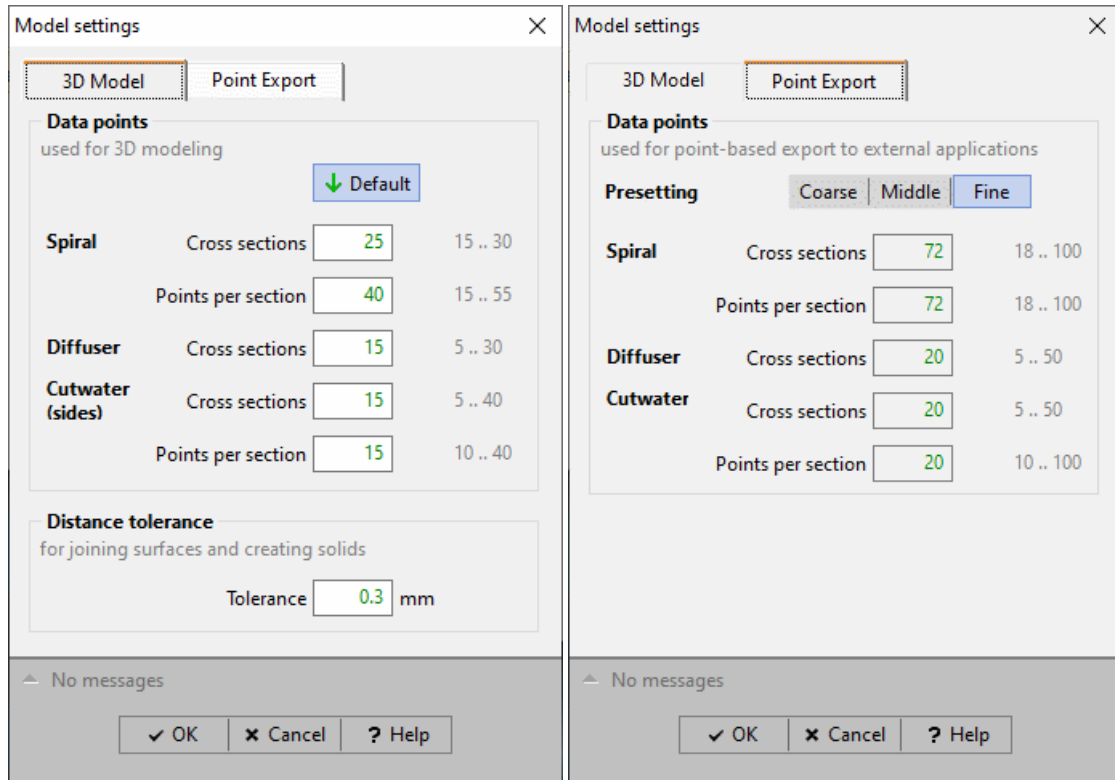
? VOLUTE | Model settings

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](#)⁵³⁷.



3D Model

Distance tolerance

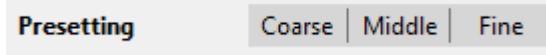
The distance tolerance defines the maximum allowed distance between sewed surfaces, e.g the faces of a solid.

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



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 Pfeleiderer, 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
Dissertation, TU Dresden, 1990

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
Hrsg. von Hans-Joachim Kleinert, Verlag Technik Berlin, 1989

VENTILATORS

Leonhard Bommers, 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

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
hrsg. von Hans-Joachim Kleinert, Verlag Technik Berlin, 1989

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
Proceedings of the ASME Turbo Expo 2011, ASME GT2011- 45502

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

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-Verlag, 1996

Redlich, O., Kwong, J.N.S.
On the Thermodynamics of Solutions. V. An Equation of State. Fugacities of Gaseous Solutions,
Chemical Reviews. 44, No. 1, pp. 233–244, 1949

Aungier, R.H.
A Fast, Accurate Real Gas Equation of State for Fluid Dynamic Analysis Applications,
Journal of Fluids Engineering, Vol. 117, pp. 277–281, 1995

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,
Industrial and Engineering Chemistry: Fundamentals, Vol. 15: pp. 59–64, 1976

Von Backström, Th. W.
A unified correlation for slip factor in centrifugal impellers
Journal of Turbomachinery, 2006; 128(1):1–10.

Mason, E.A., u. S. C. Saxena
Approximate formulae for the thermal conductivity of gas mixtures
Phys. Fluids 1 (1958), 361

10.2 Symbols

Symbol	Description
	Angle of absolute flow
	Angle of relative flow
	slip coefficient, Stagger angle
	Deviation angle flow / blade
r	Swirl number
	Drag ratio

Symbol	Description
	Efficiency
μ	Decreased power factor (sweeping)
	Diameter ratio
	Pressure ratio
	Density
	Thickness in circumf. direction; Speed coefficient
	Obstruction of flow channel by blades
	Wrap angle; Flow coefficient
	Work coefficient (= pressure and head coefficient)
	Angular velocity
α, β, γ	Sweep angles
A	Cross section area
b	Width
c	Absolute velocity
c_m	Meridional velocity ($c_m = w_m$)
c_u	Circumferential component of absolute velocity
c_L	Lift coefficient
c_D	Drag coefficient
d	Diameter
f	Camber
F	Force
h	Enthalpy
H	Pump head
i	Incidence angle

Symbol	Description
l	Chord length
L	Length
M	Torque; Meridional coordinate
m	Meridional coordinate (dimensionless)
\dot{m}	Mass flow
n	Number of revolutions
n_q, N_s	Specific speed
p	Pressure
P	Power
Q	Flow rate
r, R	Radius
s	Entropy, Orthogonal thickness
S	Static moment
t	Pitch; Tangential coordinate (dimensionless)
T	Temperature, Tangential coordinate
u	Circumferential velocity (Rotational speed)
v	Velocity
w	Relative velocity
w_u	Circumferential component of relative velocity ($w_u + c_u = u$)
Y	Specific energy
z	Geodetic height; Number of blades

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 expressly 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.

(4) 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 second 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 246, 286, 304

- 3 -

3D Model 225, 227, 234, 238

3D view 238

3D-model 181

3D-View 369

- A -

Acoustic benefit 475

Administrator 18

ALT 526

alternative speed 336

angle of flow 306, 308

ANSA 117

Ansys 153, 156, 162

ANSYS Workbench 176

approximate 349, 446

Approximation functions 198

Area circles 338

Area progression 338

Assumptions 249, 290, 308

asymmetric 520, 530

AUNGIER 400

Auto fit view 67

AutoCAD 114, 124

AutoGrid 117, 159

Automated component design 61

Automatic 44, 550

Automatic design 62

Automatic update 65

Axial extension 341, 345

Axial impeller 290

Axial position 513

- B -

Background 227

BACKSTROEM 402

Basic values 288, 306

Batch 32

Batch mode 176

Batch mode template 111

bend 530

Beta progression 405

Bezier 349, 405, 446, 447, 520, 525

Bezier curves 67

Bezier mode 338, 341, 345, 354

Bezier polynom 417

Blade 455, 473, 475

Blade angle 393, 395

Blade angles 371, 417

Blade blockage 393, 395

Blade lean angle 417

Blade lines 371

blade number 308

Blade properties 371

Blade root fillet 487

Blade shape 371

Blade thickness 371, 438

Blade thickness leading edge 198

BladeGen 82

Blades 234

Brumfield 272

- C -

CAD 9, 103

CAE 103

Calculate 256, 297, 313, 371

Calculation 44

camber angle 471

Casing 86, 485

Catia 114, 133

CFD 9, 103, 162, 562

CFT 78

CFturbo 9

CFturbo2ICEM 162

CFX-BladeGen 114

Characteristic numbers 249, 290, 308

Check 186, 216
 checksum 21
 Chord length 466, 470, 471, 473
 Circle 351, 520, 543
 Color 234
 company 18, 21
 Compare 181
 Compensation 531, 537
 Compressor 9
 Conformal mapping 405
 Constant 438
 Contact addresses 572
 continuity equation 256, 297, 313
 Contour 234, 353
 contra rotating 336
 convert 349, 446
 Coordinate system 227, 405
 Coordinates 67
 Copy 67
 copy to clipboard 21
 Coupled 341, 345
 Coupled linear 447
 Cross section 341, 345
 Cross sections 246, 286, 304, 520
 Curvature 338
 Cut-water 531, 537, 550
 Cut-water diameter ratio 198
 Cut-water width ratio 198

- D -

data points 486
 Deactivate 44
 Deceleration ratio 371
 Decreased output 371, 401
 Default 196
 Density 288
 Design point 86, 288, 306
 Design report 111
 Design rule 531, 534
 Deviation angle 371, 395
 Deviation flow - blade 371
 Diameter coefficient 198, 249, 290
 diameter ratio 249, 290, 308
 Diameter.cftdi 256, 337
 Dimensions 256, 297, 308, 313

Direction of rotation 513
 Display options 67
 distance 349, 446
 distance tolerance 238, 486
 DoE 176
 Double volute 540
 Double-click 186
 Download 216

- E -

Edge 455
 Edge position 455
 edit 71
 Efficiency 249, 290, 308
 Hydraulic 249
 Impeller 290
 Internal 249, 290
 mechanical 249, 290, 308
 Overall 249, 290
 Side friction 249
 Tip clearance 249
 total- 308
 total-to-total 308
 Volumetric 249, 290
 Ellipse 447
 emergency 52
 empirical 71
 End cross section 531, 543
 End shape 543
 Errors 238
 Euler's Equation of Turbomachinery 256, 297, 313
 Exact 386
 Exit diameter 338
 Exit width 338
 Expiration 186
 Export 32, 103, 162, 234
 Extend blade 487
 Extension 531, 543
 Extension on exit 338
 External 520
 external geometry 79

- F -

file 31

File location 198, 256, 297, 337
find 31
Finishing 184, 487
Flow angle 42, 288, 371
Flow angle inflow 198
Flow angle outflow 198
Flow angles 249, 290
Flow direction 42
Flow rate 288
Fluid 86
found 31
Freeform 417
Frontal view 405, 438, 447
Full impeller 75, 288, 306
Full volute 75
Function 71, 198
Functions.cftfu 198

- G -

General geometry 111
Global setup 86
Graphic 67
Grid 338

- H -

handling 67
Head 288
Help 215
Hub 337, 338, 341, 345
hub diameter 256, 297, 313, 337
Hydraulic efficiency 198

- I -

ICEM 162
ICEM-CFD 117
IGES 111, 225, 234
IGG 117
Impeller 9, 77, 513
Impeller diameter 256, 297
Impeller Options 196
Import 70, 79, 234
Incidence angle 371, 393
Inclination angle 341, 345

Inclination angle hub 198
Inclination angle shroud 198
Inclination angle trailing edge 198
Inducer 272
ineffective blade angle 431
Inflow 86
Inflow swirl 288
information 30
Initial design 67
Inlet 353, 485
Inlet definition 513
inlet diameter 313
Inlet triangle 393
inner 540
input 32, 67
Installation 14
Intake coefficient 198, 249
Interface 103
Interface definition 42
Interfaces 39
internal 520, 530
Inventor 114, 150
inverse design 432
Isentropic Mach number 412

- L -

Language 186
Leading edge 338, 341, 345, 354, 393, 447
Length unit for Export 563
License 21, 30, 31, 186
License agreement 572
License key 18
Licensing 9, 18
Line Segments 526
Line width 227
Linear 438
Linked 386
Load from impeller 513
local 21

- M -

machine ID 18, 21
Main dimensions 246, 247, 256, 286, 297, 304, 308
main window 59

Manual 550
 Manual dimensioning 247
 Material 234
 material density 335
 Material domain 487
 Max. curvature 341, 345
 Mean line 405
 Mechanical efficiency 198
 meridional velocity 356
 meridinal deceleration 308
 Meridional 405
 meridional boundaries 438, 447
 Meridional contour 338
 Meridional deceleration 198, 249, 290
 Meridional extension 417
 Meridional flow coefficient 272
 Middle of PS-SS 482
 Minimal relative velocity 249
 Mixed-flow impeller 290
 mixed-flow rotor 308, 313
 Model settings 563
 Model state 234
 Model-finishing 487
 Model-settings 486
 modules 21
 Mouse 225
 mulit stage 336
 multi-stage 77

- N -

NACA 473
 Navigation 61
 neck 530
 network 21
 New design 75
 NPSH 249
 number of blades 198, 499
 Number of revolutions 288
 Numeca 159
 NX 114

- O -

Obstruction 371
 Open 78

Optimal 371
 Optimimization 32
 Optimization 176
 Options 186
 Other 186
 outer 540
 Outflow coefficient 399, 400, 402
 Outlet 353, 485
 Outlet triangle 393, 395
 Outlet width 256, 297
 Outlet width ratio 249, 290
 Output 32

- P -

Parallel to z 341, 345, 354
 Parameter 32, 71, 176, 198, 247
 Parameters 103
 Parametric model 65
 Parasolid 111, 225, 227, 234
 permission 31
 permissions 31
 PFLEIDERER 371, 401, 531, 534
 Physical variable 198
 pitch 466, 470, 471
 point based export 486
 Points 198, 234
 Pointwise 117
 polyline 70, 349, 446
 Position 550
 potential flow 356
 Power loss 249, 290
 Power output 288
 Power partitioning 247
 Preferences 186, 196
 Pressure coefficient 249, 290
 Pressure difference 288
 Pressure side 393
 Primary side 42
 Print 67, 227
 prism_params 162
 Pro/ENGINEER 114, 134
 problem 31
 problems 31, 238, 482, 485
 Profile 438, 473
 Progression 70

Progressions diagrams 338
 Project information 85
 Project structure 39
 Project types 39
 Projection 485
 Pump 9

- R -

Radial 543
 Radial 2D 473
 Radial blade 389
 Radial blade fibre 389
 Radial blade section 389
 Radial diffusor 513
 Radial element blade 389, 417
 radial equilibrium 460, 463
 Radial impeller 290
 Radial rotor 308, 313
 Radius 351, 526
 Rake 405
 RDP 18
 recovery 52
 Rectangle 520, 525, 543
 Redo 61
 Reference 181
 References 566
 Register 18
 Remote 18
 Remove design steps 65
 request 21
 Required driving power 249, 290
 Resolution 227
 rights 31
 rotational speed 306
 rotor power 306
 Rotor-Stator-Interface 42
 RSI 485, 562
 RSI connection 482, 485
 RTZT 82
 Ruled surface blade 386

- S -

Save 67, 78, 227
 Secondary side 42

segment 482
 send E-mail 21
 server 31
 session code 21
 Settings 486
 Shaded 234
 shaft 337
 shaft diameter 256, 297, 337
 Shaft/ hub 313
 Sharp 550
 Shroud 338, 341, 345
 Shroud angle 249
 shroud diameter 313
 SI 190
 Side friction efficiency 198
 Simerics 117
 SimericsMP 117
 SimericsMP+ 117
 Simple 447
 Simple mode 338, 351
 Single blade 234
 Single passage 482
 single-flow 306
 Single-intake 288
 single-stage 288, 306
 Slip 371, 395
 Slip velocity 399, 400, 402
 Solid 234, 238, 360
 solid density 335
 Solids 238
 SOLIDWORKS 114
 sollidity 466, 470, 471
 SpaceClaim 114, 153, 156
 Specific energy 288
 specific speed 288, 306
 specific work 306
 Speed coefficient 288
 Spline 543
 Splitter 247, 386
 splitter blades 499
 Squirrel cage 247
 Stack 475
 stage 77
 Stagger angle 466, 470, 471, 473
 Stagnation point 393
 Standard specifications 256

Stanitz-Radius 421
 STAR-CCM+ 117
 Start 56, 103
 Start angle 543
 start date 21
 Static moment 338, 341, 345
 Status bar 67
 STEP 111, 225, 234
 Step by step 75
 STEPANOFF 531, 534
 Stepanoff constant 198
 STL 111, 225, 234
 Straight 341, 345, 354
 Straight line 351
 stream function 356
 Stress.cfst 256, 297, 337
 STRG 526
 Strictly external 520
 Suction diameter 256, 297
 Suction side 393
 Suction specific speed 249, 272
 Surfaces 238
 Sweep 475
 Sweep correction 466, 475
 Swirl 393
 swirl number 306
 Symbols 569
 Symmetric 520

- T -

Tangential 345, 353, 405, 543
 Test 198
 Thickness 438, 537
 Through - flow area 562
 tin 162
 tinXML 162
 Tip 485
 tip clearance 499
 Tip clearance efficiency 198
 Tip projection to casing 485
 torque 337
 torsional stress 337
 Trailing edge 341, 345, 354, 447
 Transmission of energy 395
 Transparency 234

Trapezoid 520, 525
 Trimming 487
 Turbine 9
 TurboGrid 117, 163
 Turbomachinery CFD 117
 Type number 288, 306

- U -

Undo 52, 61
 Uniform 447
 Units 190
 unshrouded 247, 485, 499
 unwinded length 447
 Update 215, 216
 Update warnings 61
 Updates 186
 US 190
 user 31
 User defined 438

- V -

values 67
 Velocity components 371
 Velocity triangle 313, 371, 393, 395
 Velocity triangles 256, 297
 Ventilator 9
 Version 78, 103
 View 227
 Visible 234
 Vista TF 117
 VNC 18
 Volumetric efficiency 198, 513
 Volute geometry 531

- W -

warning 438, 447
 Wastewater 247
 Website 9
 Width lines 338
 Width number 198
 WIESNER 371, 399
 Wireframe 234
 Work coefficient 198

Wrap angle 198, 417, 531

- Z -

Zoom 67, 227